

Section 1: Getting Started

Introducing flame:

projects,

preferences,

setups, hot keys,

interface,

monitor calibration,

and animation.

1

Introduction

Hello

*This chapter is the starting point for your adventures with **flame**®. It provides information about the **flame** documentation set, describes what's new in **flame** since the last release, and gives information on customer support.*

Summary

This chapter explains:

- “flame Documentation” on page 4
- “What’s New in flame 7.0” on page 6
- “Using This Guide” on page 15
- “Notation Conventions” on page 16
- “Getting More Help” on page 16

Welcome to flame

Welcome to **flame**—the image-processing system that brings the production capabilities of a high-end specialty house to your desktop. **flame** combines powerful image-processing software with the superior graphics processing power of Silicon Graphics™ workstations to deliver exceptional online digital editing, compositing, keying, colour correction, and 2D and 3D image-processing capabilities.

Images are loaded into **flame** from an external device such as a video tape recorder (VTR) and are stored in digital format in the **flame** framestore. The images are then displayed as clips on the desktop reels. The clips can be processed using any of the **flame** editing or processing commands or effects applications. The processed clips can then be recorded out to the external device.

flame Documentation

The documentation set for your **flame** system includes:

- **flame 7.0 User's Guide**
- **flame 7.0 Release Notes**
- **flame 7.0 Tutorial**
- **flame 7.0 Installation Guide**
- *Discreet Filesystem and Networking Guide*
- *Audio Hardware Configuration Guide*
- **flame 7.0 Troubleshooting Charts**

Documentation Available Online

The complete documentation set is available in PDF (Portable Document Format) for online viewing and printing. Use Adobe Acrobat Reader™ to view and print the PDF files.

You can access the PDF files from any of the following locations.

On your hard drive

The documentation PDF files are installed on the hard drive of your SGI machine when you install **flame**.

To access the PDF files installed on your hard drive:

1. Go to the product documentation directory. Type:


```
cd /usr/discreet/<product name>/documentation
```
2. View the list of available documentation for the product. Type:


```
ls
```

 and press **ENTER**.
3. To view one of the documentation PDF files using Adobe Acrobat Reader, type:


```
acroread <filename>
```

 and press **ENTER**. For example, to view the **flame** tutorial manual, type:


```
acroread flame70_tutorial.pdf
```

On Tutorial CD 1, "interactive tutorial & documentation"

- You can view and print the PDF files on CD1 using Adobe Acrobat Reader on a PC, an SGI, or a Macintosh.

To access the PDF files from an SGI:

1. Place CD1, “interactive tutorial & documentation” in the CD-ROM drive of your SGI machine.
2. Go to the *documentation* directory for **flame**. To view the documentation for **flame**, position the cursor over the UNIX shell and type:
`cd /CDROM/documentation/flame`
3. Perform steps 2 and 3 in the previous procedure to view the list of available documentation and to open the PDF files using Adobe Acrobat Reader.

On www.discreet.com

You can also find the PDF files on the Discreet web site, www.discreet.com. Go to the Support page, open the Documentation Library, and download any of the available files.

Using the PDF Documentation

The PDF files have several advantages over printed documentation:

- You can quickly search the entire document for any word in the text.
- You can click on cross-references, table of contents items, and page numbers of index entries to go directly to the related information. For example, click on the cross-references provided in the “What’s New in flame 7.0” section of this Introduction to quickly go to the sections in the user’s guide that describe the features new to this release.
- Bookmarks to the left of the document provide quick access to all the topics covered in the guide.
- Many illustrations are in colour.

Example Setups Available

Example setups are installed on the hard drive of your SGI machine when you install **flame**, for easy access as you work in **flame**. Currently, example setups for expressions and particles in Action are available.

These example setups are located in the **flame** examples directory,

`/usr/discreet/<product name>/examples`

To access the setups, load them directly into the module in which you are working. For instructions, see “To load an item from a directory:” on page 108.

What's New in flame 7.0

This release is one of the most extensive feature upgrade releases in the history of **flame**. The following is a list of the new features, sorted by module.

Action

Texture Projection

The new Surface Projector menu provides tools to project textures onto 3D models and 3D text to create an effect where a texture is animated on an object in 3D space. You can use various blend modes to blend the texture into the scene. See “Projecting Textures” on page 996.

Displaying Multiple Views

You can now display up to four views at a time in the image area, including a view of the Channel Editor. Multiple views are convenient for setting channel values, working in Schematic view, and previewing your result in perspective all at the same time. See “Displaying Multiple Views Simultaneously” on page 950.

Camera Improvements

- A node for the camera is now available in Action. See “Camera Node” on page 956.
- Free Camera (rotation on XYZ) is an option to Point of Interest. See “Camera Type box” on page 945.

Support for 3d studio max

In Action, you can import **3d studio max**® files; they contain object data, specifically, texture and materials. See “Importing 3D Models” on page 983.

Importing and Exporting Camera Data

You can now import and export camera data from and to **3d studio max**. See “Using the Camera” on page 945.

New Fog Effect

Using the Fog button, you can create a foggy atmosphere in the scene. You can also create visual effects such as mist, fog, haze, and murky water. See “Applying Fog to the Scene” on page 954.

Surface Blending Modes

New blending modes are available to specify how the front and matte clips or front and back clips are combined. See “Using Surface Blending Modes” on page 922.

3D Deformations

You can add a Deformation node, and use the Deformation menu to deform 3D models and 3D text or surfaces in the context of a composite. See “3D Deformations” on page 1011.

New Option for Animating Surfaces

You can now animate bilinear, bicubic, or extended bicubic surfaces using either Shape channel or individual vertex channels. See “Using the Shape Channel or Vertex Channels” on page 926.

New Rendering Options

Rendering options for sources are now available. See “Using Motion Blur with Source Nodes” on page 935.

Gaussian Blur has been improved. See “Using Gaussian Blur” on page 912.

New Auto Parent Node Manipulation

Automatic parenting is now available in the schematic. See “Auto Parent button” on page 899.

Multi-selection in the Layers List

For a group of selected layers, you can now add multiple objects such as surfaces, axes, textures, projectors. See “Using the Layers List” on page 909.

Archiving**Selective Restore from Compact Backup Sets**

You can now restore selected clips from a Compact Backup Set without having to restore the entire archive. See “Archiving Entries from a Clip Library” on page 364.

Estimating Archive Size

The new Tape Size Estimate button provides an estimate of the media space (in megabytes) needed to archive selected material. See “Estimating Archive Size” on page 355.

Archiving Without Audio

If you deselect Include Audio when archiving clips, the archived clips will not have audio tracks. In previous versions, the audio tracks were saved even though the restored clip had no audio. See “Saving to an Archive” on page 364.

Space Required Message

The message is now more detailed and says how much space you need to restore an archive on a framestore.

Audio

The AudioDesk is now integrated with the Player. See Chapter 24, “Using Audio.”

Audio Timestretch

Using the new Audio Timestretch feature, you can timewarp your audio. See “Audio Timewarps” on page 481.

New EQ controls

A new EQ panel provides an array of equalizing controls. See “Adjusting the EQ” on page 470.

Scrub in Modules

If you load a clip with audio into a module, such as Text or Action, you can scrub the audio by holding **CTRL** while you drag the positioner. See “Scrubbing in Modules” on page 458.

Audio Waveforms in Modules

You can now view the audio waveform in the timeline control area of some modules. See “To view a waveform in a module:” on page 458.

JL Cooper Controller

The JLCoooper MCS-3800™ external controller is supported with transport control and hot key mapping ability. See “JLCoooper MCS-3800 Interface” on page 102.

New Audio Spark

An audio VST spark is now available with the following plug-ins: reverb, modulation, dynamic compression, full parametric EQ. See “Audio Sparks” on page 483.

More Formats Supported

flame now supports the import and export of many more audio file formats. See “About Audio” on page 451.

ADAT (SGI Audio)

SGI audio can now be used with an ADAT for I/O. See “Input Source” on page 460 and “Outputs” on page 460.

Batch

New Nodes

Nodes for the following processes/modules are now available in Batch:

- GMask node
- Colour Warper node
- Degrain and Regrain nodes
- Deinterlace and Interlace nodes
- Export and Import nodes
- Field Merge node
- Film Compress and Film Expand nodes
- Library node
- LUT Editor node
- Optics node
- Reels node
- Modular Keyer node

For information on these new nodes, see Chapter 30, “Batch: Node Reference.”

Viewing Clips

You can now view the Front, Back, Matte, and Result clip for any node. See “Using Layer Views and Context View” on page 952.

New Queue Manager

With the Queue Manager, you can manage several output or export processes at a time. See “Using the Queue Manager” on page 586.

More Versatile Way to Add Clips to Processing Tree

Add clips by dragging Reel, Library and Import nodes to the schematic. This provides quicker access to material and allows for wire transfers from remote libraries. See “Adding Clips to the Schematic” on page 566.

Node Bypassing

It is not always necessary to render certain nodes to assess the final result of a composite. You can now bypass nodes to speed up interaction, by returning one of the input images (Front, Back, or Matte) instead of the Result. See “Bypassing Nodes” on page 578.

Saving a Text or HTML Log of Batch Status

You can now save Batch processing status information in a text or HTML file. See “Setting Batch Options” on page 589.

Rendering Batch Setups from a UNIX shell

You can now opt to launch **flame**, render a Batch setup, then exit **flame**, all in one command. See “Startup Options” on page 18.

LUT Editor Node

When you import or export Cineon, DPX, or any file with a different bit depth than the current framestore, Batch automatically appends a LUT Editor node to the Import or Export node. You can modify these default settings interactively using the LUT Editor, or import an existing LUT. See “LUT Editor Node” on page 605.

Scripting

Output and Export nodes now support shell scripts. The script is executed after the Output or Export render has finished. You can use this feature to, for example, automatically create QuickTime™ movies, or send an e-mail via the web. See “Using Scripts” on page 587.

New Tracer Tool for Garbage Masks

When you access the Garbage Mask menu from Batch, additional new tools are available — Region of Interest and the Tracer. See Chapter 36, “Garbage Masks and the Tracer.”

Node Caching

As a user preference, Batch can save a rendered version of a process (node), greatly increasing the speed of interaction, navigation between frames, and rendering. See “Working with Cache” on page 584.

Loading Setups and Nodes

In the Garbage Mask, Regrain and Degrain, and Colour Corrector nodes, you can go directly to the Load Setup menu by holding **ALT** when you drag a node to the schematic. See “Batch: Node Reference” on page 593.

Setups

You can now save part of a Batch setup and load and save nodes separately. See “Batch: Node Reference” on page 593.

More Versatile Output Nodes

You can now create setups with multiple output nodes. You can also change the priority of render for each of the output nodes.

You can now specify the destination for every output node in your setup, which can either be Reels or Library (local or remote). See “Output Node” on page 611.

New Logic Ops commands

Many new Logic Ops commands are available for combining the RGB channels of corresponding pixels from two source images. See “Using Logical Operations” on page 495.

New Lock Frame button

You can now lock a clip at a particular frame. See the *flame 7.0 Release Notes*.

Channel Editor

Expressions

You can now customize the animation of elements in a scene by applying functions such as noise or sin. Expressions are most useful in the Action module. See “Working with Expressions in the Channel Editor” on page 165.

New Animation Interpolation Mode

The Channel Editor includes a new interpolation mode called Natural. See “Understanding Interpolation Modes” on page 152.

New Channel List View

You can now view the channel hierarchy as a list of channels, which can make it easier to locate information. See “Animation Controls” on page 132.

Clip Libraries

Transferring Entire Clip Libraries

You can now load an entire clip library from another partition to the current partition. See “Transferring Clips Between Partitions” on page 214.

New Search Capability

You can use the Search module to find a clip on the desktop or in the clip libraries associated with projects. See “Searching for Clips” on page 46.

Colour Corrector

The Colour Corrector module has a new method for colour matching the front and back clips. See “Colour Sampling Using Match” on page 521.

Colour Warper

This new module provides intuitive colour manipulation tools for selective colour correction and scene colour matching. See Chapter 27, “Colour Warper.”

EDLs

You can now export EDLs. See “The Export EDL menu” on page 296.

Garbage Masks**Garbage Mask Node in Batch**

You can now access the Garbage Mask menu as a node in Batch. See “Using Garbage Masks from the Modular Keyer and Batch” on page 756.

New Tracer and Region of Interest Tools

When you access the Garbage Mask menu from Batch, two additional features are available, the Tracer and Region of Interest (ROI). See “Using Region of Interest (ROI)” on page 757 and “Pulling Keys with the Tracer” on page 758.

New Animation Methods

More options for animating garbage masks are now available. Mask vertices have their own x, y and z position channels, and tangent handles also have shape channels. This allows you to track or animate a garbage mask and maintain full control over shape interpolation. See “Rotoscoping a Mask” on page 750.

Hot Keys**J.L. Cooper**

The J.L. Cooper outboard controller is now supported. See “JLCooper MCS-3800 Interface” on page 102.

New Clone Tool

You can now map multiple keystroke sequences to a single button, field, or function using the Clone tool. See “Clone” on page 101.

Modular Keyer

This module provides a set of customizable keying and compositing tools. This is a pipeline-based, batch-oriented set of tools including a processing pipeline, an advanced edge-detection tool, 3D Keyer (with an RGB cube that shows you the colours in the key-in clip mapped in three-dimensional RGB colour space), and new garbage mask options, such as the Tracer.

See Chapter 34, “The Modular Keyer,” Chapter 35, “The 3D Keyer,” and Chapter 36, “Garbage Masks and the Tracer.”

Monitor Calibration

New Calibration Method

Calibration is now done using X-Rite device (or Barco CaliTalk). This takes into account the brightness and contrast settings of the monitor, which ensures more accurate measurement of colour levels.

Filmlook LUT

Visualize what the projected film will look like directly from the **flame** monitor. The calibration process is now separate from visualization.

Preferences

On-screen Keyboard

You can now turn off the on-screen keyboard, so that when you enter a name in a field, the keyboard does not appear. See “Setting the Keyboard On or Off” on page 119.

Bold Font

You can now change the font of the button text to Bold by using the Bold Font preference. See “Changing UI Settings” on page 118.

Pointer Threshold

A new button is available to test the pressure of the stylus. See “Threshold Test button” on page 125.

Jitter Filter button

The new Intuos tablets are very sensitive to magnetic fields. The Jitter Filter option provides a method of counteracting the effect. See “Jitter Filter” on page 125.

Processing Menu

Logic Ops

Many new Logic Ops commands are available for combining the RGB channels of corresponding pixels from two source images. See “Using Logical Operations” on page 495.

Schematic View/Processing Tree

In Action's Schematic view and the processing tree in Batch, the following improvements have been made to make it easier to connect nodes.

Parenting in Move mode

You no longer have to switch to Parent mode when connecting nodes in your processing tree. When creating a link, starting the link from a spot near the edge of the node or image allows you to create the link in Move mode. See “Connecting Clips and Nodes” on page 568.

Drag & Drop Parenting

In Move mode, you can now drag a node and lightly “touch” the tab of another node to create a link between the two nodes. See “Connecting Clips and Nodes” on page 568.

Removing Links between Nodes

You can now cut multiple links with one “stroke” of the cursor.

New Sparks

SparkMotionKey (Front/Back/Matte)

This spark records motion on the tablet and creates X and Y channels in the Channel Editor. You can opt to process the motion.

SparkDisplaySound

This spark pastes animated waveforms for sound amplitude directly on the clip. This can be useful to quickly time effects.

Text Module

Text Roll Improvements

The Text Roll and Crawl menu includes new buttons to make it easier to create text rolls: Fit Best Speed and Lock Leading & Scroll. See “Creating Text Rolls and Text Crawls” on page 868.

Grouping Text Layers

You can now group several text layers using the Group button. Grouped layers can be saved and loaded as one unit. See “Grouping Text Layers” on page 868.

Animating Text

The Animation menu in the Text module now includes new tools for animating paragraphs and characters. See “Animating Paragraph Channels” on page 872.

Importing Logos

You can load images from the desktop to create logos. They can be saved as *.tif* images files. See “Loading Logos” on page 864.

Tab Stops

You can now use tab stops to lay out your text. See “Tabulating Text” on page 865.

Spell Checking

You can now use a spell checker on your text. See “Spell Checking” on page 866.

TrueType and Asian Fonts

TrueType and Asian fonts are now supported. See “Working with Fonts” on page 876.

User Interface/Desktop

Grid and Guides

New, fully customizable grid and guide controls are now available in all modules. See “Grid and Guides Menu” on page 66.

New Zoom method

In addition to the + and - incremental zoom buttons, you can now zoom in and out by any amount using the Zoom field. See “The Image Window Display Controls” on page 66.

Play to Air

You can now send all clips on a desktop reel directly to the broadcast monitor. This is useful for organizing a play list on a reel and sending all your clips to the Player so that they are all immediately available for broadcast. See “Play to Air” on page 80.

Reference Buffer

Several modules now contain a buffer in which you can place a clip that you can refer to as you work. You can view both the reference clip and the current clip at the same time in the image window by placing a “split bar” over the image window. See “The Reference Buffer” on page 81.

Multiple Undo

You can now undo multiple operations from the desktop.

Real-Time Audio Playback

You now get real-time audio playback in modules.

Numeric Fields Reset

Numeric fields can now be reset to the default value by **CTRL**-clicking them.

Locking Reels

You can lock clips on one or more reels so that particular frames within the clips are lined up with each other on the desktop. When you scroll through any of the locked reels, they all scroll together. See “Locking Reels Together” on page 44.

Goto Option

The Goto option allows you to go to a specified timecode as well as to a specified frame number. See “The “Goto” Button and Hot Key” on page 41.

Using This Guide

The *flame 7.0 User's Guide* begins with a section on basics, followed by sections on the six principal areas of an online production suite: media management, editing, image-processing and formatting, creating effects, compositing, and retouching images.

Getting Started

This section includes this introduction, plus information on some fundamental concepts: starting **flame**, creating projects, using the interface, calibrating your monitor, saving setups, and so on.

Managing Clips

This section describes how to manage visual and audio media. It includes chapters on inputting and outputting clips, using EDLs, using clip libraries, archiving, and so on. For a detailed description of this section, see Chapter 11, “Overview: The Library Menu.”

Editing

This section describes how to edit and assemble clips. It also describes how to create soft edits using splices, dissolves, and timewarps. For a detailed description of the Editing section, see Chapter 19, “Overview: The Editing Menu.”

Transforming Clips

This section describes how to use basic processing commands and batch processing commands, and also details how to use the Colour Warper, Colour Corrector, and Filters modules to adjust images. For a detailed description, see Chapter 25, “The Processing Menu,” and Chapter 31, “Formatting Clips.”

Applying Effects

This section describes how to create effects using the following modules: Keyer, Modular Keyer, Stabilizer, Text, Compositor, Optics, and Quick Composite. For a detailed description, see Chapter 32, “Overview: The Effects Menu.”

Power Compositing

This section describes how to produce complex motion effects and optical effects in three dimensions in the Action module. Action is also used for multilayering clips and creating composites using a common back clip. For a detailed description, see Chapter 42, “Action: Overview and Setup Options.”

Retouching Clips

This section describes how to use the Paint module to create images from scratch and retouch existing images. For a detailed description, see Chapter 48, “Paint: Overview.”

Notation Conventions

A number of style conventions are used throughout this user’s guide. These conventions and examples of their use are shown below.

Convention:	Example:
Text that you enter into a UNIX shell appears in Courier bold.	flame -p
Variable names appear in Courier, enclosed in angle brackets.	<file_name>
Feedback from the IRIX system appears in Courier.	limit coredumpsize
Directory and file names, and UNIX utility names appear in italics.	<i>/usr/discreet</i>

NOTE: When referring to a pathname, version 7.0 is used (for example, **cd *usr/discreet/flame 7.0***). If you are running a different version, replace “7.0” with your version number of **flame**.

Getting More Help

If you need more information, contact Discreet Customer Support at one of the following numbers. You can also send queries by e-mail.

Discreet Customer Support

North America:	1-800-925-6442
International:	514-954-7199 (Country code = 1)
E-mail:	discreet.support@autodesk.com
WWW:	www.discreet.com

Starting and Exiting **flame**

Getting in... getting out...

*Starting **flame** is an easy process. However, there are several startup options you can use to specify particular startup parameters or to start **flame** under unusual circumstances. Similarly, exiting is easy, although there are a few tricks you can use if your system freezes.*

Summary

In this chapter, you learn about:

- “Starting **flame**” on page 17
- “Exiting **flame**” on page 20
- “Using the “Kill Process” Command” on page 20

Starting **flame**

If you are running **flame** for the first time, make sure you have correctly installed and initialized the software. For more information, see the **flame** *Installation Guide* for this release.

To start **flame**:

1. Log in to the **flame** user account at your workstation.
2. Open a command window (IRIX shell).

NOTE: When you log in to your workstation, a “Console” shell opens. Do not start **flame** in the console; open a new shell and start it there.

3. Use the mouse to position the cursor in the IRIX shell, and type **flame** at the command prompt.

After a few moments, the Project Management menu appears.

If you are using Discreet Audio hardware, you should type “flames” instead. In the following sections, substitute the command “flames” for “flame” if you are using Discreet Audio hardware.

4. Select the project and user you want to use or create a new one. For more information, see “Project Management” on page 23.
5. Click the Start button.
The software has finished loading when you see the yellow cross cursor, and the message Startup Complete in one of the message bars. The Main menu appears in the lower left corner of the screen.
6. Select a Main menu option.
The desktop and reels are displayed.

Startup Options

There are a number of startup options that you can use with the **flame** command under special circumstances.

NOTE: These startup options are case-sensitive. You can see an on-screen list of these startup options by entering

flame -H. This command does not launch the software.

Type:	To:
flame	Start flame using the default configuration file.

Discreet Audio-Related Startup Options

flame -a	Delete all audio files on the audio disk whether or not referenced by the current framestore volume.
flame -V	Delete all audio files on the audio disk that are referenced by the current framestore volume.
flame -I	Delete all audio files on the audio disk that are referenced by the current framestore partition.
flame -S	Skip the audio disk integrity check that normally occurs at startup.

Framestore-Related Startup Options

flame -v	Initialize the framestore volume (all partitions).
flame -i	Initialize the framestore partition selected by the current project.
flame -r	Initialize the desktop only, for the framestore partition selected by the current project.
flame -R <time>	Define the frame purge timeout (in seconds). This option is only applicable if you are using the Discreet Filesystem running on a Discreet Storage disk array.

Type:	To:
-------	-----

Project Management-Related Startup Options

- flame** Use a configuration file other than the default, where
-c <filename.> <filename.cfg> is the name of the file. The specified file supersedes
 cfg> the default configuration file. If you do not use the option, **flame** looks
 first for `init.cfg`, and if that is not found, it looks for `flame.cfg`.
- flame -J** Define the project that appears in the Project Management menu when
 <project> you start **flame** where <project> is the name of the project you want
 to use. If you start **flame** with both the **-J** and **-U** options, **flame**
 bypasses the Project Management menu.
- flame -U** Define the user that appears in the Project Management menu when you
 <user> start **flame** where <user> is the name of the user you want to use. If you
 start **flame** with both the **-J** and **-U** options, **flame** bypasses the
 Project Management menu.

Miscellaneous Startup Options

- flame -m** Initialize the mouse only (not the tablet and stylus).
- flame -F** Force **flame** to install new fonts that you added to the directory `/usr/
 lib/DPS/outline/base` (and `/usr/lib/DPS/AFM` if you have also
 installed the corresponding font metrics file). For more information, see
 Chapter 40, "Text."
- flame -B** Prevent the broadcast monitor from blanking when an image viewer or
 Player is not displayed. The broadcast monitor then displays whatever
 screen content is displayed in the corresponding region.
- flame -L** Disable writing to the `.log` file. **flame** keeps a log file of your actions in a
 text file (`.log`) in your **flame** home directory. Usually this file is cleared and
 recreated every time **flame** is started. (Note that the log file is very useful
 when reporting problems to Discreet Technical Support.)
- flame -l** Append the current session to the existing log file. (See description of the
 log file above.)
- flame -p** <n> On multi-processor machines, start **flame** using less than the full array of
 processors, where <n> is the number of processors to be used.
 Normally, **flame** uses all available processors.
- flame -f** Use a custom menu file, where <filename> is the name of the menu
 <filename> file. For information on custom menus, see "Saving and Removing a
 Custom Menu Setup" on page 126.

Type:	To:
flame -J	Launch flame and specify a Batch setup to render.
<project> -U	flame launches, renders the setup, and exits.
<user> -b	
<batch_setup>	NOTE: To avoid corrupting the render, which uses the graphics hardware, you should not use the graphics console of the system (such as UNIX shells) while the render is taking place.

NOTE: For information on starting **flame** with the Discreet Audio subsystem, see “Starting flame with Audio” on page 452.

Exiting flame

Click the Exit command in the System menu to exit the current work session and save the status of the desktop and current menu preferences. Alternatively, press **ALT+F12** from any menu and confirm. If you want to exit without confirming, press **SHIFT+INSERT**.

Using the “Kill Process” Command

If for some reason you are unable to continue in the current work session (for example, your **flame** system freezes) but the system still responds to keyboard and mouse commands, you can use the “Kill Process” command to end all **flame** processes that are still running. You need to do this before restarting **flame**.

To use the “Kill Process” command:

1. Press **ALT+F3** to display the IRIX desktop.
2. Open an IRIX shell, if one is not currently available.
3. At the command line in the IRIX shell, type:

```
ps -fu <flame account name>
```

This command displays information about all active processes. The display should be similar to the following:

PID	TTY	TIME	COMD
2420	ttyq4	0:00	csh
2519	ttyq4	0:00	flame_IRIX...
2526	ttyq4	0:00	ps

4. Determine the process number (PID) for **flame**. In this example, the process number is 2519.

- At the command line, type:

```
kill <process number>
```

where <process number> is the process number you determined in step 4. This command terminates the **flame** process that is currently executing.

NOTE: There may be more than one **flame** process running at any time. For example, there may be one process per CPU, plus some additional processes to manage the framestore. You should kill each of these processes.

You can now restart **flame**.

Using the Kill Alias

The `.cshrc` file that comes with your **flame** system contains the alias “kf” which can be used to kill all **flame** processes at once. If you are using Discreet Audio hardware, the alias is called “kfs”. As an alternative to the procedure outlined in the preceding section, you can type the alias “kf” (or “kfs”) in an IRIX shell to kill all **flame** processes. You should then use the command:

```
ps -fu <flame account name>
```

as outlined above to verify that all **flame** processes have indeed been killed. Never restart **flame** if there are still **flame** processes left over from a previous session.



WARNING: The “kf” and “kfs”(kill flame) aliases should not be used to exit **flame** unless the application is hung or otherwise not responding. Using “kf” or “kfs” indiscriminately can cause system problems. Under normal conditions, you should use the normal exit procedures described in “Exiting flame” on page 20.

[illegible]

Project Management

Where's your stuff?

Project Management lets you keep all your material organized. Every project has its own clips and setups. You can even customize and save your user preferences.

Summary

In this chapter, you learn about:

- “Starting a Work Session” on page 23
- “Creating a Project” on page 25
- “Creating a User” on page 27
- “Saving Space on the UNIX File System” on page 28
- “Modifying Projects and Users” on page 29
- “Deleting Projects, Setups, or Clips” on page 30
- “Deleting Users” on page 31
- “Loading Setups from Other Projects” on page 32
- “Loading Preferences from Other Users” on page 32
- “Switching to a Different User” on page 33

Starting a Work Session

When you start **flame**, the Project Management menu prompts you to select a Project and a User for the current session. You can load an existing Project or User or create new ones. By default, the Project and User from the previous **flame** session appear in the Project Management menu.

NOTE: You can also choose to work in the default Project and User, which allows you to work with all your setups and preferences in the **flame** home directory. You cannot delete the default Project and User. Keep in mind that if all your jobs use the default Project and User, they will all share the same setups, clips, and preferences.

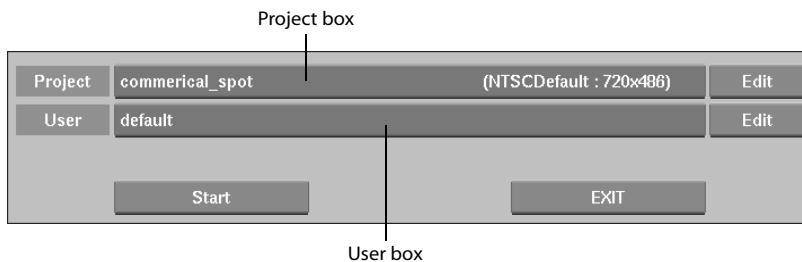
You can only work in one project at a time. To switch to another project, you must exit and restart **flame** with a different project.

You can create projects in any resolution available in the Setup Framestore menu. When you create a project, you specify the partition to use, and **flame** automatically uses the partition's resolution for that project.

To start flame:

1. Log in to the **flame** user account at your workstation. After a few moments, a command UNIX shell appears on the screen.
2. Use the mouse to position the cursor in the IRIX shell and type **flame** at the command line. (For other startup options, see “Startup Options” on page 18.)

The Project Management menu appears.



3. Specify a project:
 - To load an existing project, select it from the Project box.
 - To create a new project, select <new> from the Project box. See “Creating a Project” on page 25 for more information.
4. Specify a user:
 - To choose an existing user, select it from the User box.
 - To create a new user, select <new> from the User box. See “Creating a User” on page 27 for more information.
5. Click Start.

NOTE: From the Project Management menu, you can also modify or delete Projects or Users. For more information, see “Modifying Projects and Users” on page 29, “Deleting Projects, Setups, or Clips” on page 30, and “Deleting Users” on page 31.

Creating a Project

Projects you create are associated with setups for each module. Each group of setups is stored in a different directory on the file system. A setup file contains data such as text styles, 3D geometries, and keyframe information. When you create a project, you specify which setups to use. You can create new setups, or you can copy or share an existing project's setups.

Projects are also associated with clips. Clips are stored on the partition associated with the project. When you create a project, you specify the partition to use. You can share an existing project's clips by selecting an existing partition, or you can create a new partition for the project.

You cannot copy clips from another project when you create a project (as you can with setups), but you can load clips and setups from other projects during a work session. For more information, see “Loading Setups from Other Projects” on page 32.

When you copy setups from another project, you can modify them later without affecting the source. However, when you share setups or clips, modifying them from one project affects all shared projects.

To create a project:

1. Select <new> from the Project box in the Project Management menu.

The Create Project menu appears.

EXIT Project		Reset	
Name	NTSC		
Description			
Partition	NTSCDefault	(720x486)	Shared with: commerical_spot
Setup Dir.	[usr/discreet/project/editing/NTSC]		New Setups
Create Project	Cfg Template	ntsc.cfg	Memory 30

2. Enter a name for the new project in the Name field.

NOTE: Names can be up to 60 characters long.

3. Enter a description of the project in the Description field.

4. Specify the partition on which you want to store the project's clips:

- To specify a new partition for the project, select <new> from the Partition box. The Framestore Setup menu appears. Create the new partition and click Apply. The partition is created and you are returned to the Create Project menu.
- To share an existing project's clips with the new project, select an existing partition from the Partition box.

NOTE: Click the Reset button at any time to clear all fields in the Create Project menu.

5. Specify the project's setups:

- To share an existing project's setups with the new project, select an existing setup directory in the Setup Directory box.
- To create a new directory for the setups, use the suggested setup directory or select *<new>* from the Setup Directory box.
- To copy setups from an existing project, select Copy From from the Setup Directory box. An additional box appears, from which you select the name of the existing project.

When you create or copy setups, the setups directory is set to `/usr/discreet/project/effects/<project name>`, where `<project name>` is the name of your project. To specify a different directory for your setup files, select *<new>* in the Setup Directory box and create or select a different directory using the file browser.

Also, when you create or copy setups, all the factory setups (such as filters) are automatically copied to the new project. To prevent factory setups from being copied for each new project, you can have **flame** create UNIX file-system links from the new project to the default project. For more information, see “Saving Space on the UNIX File System” on page 28.

6. In the Configuration Template box, specify the file to use as the template for your project configuration file. By default, if you specified an NTSC or PAL partition, the correct template file appears in the Configuration Template box. If you specified a resolution other than NTSC or PAL, **flame** searches for a configuration file corresponding to the resolution of the partition. For example, if you are using a partition with a resolution of 960 x 576, **flame** searches for a configuration file called:

960x576.cfg

in the following directory:

~/cfg

where `~` is the home directory for **flame**.

NOTE: If the configuration file does not exist, you must create it and put it in the correct directory.

7. Change the memory value if necessary. The value in the Memory box is the same as the value in the Memory token of the configuration file. For more information, refer to the **flame 7.0 Installation Guide**.
8. Click Create Project.

The project is created, and you are returned to the Project Management menu.

NOTE: Click Exit Project to return to the Project Management menu without saving your changes.

Sharing Material with Other Projects

If you use the same material for more than one project, you can share clip libraries between projects. When you share partitions, each project opens the same partition so that you can access the same material.

The easiest way to share material is when you create a project. When you create a project, specify an existing partition to access all clip libraries on that partition. For information on creating a new project, see “Creating a Project” on page 25.

If the projects already exist and they do not share the same partition, use **wire**® (formerly Discreet Network) to transfer the clips from the remote partition and then save them in the local partition. For more information on using **wire**, see “Using the Network Menu” on page 226.

Creating a User

Creating a user allows you to save your preferences and load them in subsequent **flame** sessions. You can set preferences in most modules, such as Paint, Text and Action, as well as in the Hot Key Editor (hot key settings). The desktop preferences, which you access by clicking Preferences on the **flame** menu, are also saved with the user.

When you create a user, you specify which preferences you want to use: you can create your own preferences, or copy or share an existing user’s preferences. If you copy preferences, you can modify them later without affecting the source. However, if you share preferences, modifying them from one user affects all shared users.

NOTE: If you want to use another user’s preferences for only some modules, see “Loading Preferences from Other Users” on page 32.

To create a user:

1. Select <new> from the User box in the Project Management menu.

The Create User menu appears.

EXIT User	
Name	night_editor Reset
Pref. Dir.	[usr/discreet/user/editing/night_editor] New Prefs
Create User	

2. Enter a user name in the Name field.

NOTE: Names can be up to 60 characters long.

3. Specify the preferences directory:

- To create your own preferences, use the suggested preference directory or select *<new>* from the Preferences Directory box. Use the file browser to create a new directory.
- To copy an existing user's preferences, select Copy From from the Preferences Directory box. When you select Copy From, an additional field appears from which you select the existing user.
- To share an existing user's preferences, select an existing directory from the Preference Directory box.

When you create or copy preferences, the preferences directory is set to `/usr/discreet/user/effects/<user name>`, where *<user name>* is your user name. To change the default preferences directory, select *<new>* from the Preferences Directory box and use the file browser to create or select a new directory.

When you create or copy preferences, all the factory preferences are automatically copied to the new user. Alternatively, to save space on the UNIX file system, you can have **flame** create IRIX file-system links from the new user to the default user. For more information, see "Saving Space on the UNIX File System" on page 28.

4. Click Create User.

The user is created, and you are returned to the Project Management menu.

Saving Space on the UNIX File System

If you copy or create setups when you create a project, the default setups are automatically copied to the new project. Similarly, if you copy or create preferences when you create a user, the default preferences are automatically copied to the new user. Copying the default setups or preferences can take up a lot of space on the UNIX file system. To save space, you can change the project environment so that **flame** creates UNIX file system links to the default project or default user.

To create UNIX file system links to the default project or default user:

1. Before starting **flame**, open an IRIX shell.
2. Type the following at the command line:
setenv FLAME_PRJLINKDEFAULTS 1
3. In the same IRIX shell, load **flame** by typing **flame** at the command line.
4. Create a project and choose the copy or create setups option, or create a user and choose the copy or create preferences option. For more information, see "Creating a Project" on page 25 and "Creating a User" on page 27 respectively.

When you change the project environment, the `FLAME_PRJLINKDEFAULTS` environment variable is only set for the current **flame** session. To permanently set the

FLAME_PRJLINKDEFAULTS environment variable, add the following line to the `.cshrc` file located in the **flame** home directory:

```
setenv FLAME_PRJLINKDEFAULTS 1
```

Archiving a Project That Contains Linked Setups

If you archive a project that contains linked setups, the links to the setup files are restored when you restore the archive. As such, you must ensure that the files specified in the links still exist when you restore the archive.

Modifying Projects and Users

You can modify the name and description of an existing project. You can also modify the name and preferences directory of an existing user.

To modify a project's name and description:

1. Select a project from the Project box in the Project Management menu, and then click Edit. The Modify Project menu appears.

EXIT Project				
Modify Project	Name	commercial_spot	Reset	
	Description			
	Partition	NTSCDefault	(720x486)	Shared with CDdemo
	Setup Dir.	/usr/discreet/project/editing/commercial_spot		
Modify Project	Ctg File	commercial_spot.cfg	Memory	30
			Not shared	

2. Select Modify Project from the Edit box.
3. To modify the project's name, enter a new name in the Name field.
4. To modify the project's description, enter a new description in the Description box.
5. To modify the Memory token, enter a new value in the Memory field.
6. Click the Modify Project button.

The project is modified and you are returned to the Project Management menu.

To modify a user’s name and preferences directory:

1. Select a user from the User box in the Project Management menu or the Preference menu, and then click Edit.

The Modify User menu appears.

EXIT User

Modify User

Name

night_editor

Reset

Pref. Dir.

/usr/discreet/user/editing/night_editor

Not shared

Modify User

2. Select Modify User from the Edit box.
3. To modify the user name, enter a new name in the Name field.
4. To modify the preferences directory, select <new> in the Preferences Directory box and use the file browser to select a new directory. If the preferences are shared with one or more users, modifying the preferences directory affects all shared users.

NOTE: When preferences are shared, the Shared With label appears beside the Preference Directory box.

5. Click the Modify User button.

The user is modified and you are returned to the Project Management menu.

Deleting Projects, Setups, or Clips



Projects are associated with setups and clips. You can delete the entire project, its setups, or clips.

NOTE: To delete only some of the clips associated with a project, use the Remove button in the Library menu instead of the following procedure.

To delete a project, setups, or clips:

1. Select a project from the Project box in the Project Management menu, and then click Edit.
2. Select what you want to delete from the Edit box:

Select:	To:
Delete Project	Delete a project.
	If you delete a project, the setups and clips are also deleted from the file system and the framestore only if they are not shared by other projects.

Select:	To:
Delete Setups	Delete all the setups, but keep the project and the clips. The project reverts to the default setups.  WARNING: This also deletes the setups for all projects that share the same setups after you confirm the deletion.
Delete Clips	Delete all the clips, but keep the project and the setups.  WARNING: This also deletes the clips for all projects that share the same partition after you confirm the deletion.

NOTE: Clips or setups are shared if a Shared With label appears beside the Partition or Setup Directory box.

- 3. Click Confirm.
 - 4. You are returned to the Project Management menu.
- NOTE:** You cannot delete the default Project or its setups.

Deleting Users

You can delete a user if you no longer want to use the associated preferences.

To delete a user:

- 1. Select a user from the User box in the Project Management menu or the Preferences menu, and then click Edit.
The Modify User menu appears.

EXIT User			
Modify User	Name	night_editor	Reset
	Pref. Dir.	/usr/discreet/user/editing/night_editor	Not shared
Modify User			

- 2. Select Delete User from the Edit box.
If you delete a user that has shared preferences, the preferences directory is not deleted from the file system; the preferences directory is deleted only if no other user shares the preferences.
NOTE: Preferences are shared if a Shared With label appears beside the Preference Directory box.
- 3. Click the Delete User button.

4. Click Confirm.

You are returned to the Project Management menu.

NOTE: You cannot delete the default user.

Loading Setups from Other Projects

During a work session, you can load setups from other projects into the current project. You can then modify them and save them to the current project or any other project.

NOTE: When creating a project, you can copy all the setups from another project into the new project. You can also share clips and/or setups between projects. See “Creating a Project” on page 25.

Loading a setup from another project does not automatically save it in the current project. You must save the setup explicitly if you want to have the file in the current project, such as when archiving a project.

To load a setup from another project:

1. Click Load in the appropriate module.

The Load menu appears.

2. Use the Project box to select the project that contains the setup you want to load.

The available setups are listed.

NOTE: To revert to the current project, click Reset.

3. Select the setup you want to load.

The selected setup is loaded into the current project.

Loading Preferences from Other Users

You can set preferences in most modules, such as Paint, Text and Action, as well as in the Hot Key Editor (hot key settings). You can also load another user’s preferences for one or more of these modules. You can then modify the preferences and save them to the current user or any other user.

NOTE: When you create a user, you can copy or share another user’s preferences. Keep in mind that you must copy or share all the preferences. See “Creating a User” on page 27.

Current preferences are saved automatically when you exit or switch to a different user. If you want to save them under a different name in order to store several sets of preferences, you must explicitly save them under a different name.

To load preferences from another user:

1. Click Load in the appropriate module.
The file browser appears.
2. Select preferences from the Load Option box.
3. Use the User box to select the user that contains the preferences you want to load.
The available preferences are listed.
4. Select the preferences you want to load.

Switching to a Different User

During the current work session, you can load a different user if you want to use a different set of preferences. The current project and user appear in the Preferences menu. You cannot switch to a different project from the Preferences menu. To use a different project, you must exit and restart **flame**.

To switch to a different user:

1. Click Preferences.
The Preferences menu appears.
2. Select the new user from the User box.

NOTE: Select <new> if you want to create a new User.

The new User preferences are loaded to the current work session.

NOTE: Current preferences are saved automatically when you exit or when you switch to a different user.

This image shows a full page of white paper designed for handwriting practice. It features 18 evenly spaced, horizontal dashed lines that run across the entire width of the page. The lines are thin and light gray, providing a guide for letter height and placement without being distracting. There is no text or other markings on the page.

*Mastering the tools and elements of the interface is the first step to literacy in **flame**.*

Summary

In this chapter, you learn about:

- “The Primary Interface” on page 35
- “The Image Window” on page 64
- “The Desktop” on page 38
- “The Player” on page 77
- “The Menu Area” on page 49
- “System-Wide Hot Keys” on page 83
- “The File Browser” on page 61

The Primary Interface

The primary interface is the interface through which you access all the features of **flame**.

To access the primary interface:

1. Start **flame**.

The Project Management menu appears.

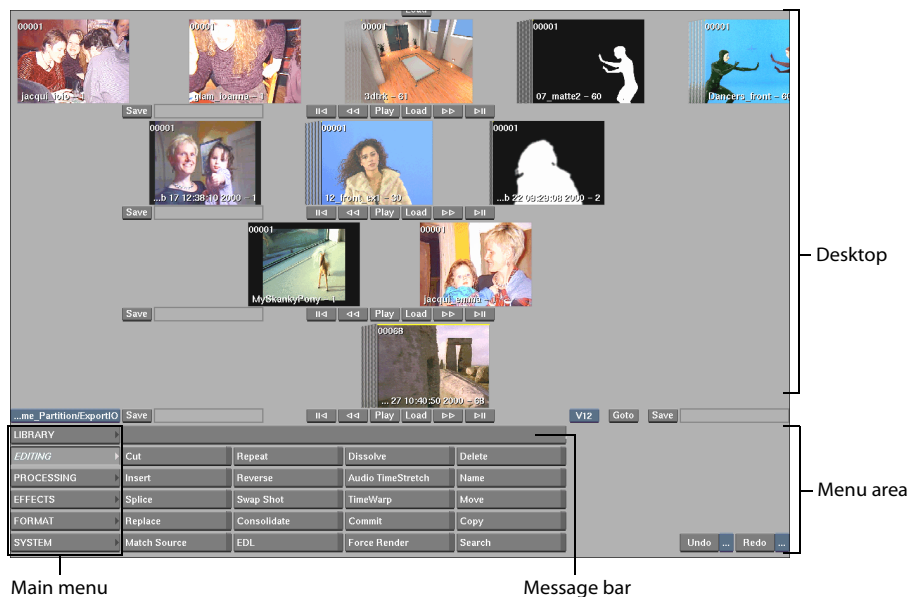
2. Select a project and a user and click Start. For more information, see Chapter 3, “Project Management.”

The Main menu appears.



As **flame** initiates, status information appears in the status bar to the right of the Processing button. When the startup sequence completes, the message “Startup Complete” appears in the status bar, and the wait cursor changes to a yellow cross.

3. Click any Main menu button.
The primary interface appears.



The primary interface is divided into two regions: the desktop and the menu area. The desktop provides an area where you can place, view and work on clips. The buttons in the menu area provide commands you can apply to clips, as well as access to other areas, or modules, in **flame**.

The Cursor

The shape and colour of the cursor change as you perform a series of actions. This section describes the various cursors you use in the primary interface.

The cursors common to all modules are the yellow cross cursor and the red, green, blue, and white arrow cursors. Consult the appropriate module or interface description for explanations of other cursors you encounter in **flame**.



Use the yellow cross cursor to click menu buttons, select options in option boxes, and enter values in fields.



Use the arrow cursor to select frames and clips for editing and processing. You can also perform intermediate actions, such as entering numeric values or selecting options in option boxes while the arrow cursor is active.

Use:	To:
Red arrow	Select the first clip (the front, or foreground clip).
Green arrow	Select the second clip (the back, or background clip).
Blue arrow	Select the matte clip.
White arrow	Select the destination reel for a clip.

Moving the Cursor Using the Stylus

To move the cursor on the screen, move the stylus across the tablet.

NOTE: If for some reason you cannot use the tablet and stylus, you can use the mouse to move the cursor. For information on initializing the mouse at startup time, see “Startup Options” on page 18. To switch to the mouse while running **flame**, press Shift-M-Insert. To switch to the tablet and stylus, press Shift-T-Insert.

The four basic cursor actions you use in a work session are: pointing, clicking, pressing, and dragging:

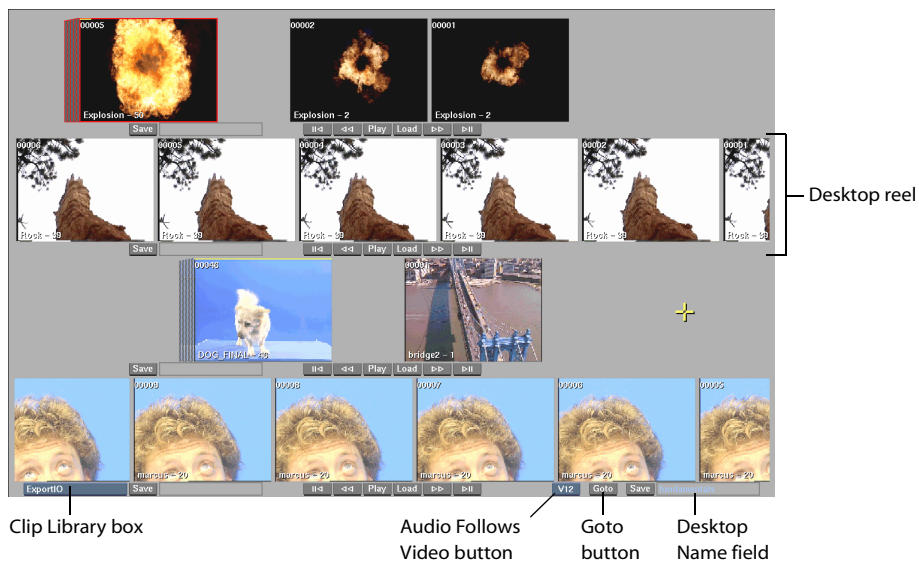
- To point to an object or menu button on the screen, position the cursor over the object or menu button.
- To select a frame or other object, or to enable a menu button, click on it. To click using the stylus, point to the item, push down on the stylus and then release.
- To select an option in an option box, position the cursor over the option box, push down on the stylus and hold. When the list of options appears, drag the stylus to the option you want, and release. See “Option Boxes” on page 54 for information on option boxes.
- To drag the cursor, push down on the stylus as you move it across the tablet.

The Desktop

The desktop is where you manage the clips associated with the project. You can also perform certain operations on clips on the desktop.

The desktop contains a number of desktop reels. When you start **flame** for the first time, create a new project, or load a project that has no clips associated with it, the desktop reels are empty. For information on how to load clips onto the desktop reels, see Chapter 12, “Clip Libraries.”

You can configure the number of reels that appear on the desktop, as well as the orientation (vertical or horizontal) of the reels. See “Changing the Number of Reels on the Desktop” on page 123, and “Changing the Orientation and Direction of Reels” on page 123.



See “Cutting Audio Gesturally” on page 400 for a description of the Audio Follows Video button.

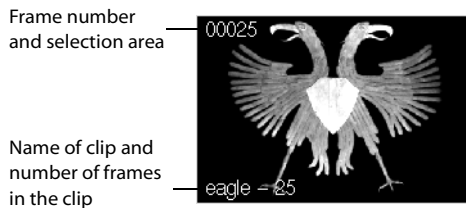
Desktop Reels, Clips, and Frames

When you bring a clip into **flame**, you add it to a desktop reel. A desktop reel can contain a sequence of one or more clips.

A clip is a sequence of one or more frames. A frame is a single image. Only the size of the framestore partition limits the number of frames in a clip. You can load clips from a framestore, import clips from the UNIX file system, or input clips from an external device such as a VTR.

Frame Numbers and Timecodes, Names and Durations

The frames in a clip are numbered sequentially using either frame numbers or timecodes. The frame number or timecode appears in the upper left corner of each frame. By default, the first frame in a clip has a frame number of 00001 or timecode of 00:00:00:00. You can change the timecode of a frame using the Change Timecode command in the Format menu (see “Changing Timecode” on page 617).



The name of the clip and the number of frames in the clip appear in the lower left of each frame. You can change the name of a clip using the Name command in any of the six menus available from the Main menu.

You can choose to display either timecodes or frame numbers. You can also control the display of the clip name and frame duration. See “Displaying Timecodes, Frame Numbers, and Names” on page 120.

Selecting Clips and Frames

You select a clip by clicking on the timecode or frame number of any frame in the clip. To select a particular frame, click the timecode or frame number of that frame.

Collapsing Clips

You can make more clips visible on the desktop reels at the same time by collapsing clips. In a collapsed clip, only one frame is visible; the other frames are stacked underneath. The order of the frames in the clip does not change.



To collapse a clip:

1. Position the cursor over any frame in the clip.
2. Press the **C** key on the keyboard.

The clip collapses.

To collapse all clips on a reel:

1. Position the cursor in an area between clips on a reel.
2. Press the **C** key on the keyboard.

All clips on the reel collapse.

To collapse all clips on the desktop:

1. Position the cursor in the menu area.
2. Press the **C** key on the keyboard.

All clips on the desktop collapse.

NOTE: The collapse feature works as a toggle. Thus, to uncollapse a collapsed clip, reel, or desktop, repeat the applicable procedure.

Playing and Scrubbing through a Clip

To play a clip on a desktop reel, position the cursor over the clip and press the **ENTER** key.

You can also scrub through a clip by dragging left or right on the upper left corner of a frame. A yellow timeline indicator along the top edge of the frame indicates the relative position in the clip of the currently displayed frame.

Yellow timeline indicator



When you scrub through an uncollapsed clip and then release the cursor at a particular frame, the clip repositions to place that frame under the cursor.

The “Goto” Button and Hot Key

To go to a specific frame in a clip, use the Goto button in the lower right of the desktop, or the Goto hot key.

To go to a specific frame in a clip using the Goto button:

1. Click Goto.

The cursor changes to a purple arrow, and the Goto menu appears in the area below the Goto button.

2. Use the Goto menu to set the parameters for the command.

Use:

Absolute/
Relative box

Source Timecode/
Frame box

Timecode/Frame field

To:

Set whether the frame or timecode you enter in the Timecode/Frame field is relative to first frame in the clip, or relative to the frame you click on when you perform the “go to” command.

Indicate whether you want to go to a timecode or a frame number.

Enter the timecode or frame number.

3. Position the cursor over the clip containing the frame you want to go to.

4. Click on the clip.

The frame you specified appears under the cursor. If you selected Relative in the Absolute/Relative box, click again to go to the next frame relative to the current frame. For example, if you entered 8 frames in the Timecode/Frame field, each subsequent click advances the clip 8 frames.

5. Click anywhere in the menu area *except* on a menu item, to end the “go to” operation.

The cursor changes to the yellow cross cursor.

To go to a specific frame in a clip using the Goto hot key:

1. Place the cursor over the clip containing the frame.

2. Press the G key on your keyboard.

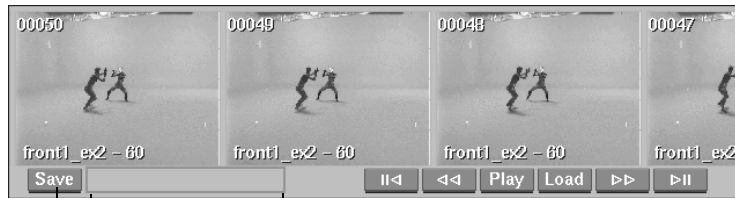
The numeric keypad appears.

3. Enter the number of the frame that you want to find.

The clip advances to the specified frame number.

Naming and Saving a Reel

You can give each reel a different name, and save it in a clip library.



Save Reel
button

Reel Name field

To name a reel:

1. Click the Reel Name field for the reel you want to identify.
2. Type the name in the keyboard that appears, and then click Enter.
The name appears in the Reel Name field.

To save a reel:

1. Click Save Reel on the desktop reel you want to save.
2. Type the name of the reel in the keyboard that appears, and then click Enter.
The reel is saved in the current clip library.

NOTE: If you click the Save Reel button, and the reel you are saving does not have a name, **flame** prompts you for a name for the reel. For more information on saving reels, see Chapter 12, "Clip Libraries."

Naming and Saving the Desktop

You can name a desktop and save it in a clip library. When you save a desktop, all clips on all reels are saved.

To name a desktop:

1. Click the Desktop Name field.
The keyboard appears.
2. Type a name and click Enter.
The name appears in the Desktop Name field.

To save a desktop:

1. In the Clip Library box, select the clip library to which you want to save the desktop.
2. Click the Save button next to the Desktop Name field.
The keyboard appears.

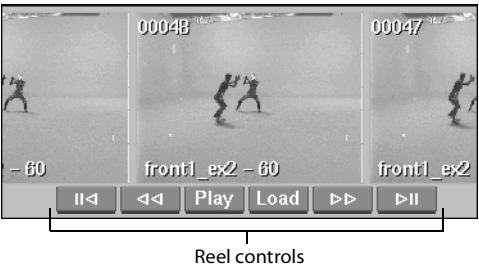
3. Type a name and click Enter.
flame saves the desktop in the clip library.

Moving the Clips on a Reel

You can move the clips on a reel using the reel controls, or the reel scrolling area.

The Reel Controls

The reel controls appear at the bottom centre of each desktop reel. Use these controls to scroll through the clips on the reel. If a reel does not contain any clips, only the Load button appears.



In the following table, “current clip” refers to the clip that is positioned over the reel controls.

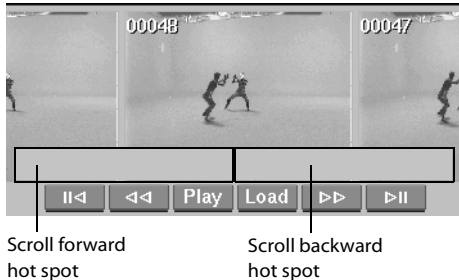
Click:	To:
	Go to the last frame of the current clip on the desktop reel. Clicking a second time advances to the first frame of the next clip. See Note.
	Scroll forward through the clips on the reel. See Note.
	Open the Player with the current clip. In the Player you can play a clip, play a clip with audio, or modify a soft edit. See “The Player” on page 77. See also “Playing and Scrubbing through a Clip” on page 40.
	Load a clip from the clip library. This is similar to clicking Load in the Library menu and selecting the destination reel. The difference is that the Load reel control button is available whenever the desktop is visible.
	Scroll backward through the clips on the reel. See Note.
	Go to the first frame of the current clip on the desktop reel. Clicking a second time advances to the last frame of the next clip. See Note.

NOTE: These descriptions assume the default reel direction of “Right --> Left”. For more information, see “Changing the Orientation and Direction of Reels” on page 123.

Scrolling through a Desktop Reel

You can scroll through a desktop reel by clicking anywhere on the reel *except* on a frame number or timecode, and dragging left or right.

You can also scroll through a reel by holding the cursor down on one of the scroll hot spots, located above the reel controls. To increase the speed of the backward scroll, drag the cursor to the right of the scroll backward hot spot. To increase the speed of the forward scroll, drag the cursor to the left of the scroll forward hot spot.



NOTE: These hot spots assume the default reel direction of “Right --> Left”. If the reel direction is reversed, the hot spots are reversed. For more information, see “Changing the Orientation and Direction of Reels” on page 123.

Locking Reels Together

You can lock reels together and then navigate the reels simultaneously. This is useful for viewing different versions of a clip. You can select a specific frame on a clip on each of the reels that you lock together — **flame** will align the clips based on the frames you select. The frames you select are referred to as “alignment frames.”

A light blue border appears around all the frames in a locked clip. **flame** automatically expands all collapsed clips on a locked reel.

To lock two or more reels together:

- For each reel you want to lock, position the cursor over the alignment frame and press the **L** key. **flame** centres the alignment frame on its reel and locks the reel to the other locked reels. If you decide on a different alignment frame, after you press the **L** key, simply press it again over the new alignment frame.

To unlock a single reel, or all reels:

- To unlock a single reel, position the cursor between clips on the reel and press the **L** key.
- To unlock all reels, position the cursor anywhere in the menu area and press the **L** key.

NOTE: Either of the following automatically unlocks all reels:

- Selecting a menu command.
- Dropping a clip gesturally onto a locked reel.

Scrolling Through Locked Clips

The reel controls work as follows on locked clips:

Click:

To:



Go to the next end of clip among all locked reels. For example, if the next end of clip occurs on reel 3, **flame** aligns all locked reels to that end of clip on reel 3.

ALT+



Go to the next transition on the locked reel containing the reel controls you clicked.



Scroll forward through the locked reels.



Scroll backward through the locked reels.



Go to the next start of clip among all locked reels. For example, if the next start of clip occurs on reel 2, **flame** aligns all locked reels to that start of clip on reel 2.

ALT+



Go to the previous transition on the locked reel containing the reel controls you clicked.

The scroll forward hot spots and the scroll backward hot spots also work on locked clips.

You can also use the left and right arrow keys to scroll locked reels to the left or right by a specified number of frames. See “Step Offset” on page 123 for details.

Scrubbing Through Locked Clips

You can scrub through the aligned clips on the locked reels simultaneously. Click and drag on the timecode of any frame in any of the locked clips. The yellow timeline indicator appears at the top edge of each aligned frame, and changes as you scrub through the clips.

NOTE: None of the following are available for locked reels:

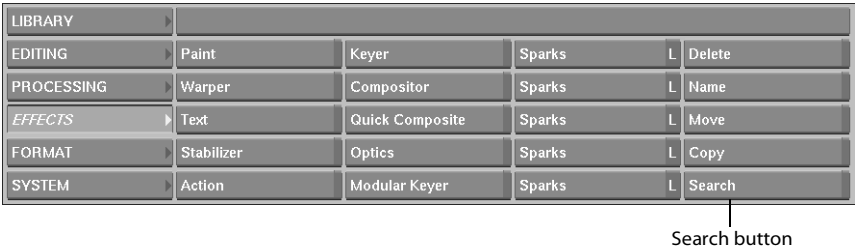
- Scrubbing through a clip on only one of the reels.
- Collapsing a clip or a reel.
- Using the Goto button or hot key.
- Using gestural editing on the reel.

Searching for Clips

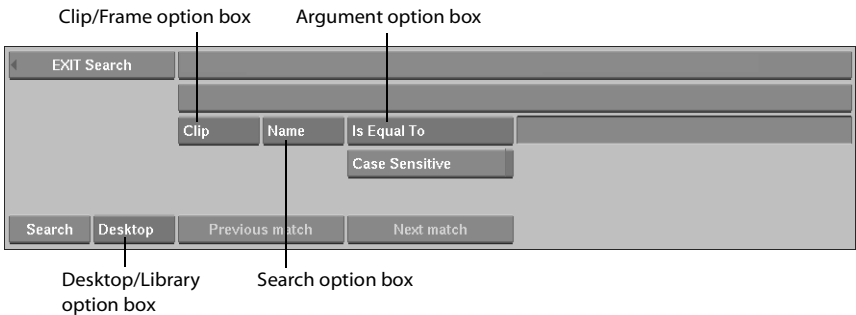
You can use the Search module to find a clip on the desktop or in the clip libraries associated with the project.

To search for a clip:

1. Click any button in the Main menu to display a menu containing the Search button.



2. Click Search.
The Search menu appears.



3. Select either a desktop search or a clip library search using the Desktop/Library option box. In a desktop search, **flame** searches the current desktop reels. In a clip library search, **flame** searches the clip libraries associated with the project. For more information on clip libraries, see Chapter 12, “Clip Libraries.”
4. Select a search option from the Search option box, and use the field and/or option box that appears to supply the necessary arguments for that option.

Option:	Specify:	Possible Arguments:
Name	A clip name	Contains; Does Not Contain; Matches Pattern
SrcTC	A source timecode	Is Greater Than; Is Less Than; Equals; Does Not Equal
RecTC	A record timecode	Is Greater Than; Is Less Than; Equals; Does Not Equal
Date	A date	Is Earlier Than; Is Later Than; Equals; Does Not Equal

Option:	Specify:	Possible Arguments:
Duration	A clip duration	Is Greater Than; Is Less Than; Equals; Does Not Equal
Audio	n/a	Contains; Does Not Contain
SoftEdit	n/a	Is A; Is Not A
Archive Flag	n/a	Is Set; Is Not Set
EDL Reel	A reel name or number	Is Equal To; Is Not Equal To; Contains; Does Not Contain; Is Less Than; Is Greater Than; Matches Pattern
Comment	A comment	Is Equal To; Is Not Equal To; Contains; Does Not Contain; Is Less Than; Is Greater Than; Matches Pattern

- Click Search in the lower left of the Search menu to perform the search.

In a desktop search, if **flame** finds a clip that matches the search criteria, it centres the clip on its reel. The message bar identifies the reel on which it is located. Use the Next Match button to locate each subsequent match.

In a clip library search, if **flame** finds a clip, the cursor changes to a white arrow. You then select a destination reel on which to load the clip. For details on selecting a destination reel, see “To select clips for a command or module:” on page 51.

If **flame** does not find a match, the message “No matches” appears in the message bar.

Matching Clips on the Desktop

You can also use the Search command to match frames in two different clips. Select a frame on the desktop to match, and use the Search command to find a matching frame in another clip on the desktop.

NOTE: If you want to search for source clips that you previously used to create a soft edit, use the Match Source command from the Editing menu. See “Matching a Soft Edit with Its Source Clips” on page 408.

To match a frame:

- In the Search menu, select Frame in the Clip/Frame option box.
The Desktop/Library option box changes to the Clip/Reel option box.
- In the Clip/Reel option box, select either Clip (to search a clip) or Reel (to search all clips on a reel).

3. Select an option from the Search option box.

Select:	To:
By Pixels	Perform a pixel-by-pixel comparison of the frames.
By ID	Compare frame identification numbers. Frames with the same frame identification number match. This search mode is faster than By Pixels.

4. Click Search in the lower left of the Search menu.

The cursor changes to a red arrow.

5. Select the frame to match.

The cursor changes to a green arrow if you are searching a clip, and to a white arrow if you are searching a reel.

6. Select the clip or reel on the desktop that you want to search for a match.

If the search is successful, the matching frame aligns to the centre of its reel, and the message

“Matches frame <n> in clip <x> centred on reel <y>”

appears in the message bar (where <n> is the frame number, <x> is the clip name, and <y> is the reel number on which the frame was found).

The reels on the desktop are numbered from bottom to top The red arrow cursor is active, and the message bar asks “Match Next? Y/N”. Press the **Y** key to search for another match, or the **N** key to end the search.

If the search is unsuccessful, the message “No matches” appears in the message bar, and the cursor changes to a red arrow. Use the red arrow to select another reel to search, or click anywhere outside of a clip to cancel the search.

NOTE: To cancel a search at any point, click anywhere inside the menu area.

The Menu Area

The menu area is where the Main menu, and other menus available through the Main menu, appear. The menu area also contains the message bar, which displays processing information, error messages, as well as the number of frames available on your local framestore.

The Main menu is the top of a menu hierarchy. If you click a Main menu button, the area to the right of the Main menu displays the menu associated with that button. For example, click Processing to display the Processing menu.

There are two important types of menu buttons: command buttons and module buttons.

Command Buttons

You use command buttons to apply an operation to a clip, or clips. A command button may bring up a new menu, at the right of the menu area, that you use to complete the command.

For example:

In the Processing menu, click the Consolidate command button. . .

LIBRARY					
EDITING	Cut	Repeat		Dissolve	Delete
PROCESSING	Insert	Reverse		Audio TimeStretch	Name
EFFECTS	Splice	Swap Shot		TimeWarp	Move
FORMAT	Replace	Consolidate		Commit	Copy
SYSTEM	Match Source	EDL		Force Render	Search

. . . and the Consolidate menu is displayed.

LIBRARY					
EDITING	Cut	Repeat	Dissolve	Delete	Clip
PROCESSING	Insert	Reverse	Audio TimeStretch	Name	Handles 60
EFFECTS	Splice	Swap Shot	TimeWarp	Move	
FORMAT	Replace	Consolidate	Commit	Copy	
SYSTEM	Match Source	EDL	Force Render	Search	

The Consolidate menu provides options for consolidating a clip.

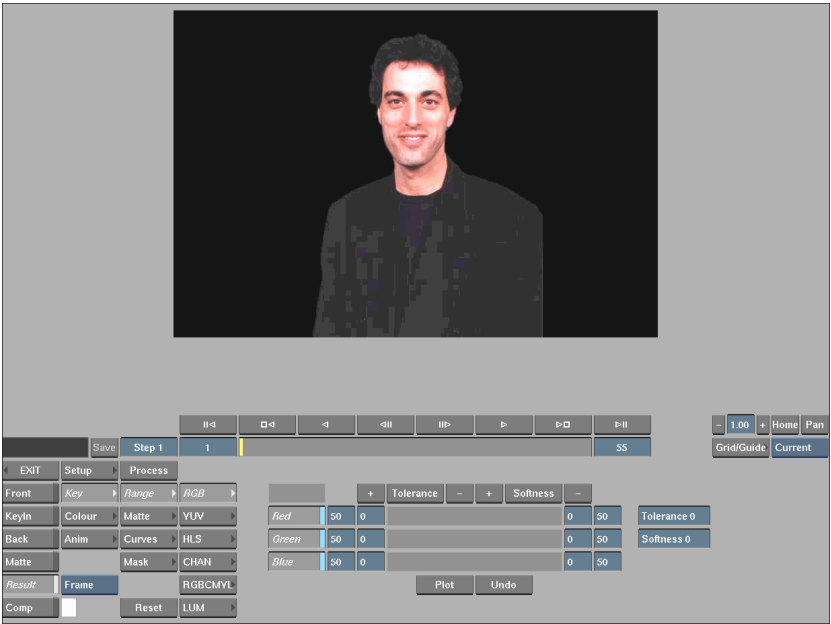
Module Buttons

A module button takes you into a module. When you enter a module, the image window area (see “The Image Window” on page 64) replaces the desktop, and the menu for the module appears in the menu area. For example:

In the Effects menu, click the Keyer module button, and select your clips...



... to enter the Keyer module.



Selecting Clips for a Command or a Module

After you click a command or module button, you select the clips to which you want to apply the command, or with which you want to enter the module. The number of clips you select depends on the command or module. The following outlines the procedure for applying a command to one or more clips, or accessing a module.

To select clips for a command or module:

1. Click the command or module button.

The cursor changes from a yellow cross to a coloured arrow. If you click a command button, a new menu specific to that command may appear in the menu area. Use the menu to set parameters and/or adjust options for the command.

2. Select the clips. You can select any clip in the desktop reels. To select a clip, click on the timecode or frame number of any frame in the clip. The colour of the arrow is significant when you select clips.

Use:	To:
Red arrow	Select the first clip (the front, or foreground clip).
Green arrow	Select the second clip (the back, or background clip).
Blue arrow	Select the matte clip.
White arrow	Select the destination reel for a processed clip.

3. Select the destination desktop reel for the processed clip by clicking on a desktop reel. When you select a destination reel, where you click on it determines where the generated clip appears on the reel:
 - Click anywhere on the destination reel *except* on a timecode or frame number, to place the generated clip at the end of the reel.
 - Click the timecode or frame number of a frame of a clip to place the generated clip after that clip.

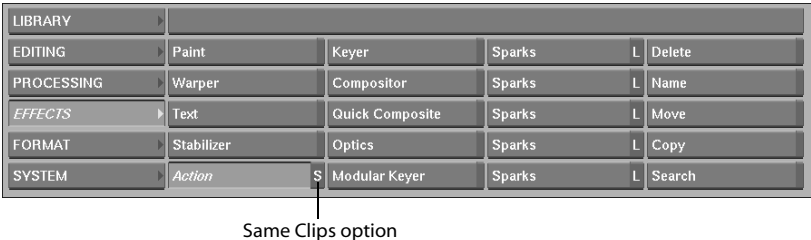
The command is applied, or the module opens.

Reapplying a Command or Reentering a Module with the Same Clips

The Same Clips option, available for some buttons, is a shortcut for selecting clips from the desktop reels. When you use the Same Clips option, **flame** uses the same clips you selected the last time you executed the command, or entered the module, in the current work session.

To use the Same Clips option:

1. Click a command or module button.
- If the letter S appears on the right side of the button, the Same Clips option is available.



2. If you selected a command with an associated menu, set the parameters for the command in the menu.
3. Click the button a second time.
- The status box containing the letter “S” turns light grey, and the cursor changes to a white arrow cursor.
4. Select the destination reel.
- The command is applied to, or the module loads, the same clips you selected when you last invoked the command or module.

Undoing Commands

You can undo most commands using the Undo controls. In the primary interface, the Undo controls appear in the lower right of the screen.



Click the Undo button to undo only the last command executed. Use the option box on the right of the Undo button to perform a multiple undo by selecting a command from the list. The selected command, and all commands appearing under the selected command, are undone.

Click the Redo button to redo the last command undone by the Undo command. Use the option box on the right of the Redo button to perform a multiple redo by selecting a command from the list. The selected command, and all commands appearing under the selected command, are redone.

You can set the number of possible Undo operations, and clear the Undo buffer. See “Undo Levels” on page 124 and “Clear Undo Buffer” on page 124.

Buttons, Option Boxes, and Fields

Buttons, option boxes, and fields are the principal elements of all menus.

Buttons

The following table illustrates the various button types, and describes their functions and purpose.

Item	Appearance	Use
Toggle buttons	Grey button with a blue bar on the right side.	Toggles the status of the item on or off. When enabled, button is recessed and text appears in <i>italic</i> face.
	<div><div>Fixed</div><div>enabled</div></div> <div><div>Fixed</div><div>disabled</div></div>	
Radio buttons	Same as toggle button, except the bar on the right is dark grey.	Sets a mode from a series of mutually exclusive options. In a radio button series, only one can be enabled at a time. Enabling one disables any other in the series.
	<div><div>Warp</div><div>enabled</div></div> <div><div>Warp</div><div>disabled</div></div>	
“Action” buttons	Grey raised button.	Initiates an action, such as process or “undo”.
Button arrows	A grey arrow on the left or right side of a button.	Navigates the item’s menu structure.
	<div><div>Camera</div><div>Arrow</div></div> <div><div>Camera</div><div></div></div>	Arrow on the right side indicates that clicking the button brings you into the item’s menu structure. Arrow on the left indicates that clicking the button takes you out of the item’s menu structure.
LED indicators	A rectangular coloured “light” on a button.	Determine the status of an item. When the LED is lit, the status is “on” or “active”. A red LED indicates a warning.
	<div><div>Local</div><div>LED</div></div>	

Item	Appearance	Use
Labels	Flat grey box (neither recessed nor raised).	Labels, or categorizes, the items that lie under it.
<div>Free Guide</div>		

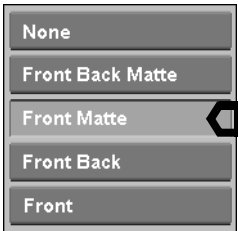
Option Boxes

An option box, or box, is a blue raised button that displays a list of mutually exclusive options when you press it. The name that appears in the option box is the currently active option. For example, in the following Clip option box, Front is the currently active option.



To select an option in an option box:

- Press the option box.
A list of options appears.
- Drag the cursor to the option you want.



- Release the cursor.
The list of options disappears, and the box displays the option you selected.

Fields

A field is a recessed box you use to enter text or numeric values. If a field appears blue, you can edit it. If it appears black, or greyed out, you cannot edit it.



When you click a field that accepts text, the keyboard appears. See “The Keyboard” on page 55 for more information on entering text. If the keyboard does not appear, use the workstation keyboard to type directly in the field. See “Desktop Preferences” on page 118.

Use one of the following methods to enter a numeric value in a field:

- Click on the numeric field, and use the numeric keypad to enter a value. See “The Numeric Keypad” on page 56.
- Press on the numeric field and drag the cursor. To increase the value, press in the field and drag to the right. To decrease the value, press in the field and drag to the left. To reset the value in the field to its default, **CTRL**-click in the field.

NOTE: When you drag the cursor, the rate at which the value increments increases as you apply more pressure to the stylus. When you use a mouse, the right mouse button changes the values at a faster rate than the left button.

The Keyboard

The keyboard appears when you are required to enter text, for example, when you click the Name button and select a clip to rename.

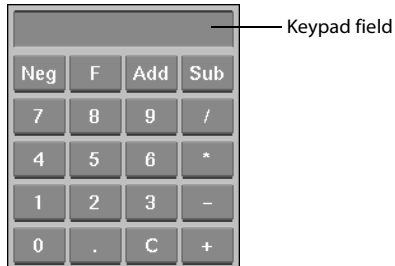
Keyboard display



You can use the keyboard by clicking directly on the keyboard on the screen, or by typing on the workstation keyboard. For example, you can click the letter C, or type it on the workstation keyboard. In either case, C appears in the keyboard display.

The Numeric Keypad

The numeric keypad appears in the lower right corner of the screen when you click on a channel value or a field that accepts numeric values.



Use the keypad as you would a calculator. When you click a number key, the number appears in the keypad field. Click the C to clear the keypad field.

Click the keypad field to transfer the value to the field or channel value, and close the numeric keypad. To transfer the value without closing the keypad, click the field or channel value.

NOTE: The F key is only available when you use the numeric keypad with timecodes and frame numbers.

Working with Timecodes and Frame Numbers

You can also use the numeric keypad to enter timecodes, and to convert between timecodes and frames.

To enter a timecode:

1. Click a timecode field.

The numeric keypad appears. The timecode appears in the keypad field.

2. Enter a new timecode.

NOTE: The period (.) key behaves differently when you work with timecodes. When you click the period key, **flame** inserts two zeroes at the current position in the keypad field. If you have already entered part of the timecode, **flame** inserts a colon followed by two zeros after the existing values.

To convert a timecode to frames:

1. Click a timecode field.

The numeric keypad appears.

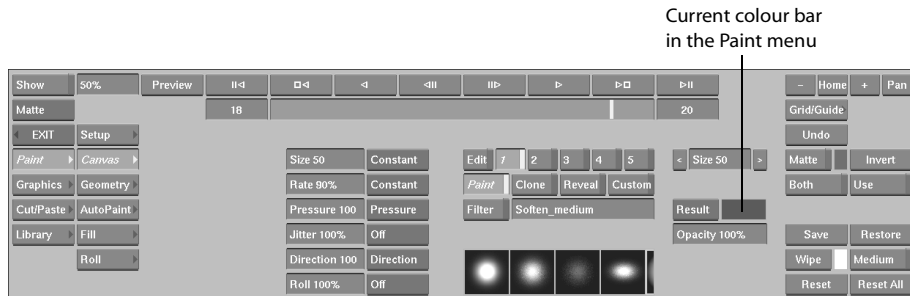
2. If necessary, enter a new timecode.
3. Click the F key to convert the timecode to frames.

NOTE: The F key converts the value in the keypad field. If the value is a timecode, **flame** converts it to frames. If the value is frames, it converts it to a timecode.

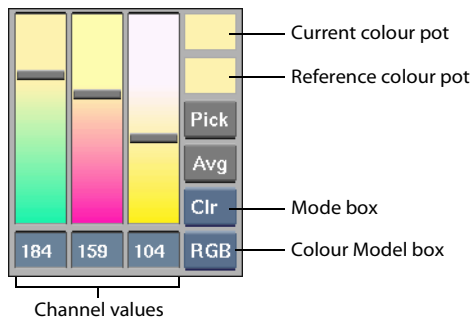
The Colour Picker

Use the colour picker to pick and mix colours from the image, and to display or change the colour values of any colour. It is available in most modules.

To display the colour picker, click on the current colour bar.



By default, when the colour picker appears, it appears in colour (Clr) mode. In this mode, you see three slider bars. Each slider bar represents one channel of the current colour model: RGB, YUV, or HLS (as specified in the Colour Model box). The colour value for each channel of the current colour appears at the bottom of the corresponding slider bar.



The Current colour pot displays the current paint colour in use, and the Reference colour pot displays the previous paint colour used.

The Mode box contains three options: Clr (default), Pot, and Paint. You can view and set channel values in Clr Mode, use and customize a colour palette in Pot mode, and mix colours in Paint mode. Certain operations are only available from a particular mode. The mode that is active when you exit the colour picker is the one that is active when you reenter the colour picker.

To hide the colour picker, click anywhere outside the colour picker.

Picking Colours

Use the Pick button to choose a colour from the image. The chosen colour can be transferred to the current colour bar or the colour palette.

To pick colours:

1. Click Pick.
The cursor changes to a colour picker.
2. Pick a colour from the image by clicking on that colour.
The selected colour appears in the Current colour pot.
3. Transfer the selected colour. To transfer it to the current colour bar, click the current colour bar. To transfer it to the colour palette, select Pot in the Mode box, then click and hold on a colour pot.

Mixing Colours

You can use the colour picker to mix colours.

To mix colours:

1. Select Paint in the Mode box.
The colour mixing area appears.
2. Click Pick, select a colour from the image, and brush the selected colour onto the mixing area of the colour picker. Repeat this for each additional colour you want to add to the mixing area.
3. To select a mixed colour, click Pick and click on the colour in the mixing area.
The mixed colour appears in the Current colour pot.
4. To clear the mixing area, click Clear.
5. Transfer the mixed colour. To transfer it to the current colour bar, click the current colour bar. To transfer it to the colour palette, select Pot in the Mode box, and then click and hold on a colour pot.

Choosing an Average Colour

You can use the Avg button or the Pick button to calculate the average colour of a sample of image pixels. You can choose sample pixels in two ways:

- Drag a selection box over the sample area.
- Drag the colour picker cursor over the sample pixels. This method is usually faster because it samples a smaller area of the image.

To sample colour values using a selection box:

1. Click Avg.

The cursor changes to the colour picker.

2. Click and hold the mouse button (or press down on the stylus) and drag the cursor diagonally to draw a selection box over the area of the image to sample.

The average colour of the sampled area appears in the Current colour pot.

NOTE: Alternatively, click Pick, hold down CTRL, and draw a selection box over the image area to sample. The average colour of the sampled area appears in the Current colour pot.

NOTE: Processing time is proportional to the size of the area sampled — the larger the sample area, the longer it takes to process.

To sample colour values using the colour picker cursor:

1. Click Pick.

The cursor changes to the colour picker.

2. ALT-drag the colour picker cursor over the image pixels you want to sample.

The average colour of the sampled pixels appears in the Current colour pot.

Viewing and Setting Channel Values

Use the Clr option in the Mode box to view and set the colour values for the individual channels of the current colour model.

To view and set channel values:

1. Select Clr in the Mode box.
2. Select a colour model in the Colour Model box.

Select:	To:
RGB	Display the red, green, and blue channel values of a colour.
YUV	Display the luminance, U-chrominance, and V-chrominance channel values of a colour.
HLS	Display the hue, lightness, and saturation of a colour.

3. Set the value for each channel using the slider bar, the numeric keypad, or the colour picker cursor:
 - To use the slider bar, press on a slider bar and drag it up or down. Dragging with the left mouse button changes values more slowly than dragging with the right mouse button. If you are using a stylus, apply more pressure to increase the rate at which values change.
 - To use the numeric keypad, click on the channel value at the bottom of the slider bar, and enter the value.

- To use the colour picker cursor, click Pick, and drag the colour picker cursor along a channel or an image until the required colour displays.

The colour pot in the upper right corner of the colour picker interactively updates as you change the channel values.

4. Transfer the mixed colour. To transfer it to the current colour bar, click the current colour bar. To transfer it to the colour palette, select Pot in the Mode box, and then click and hold on a colour pot.

Using and Customizing a Colour Palette

Use the Pot option in the Mode box to use and customize a 15-pot colour palette.

NOTE: The palette in the colour picker is independent from the palette in the Paint module.

To use the colour palette:

1. Select Pot in the Mode box.
The colour palette appears.
2. To select a colour, click one of the colour pots.
The selected colour appears in the Current colour pot.

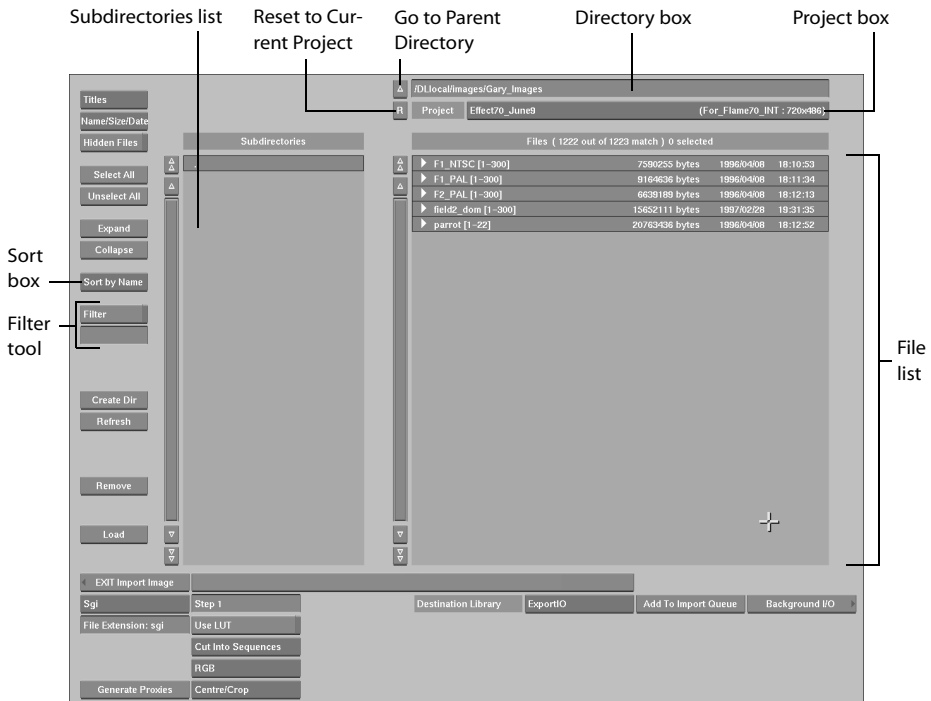
To customize the colours in the colour palette:

1. Choose a colour to store in the colour palette.
The selected colour appears in the Current colour pot.
2. Select Pot in the Mode box.
The colour palette appears.
3. Click and hold on the pot in which to store the selected colour.
The colour transfers to that colour pot.
4. Repeat this procedure for each colour you want to store in the colour palette.

When you select Pot in the Mode box, Save and Load buttons appear at the bottom of the colour picker. For information on saving and loading colour palettes, see Chapter 7, “Saving Setups and Preferences.”

The File Browser

You use the file browser to browse, load, and save files in the UNIX file system. Some of the buttons on the file browser may vary, depending on where you are in **flame** when you invoke the file browser. The following file browser appears when you click Import Image in the Library menu. This section describes only those buttons common to all file browsers.



Directory Box

The current directory appears in the Directory box. You can navigate to other directories using the navigation aids (such as the Go To Parent Directory arrow). Alternatively, click the Directory box and enter a new path to go to another directory.

You can create a new directory within the current one by clicking the Create Directory button and entering the new name.

NOTE: You can opt to have the on-screen keyboard appear when you click the Directory box. Use **CTRL+ALT+K** to toggle the keyboard on. Or, in the System menu, select Preferences, then Desktop Preferences, and toggle Keyboard On.

Project Box

The current project appears in the Project box. To load or save images, setups or preferences to a different project, select the project in the Project box. To reset to the current project, click the “R” button.

Subdirectories List

All available subdirectories within the current directory appear in the Subdirectories list. To go to a subdirectory, click its name in the list. All available files within the current subdirectory appear in the File list.

File List

All available files in the current subdirectory appear in the File list, depending on the options you specify in the File Extension box, the Filter settings (if any), and whether or not you are showing UNIX hidden files. You can view the File list in Titles or Proxy mode. Select a file to load by clicking the file in the File list. You can select a single file, a range of files, or all the files in a sequence. For more information, see “Selecting and Loading Files” on page 63.

Sort Box

You can sort files in the File list by name or by date.

Select:	To:
Sort by Name	Sort files alphabetically by name.
Sort by Date	Sort files by their or modification date.

Using the Filter Tool

When there are many files in a library, it can be cumbersome to search for a particular file. Use the Filter tool to display a particular subset of the existing files.

To use the Filter tool:

1. Click the Filter field.
The keyboard appears.
2. Type in a character string, including wildcards such as *, and click Enter.
For example, suppose you are loading a font to use in the Text module and you only want to display oblique fonts. All of the oblique fonts have “oblique” in the file name, for example, Courier-Oblique, Helvetica-BoldOblique, and Helvetica-Oblique. You would enter “*Oblique*” in the Filter field.

NOTE: The Filter field is case-sensitive.

The file browser reappears. The character string appears in the Filter field.

3. Enable the Filter button.

flame applies the filter. Only files that have the character string in their file name appear in the file browser.

NOTE: The Filter tool uses standard UNIX wildcard conventions. For example, if you filter on “Oblique” (without a wildcard), the tool does not recognize “Helvetica-Oblique”.

Selecting and Loading Files

To load a file into **flame**, you must first select it in the File list. You can load a single file, a range of files, or all files in a sequence. A sequence is a set of files that make up a clip. Each file in a sequence contains a single image, or frame.

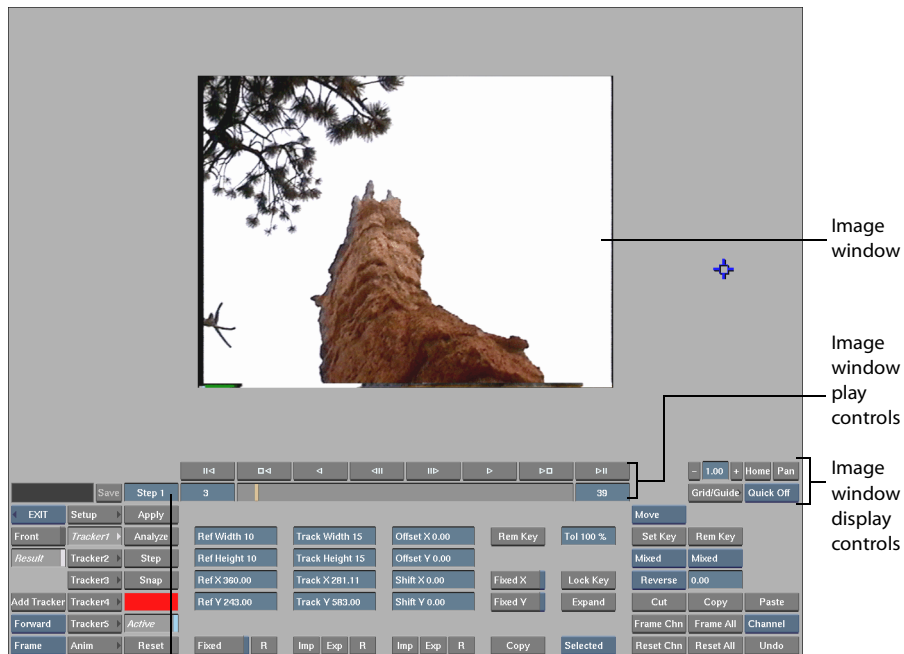
To select and load files:

1. Select the files you want to load:
 - To select a single file, click the file name in the File list.
 - To select a range of files, click the first file you want to include in the range and then **SHIFT**-click the last file you want to include in the range.
 - To add to a range of files, **CTRL**-click a file in the File list.
 - To select all files in a sequence, double click any of the files in the sequence in the File list.
 - To select all files in the list, click Select All. Click Unselect All to unselect all files.
2. Click Load.

The Image Window

When you open a module, the image window appears in the upper area of the screen.

The image window is where you view the clip, or clips, loaded into the module. You can view only one frame of a clip at a time. The image window is also the work area in which you apply effects to clips. The result clip updates automatically as you apply effects.

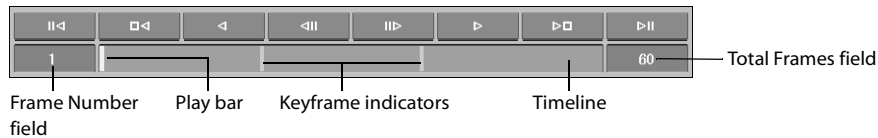










Step Frame Processing field

Step Frame Processing Field — Appears in the following modules: Warper, Text, Stabilizer, Action, Keyer, Compositor, Optics, Filter, and the Colour Corrector. You use this field to set which frames of the clip **flame** processes when it processes the clip. For example, a value of five means **flame** only processes every fifth frame. If the total length of the clip is 30 frames, and **flame** only processes every fifth frame, the result clip has a total length of six frames.

The Image Window Play Controls

Use the image window play controls to move forward and backward through the clip.



Click:	To:
	Go to the first frame in the clip.
	Go to the previous keyframe.
	Play the clip backward.
	Move one frame backward.
	Move one frame forward.
	Play the clip forward.
	Go to the next keyframe.
	Go to the last frame of the clip.

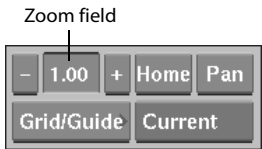
Frame Number field — Displays the frame number of the frame currently in the image window. To move quickly to any frame in the clip, enter the frame number in this field. You can change this field to a Timecode field. See “Displaying Timecodes, Frame Numbers, and Names” on page 120.

Total Frames field — Shows the total number of frames in the clip. In some modules, such as Action, you can change the length of the clip to be generated by entering the total number of frames in this field. You can change this field to a Timecode field. See “Displaying Timecodes, Frame Numbers, and Names” on page 120.

Timeline — Contains a vertical yellow bar called the play bar. The play bar indicates the relative position of the current frame in the clip. You can play the clip forward or backward by dragging the play bar right or left.

The Image Window Display Controls

Use the image window display controls to zoom in or out, to pan the image, and to overlay the grid and/or guides on the image.



Click:	To:
+	Zoom in on the image. See Note.
-	Zoom out from the image. See Note.
Zoom field	Zoom in or out by dragging on the field (left to zoom out, right to zoom in), or by clicking on the field and entering a specific zoom factor. See Note.
Home	Return to the default size image window.
Pan	To pan the image. When you click Pan, the cursor changes to a grabber hand. Move the grabber cursor over the image window, press down and drag the image window. See Note.
Grid/Guide	Toggle the Grid and Guides menu on and off.
Grid/Guide box	Select a set of grid and guides preferences, or turn off the grid and all guides simultaneously.

NOTE: Use the Save Zoom/Pan button in the Grid and Guides menu to save Zoom and Pan factors.

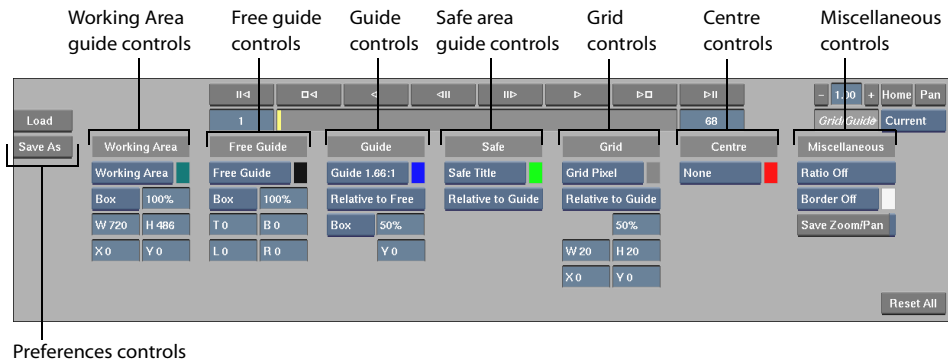
Grid and Guides Menu

The grid and guides are references you superimpose on the image. Neither the grid nor the guides appear on the processed clip.

A grid is a set of horizontal and vertical reference lines that overlay the image and the upper area of the screen. A grid is useful when you need to make relative measurements, or need alignment information about various features of a shot. You can choose the colour and opacity of a grid.

There are five kinds of guides: working area guides, ratio guides, free guides, safe area guides, and the centre guide. You can choose the colour of each guide independently. Choosing a different colour for each guide makes it easier to distinguish between guides when more than one is active. You can also set opacity for working area guides, ratio guides, and free guides. Reducing opacity is useful when you want to see both a guide and what it masks.

You use the Grid and Guides menu to turn on and configure the grid and guides.



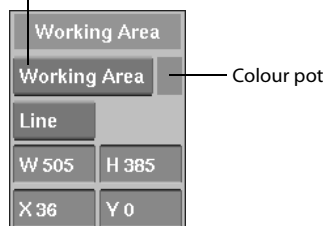
NOTE: The Paint module Grid and Guides menu varies from the Grid and Guides menu described here. See “The Grid and Guides Menu in the Paint Module” on page 75.

Working Area Guide

A working area guide defines a portion of the image that you do not use. For example, if the result resolution is film, and audio will be printed on the film, you might use a working area guide to indicate the part of the image that will disappear when the audio is added.

Use the Working Area controls to create a working area guide on the image.

Working Area box



Select None in the Working Area box to use the entire image as the work area. Select Working Area to define a working area guide on the image. Use the following to configure the position and appearance of the guide.

Use:	To:
Colour pot	Select a colour for the guide. To select a colour, click the colour pot, and use the colour picker to choose a colour.
Box/Line box	Select a style for the guide. If you select Box, use the field that appears beside the box to set the opacity of the guide (100% is completely opaque and 0% is completely transparent).

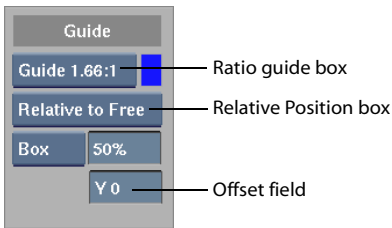
Use:	To:
W, H fields	Set the width and height of the guide respectively. Units are in pixels.
X, Y fields	Offset the guide along the X- or Y-axis respectively. Units are in pixels. By default, when you set the width or height of the guide, flame centres the guide on the image. This option lets you move it left or right (X-axis), or up or down (Y-axis), on the image. The maximum offset along the Y-axis is the height of the lower area of the guide. The maximum offset along the X-axis is the width of the left area of the guide.

Ratio Guide

A ratio guide is a guide that represents a particular aspect ratio. This is useful, for example, for viewing the result at different aspect ratios. There are ratio guides for Academy, CinemaScope, American 35mm, European 35mm, and HDTV aspect ratios.

You can also create a custom ratio guide using a specific aspect ratio, or specific pixel dimensions. A custom ratio guide always has its top and bottom areas equal in size, and its left and right areas equal in size.

Use the Guide controls to select a standard ratio guide, or to create a custom ratio guide using either a custom aspect ratio or a custom width and height. Only one ratio guide can be active at a time.

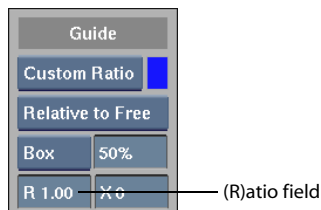


To use a standard ratio guide, select one from the Ratio guide box. Use the controls that appear to configure the position and appearance of the guide. See “Working Area Guide” on page 67 for descriptions of the colour pot and Box/Line controls.

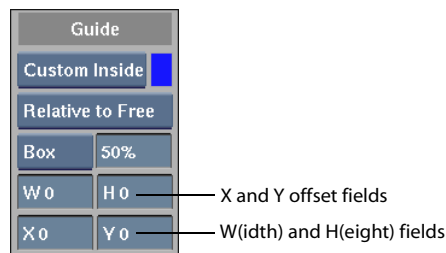
Use:	To:
Relative Position box	Select a position for the guide. Select Relative to Border to centre the guide with respect to the Border of the image. Select Relative to Work to centre the guide with respect to the working area of the image (as defined by the Working Area controls). Select Relative to Free to centre the guide with respect to the free guide. If no free guide exists, flame centres the guide with respect to the working area of the image.

Use:	To:
Offset field	Offset the guide along the X- or Y-axis, depending on the guide. Units are in pixels. By default, when you set the width of the guide, flame centres the guide on the image. This option lets you move it up or down on the image. The maximum offset along the Y-axis is the height of the lower area of the guide. The maximum offset along the X-axis is the width of the left area of the guide.

To create a custom ratio guide using a specific aspect ratio, select Custom Ratio. Enter the aspect ratio for the guide in the R(atio) field. Use the remaining controls to configure the position and appearance of the guide, as described for standard ratio guides.



To create a custom ratio guide by specifying the width and height of the guide, select Custom Inside.



Use the W(idth) and H(eight) fields to enter the width and height of the guide respectively. Units are in pixels. **flame** creates the guide from the centre of the image out. Initially the guide covers the entire region of the image selected in the Relative Position box.

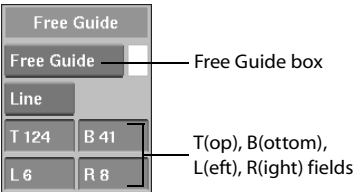
Use the X and Y fields to offset the guide along the X- or Y-axis respectively. By default, when you set the width or height of the guide, **flame** centres the guide on the image. This option lets you move it left or right (X-axis), or up or down (Y-axis), on the image. Units are in pixels.

Use the remaining controls to configure the position and appearance of the guide, as described for standard ratio guides.

Free Guide

A free guide differs from a custom ratio guide in that you can vary the size of the top, bottom, left, and right areas of the guide independently. A free guide is useful if you need to have more than one Guide active at a time.

flame always creates the free guide relative to the working area guide.

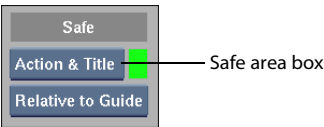


Select Free Guide in the Free Guide box to create a free guide. Use the controls that appear to configure the position and appearance of the guide. Use the T(op), B(ottom), L(ef), and R(ight) fields to set the size, in pixels, of the top, bottom, left, and right areas of the guide, respectively. See “Working Area Guide” on page 67 for descriptions of the colour pot and Box/Line controls.

Safe Area Guide

A safe area guide represents the safe title, safe action, or both safe title and safe action area of a defined region of the image. You can also create a custom safe area guide.

Use the Safe area guide controls to select a standard safe area guide, or to create a custom safe area guide.



To use a standard safe area guide, select one from the Safe area box. Configure the position and appearance of the guide using the colour pot, and the Relative Position box. See “Working Area Guide” on page 67 for a description of the colour pot.

Use:

Relative
Position box

To:

Select a position for the guide.

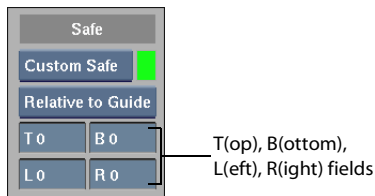
Select Relative to Border to centre the guide with respect to the Border of the image.

Select Relative to Work to centre the guide with respect to the working area of the image (as defined by the Working Area controls).

Select Relative to Free to centre the guide with respect to the free guide. If no free guide exists, **flame** centres the guide with respect to the working area of the image.

Select Relative to Guide to centre the guide with respect to the ratio guide.

To create a custom safe area guide, select Custom Safe.

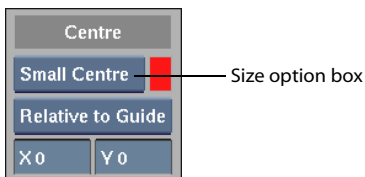


Use the T(op), B(ottom), L(ef), and R(ight) fields to set the sizes of the top, bottom, left, and right sides of the guide respectively. Units are in pixels. Use the remaining controls to configure the position and appearance of the guide, as described for a standard safe area guide.

Centre Guide

A centre guide indicates the centre of a defined region of the image.

Use the Centre controls to turn on the centre guide, and to configure its position and appearance.

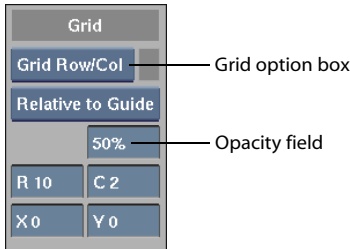


See “Working Area Guide” on page 67 for a description of the colour pot, and “Safe Area Guide” on page 70 for a description of the Relative Position box.

Use:	To:
Size box	Select a size for the centre guide, or turn it off.
X, Y fields	Offset the guide along the X- or Y-axis respectively.Units are in pixels.

Grid Controls

Use the Grid controls to turn on the grid, and to configure its position and appearance.

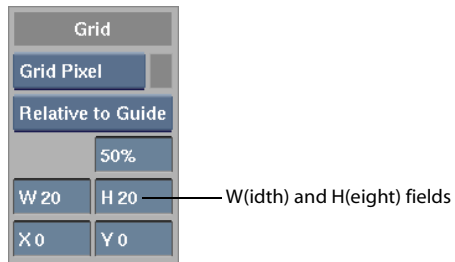


To define a grid using row and column values, select Grid Row/Col from the Grid box. Use the controls that appear to configure the position and appearance of the grid. See “Working Area Guide” on page 67 for a description of the colour pot.

Use:	To:
Opacity field	Set the opacity of the grid colour (100% is completely opaque and 0% is completely transparent).
Relative Position box	<p>Select a position for the grid.</p> <p>Select Relative to Border to centre the grid with respect to the Border of the image.</p> <p>Select Relative to Work to centre the grid with respect to the working area of the image (as defined by the Working Area controls).</p> <p>Select Relative to Free to centre the grid with respect to the free guide. If no free guide exists, flame centres the grid with respect to the working area of the image.</p> <p>Select Relative to Guide to centre the grid with respect to the ratio guide.</p>
	<p>NOTE: If you resize or offset the guide (or image) referred to here, the grid adjusts accordingly. For example, if you select Relative to Work and you resize or offset the working area guide, the grid resizes or offsets accordingly.</p>
R, C fields	Set the number of rows and columns, respectively, that appear over the region of the image selected in the Relative Position box.

Use: X, Y fields **To:** Offset the grid along the X- or Y-axis respectively. Units are in pixels.
 Maximum offset along the X-axis is the width of one column. Maximum offset along the Y-axis is the height of one row.

To define a grid using pixel values, select Grid Pixel.



Use the W(idth) and H(eight) fields to set the width and height, respectively, of each square in the grid. Units are in pixels. Use the remaining controls to configure the position and appearance of the grid, as described above for defining a grid using row and column values.

Miscellaneous Controls

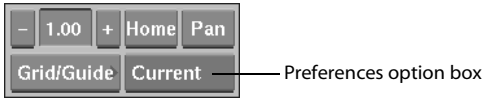
The Miscellaneous controls handle aspects not specific to the grid or to a particular guide.



Use:	To:
Ratio box	<p>Toggle between the two kinds of pixels to use to display the image.</p> <p>Select Ratio Off to display the image using square pixels. This option enhances performance by eliminating the need to map pixels with a non-square aspect ratio onto the square pixels of the SGI monitor.</p> <p>Select Ratio On to display the image using the pixel aspect ratio specified in the framestore setup for the project. This option requires more processing time to calculate the mapping of non-square pixels onto the square pixels of the SGI monitor. See “Framestore Setup” on page 85 for more information on pixel aspect ratio.</p> <p>To properly view video aspect ratio, it is recommended you select Ratio Off and view the result on a broadcast monitor.</p> <p>NOTE: If the pixel aspect ratio in the project’s framestore partition is 1, there is no difference between Ratio On and Ratio Off.</p>
Colour pot	Select a colour for the image border. To select a colour, click in the colour pot and use the colour picker to choose a colour. Click in the Current colour pot of the colour picker to transfer the colour to the image border colour pot.
Border box	Toggle the image border (the line around the perimeter of the image) on and off.
Save Zoom/Pan toggle button	<p>Toggle saving the zoom and pan factors on and off.</p> <p>Enable Save Zoom/Pan to include the zoom and pan factors whenever you save the Grid and Guides preferences.</p> <p>Disable Save Zoom/Pan to exclude the zoom and pan factors whenever you save the Grid and Guides preferences.</p>

Preferences Controls

Use the Preferences controls to load and save preferences. All loaded preferences appear in the Preferences box, allowing you to quickly switch from one set of preferences to another.



When you change a setting in the Grid and Guides menu, **flame** immediately saves the change to the Current preferences, and the Preferences box displays Current. The change occurs across all modules.

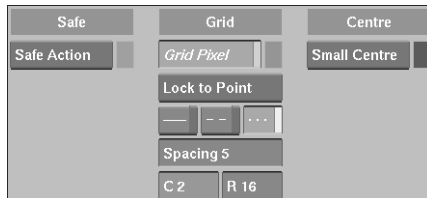
When you reenter a module, the preferences appear as they were when you last exited the module, or, if you have made changes since you last exited, the Current preferences are active.

To save preferences, make the changes to the settings, select the Preferences you want to save to from the Preferences box, and click Save As. Use the file browser to choose a file name and save the preferences.

NOTE: You cannot save or load Grid and Guides preferences in the Paint module.

The Grid and Guides Menu in the Paint Module

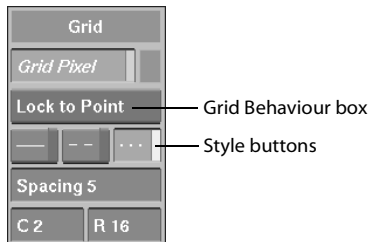
The Grid and Guides menu in the Paint module contains only the Safe area guide controls, the Grid controls, and the Centre controls.



The Safe area guide controls and Centre controls are subsets of their corresponding controls in the Grid and Guides menu in the other modules. The Grid controls are different.

Grid Controls

In the Paint module you can set the behaviour of the grid so that paint strokes snap or lock to points on the grid. Use the Grid controls to configure the appearance and behaviour of the grid.



Click Grid Pixel to toggle the grid on and off. When Grid Pixel is on, use the following controls to configure the appearance and behaviour of the grid.

Use:	To:
Grid Behaviour box	<p>Select the behaviour of the grid with respect to paint strokes.</p> <p>Select Snap to Point to snap each point of a stroke to the nearest intersection of a horizontal and a vertical grid line.</p> <p>Select Snap to Line to snap the current point of a paint stroke to the nearest point on a horizontal or vertical grid line.</p> <p>Select Lock to Point to lock each point of a paint stroke to the nearest intersection of a horizontal and a vertical grid line.</p> <p>Select View to use the grid without snap to or lock to options.</p>
Style buttons	Select a style for the lines of the grid. The style is either solid line, dashed line, or dotted line.
Spacing field	Set the number of pixels between the dashes in a dashed line style, or between the dots in a dotted line style.
C, R fields	Set the number of columns and rows respectively in the grid.

Resetting the Grid and Guides

Use the Reset All button in the lower right corner to reset the grid and all guides to their default values.

Turning the Grid and Guides On and Off

Turn the individual grid or guide on or off using their respective controls. Turn off both the grid and all guides at once by selecting Quick Off in the option box to the right of the Grid/Guide button.



This option box remains available in the module, outside of the Grid and Guides menu.

NOTE: Quick Off is not available in the Paint Module.

The Player

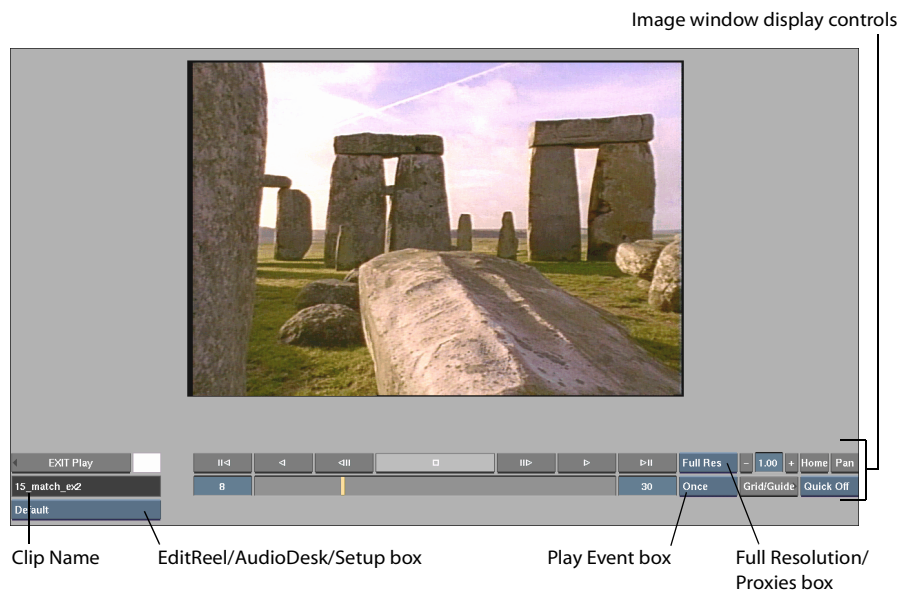
Use the Player to play a clip at a specified rate. The image resolution depends on the image size that you choose. Since the frames can be read directly from the disk, you can play a clip of any length.

The Player contains the image window and the Play menu area.

To play a clip in the Player:

1. In the primary interface, position the clip you want to play at the centre of the desktop reel.
2. In the reel controls, click Play. Alternatively, position the cursor over the clip on the desktop and press the **ESC** key.

The Player appears with the selected clip loaded in the image window.



3. Use the image window controls to play the clip. See “The Image Window Play Controls” on page 65.

To play the clip repeatedly, select Repeat from the Play Event box, and then play the clip.

You can also shuttle through a clip by clicking and holding in the image window. The clip plays until you release the stylus or mouse button.

NOTE: If you are playing from memory, you also have a PPong (ping-pong) option in the Play Event box. For more information on playing from memory, see “Player Setup Options” on page 78.

The Full Screen Player

Use the Full Screen Player to play an HD image, or proxies larger than NTSC, in the Player. To enter the Full Screen Player, in the Play menu area or in the image window controls area, swipe the cursor against the side of the screen. To return to the Player, swipe the cursor against the side of the screen a second time. This assumes you have Swipe Enabled in the Setup menu of the Player.

NOTE: The Setup/AudioDesk/EditReel box is not available in the Full Screen Player. This means you cannot play an audio track, or modify a soft edit in the Full Screen Player.

The Play Menu Area

The Play menu area consists of the Play Event box, the Full Resolution/Proxies box, and the EditReel/AudioDesk/Setup box.

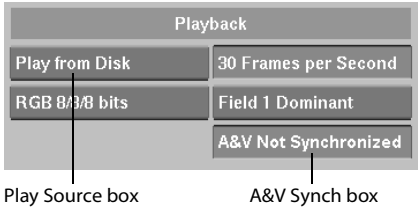
Play Event Box — Determines what happens when you play the clip to the end. Select Once to have the clip stop when it reaches the end. Select Repeat to have the clip loop back to the beginning when it reaches the end.

Full Resolution/Proxies Box — Use the Full Resolution button to play the specified clip at Full resolution, but at a slower frame rate.

EditReel/AudioDesk/Setup Box — Use the EditReel option to modify soft edits. For more information, see “The EditReel” on page 386. For more information on the AudioDesk option, see Chapter 24, “Using Audio.” For more information on the Setup option, see “Player Setup Options” on page 78.

Player Setup Options

When you select Setup in the EditReel/AudioDesk/Setup box, Playback and Edit Reel appear in the menu area.



Use the Play Source box to select the source of the playback. The option you select affects the playback speed.

Play from Memory — Plays high-resolution material that cannot be played at full speed from the framestore. **flame** loads the clip into memory when you enter the Player and when a change is made in the EditReel. If there is not enough memory for the clip, a region of the clip that fits

into memory is loaded. For example, if the clip is 1000 frames but only 500 can fit into available memory, **flame** starts from the current frame position and loads frames until all the memory has been used. See “Playing Clips from Memory” on page 79.

Play from Disk — Plays from the framestore. You do not have to wait for **flame** to load the clip into memory; however, the clip plays at slower frame rate.

Play to Video — Plays field-based material at full-speed with correct field dominance on the broadcast monitor. This option is necessary on systems where the Hires token cannot be set to 30 Hz (such as the O2). In this case, using the other options causes random field dominance on the broadcast monitor.

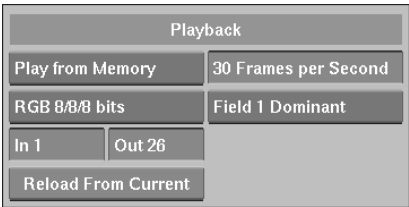
Play Audio Only — Plays only the audio.

Playing Clips from Memory

When playing clips from memory, you can specify start and stop points for the clip being played by dragging the start and stop point indicators from either end of the timeline to the appropriate frames. Alternatively, you can set the start and stop (in and out) points in the Playback area of the Setup menu.



If the clip you are playing from memory is very long (i.e. too long to fit into system memory), it loads as much as it can into memory beginning with the current frame. As you move the indicator on the timeline, you may want to reload the clip using the Reload From Current button.



Maintaining Audio or Video Playback Fidelity (A&V Synch Box)

Use the A&V Synch box to maintain either audio or video playback fidelity. When playing long clips, the system resources may run low, causing either audio frames to slip or video frames to be dropped.

Select:	To:
A&V Synchro-nized	Maintain audio fidelity. If the system's hardware resources run low as the clip is playing, video frames are dropped to maintain audio playback.
A&V Not Synchronized	Maintain video fidelity. If the system's hardware resources run low as the clip is playing, audio frames may slip to maintain video playback.

For more information about working with audio, see Chapter 24, "Using Audio."

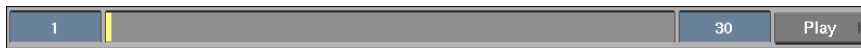
Playing a Clip in a Module

You can access the Player from most modules. This allows you to process a clip and then play it without exiting to the desktop reels. The Play button appears in the menu after the clip has been processed.

To play a clip in a module:

1. Process the clip in the module.

The Play button appears to the right of the timeline.



2. Click Play.

The Player appears.

Play to Air

You can organize a play list on a desktop reel, then enter the Player to make all the clips on the play list immediately available for broadcast.

NOTE: The EditReel is not available when you use the play-to-air feature.

To assemble and play a play list:

1. Assemble all of the clips in the play list on the same desktop reel, in the order in which you want to play them.
2. Press **CTRL+ESC** over the first clip on the play list.

The Player appears. The image window displays the first frame of the first clip on the play list. **flame** cues all clips on the play list for immediate playback.

3. Use the play controls to play through the play list. The play controls function as follows for the play-to-air feature.

Click: To:



Play the clip from the current location. If you click this button while parked on the last frame, the next clip loads and plays to the end.



Play the clip backwards from the current location. If you click this button while parked on the first frame, the previous clip loads and plays to the beginning.



Step forward one frame. If you click this button while parked on the last frame, you advance to the first frame of the next clip in the play list.



Step backward one frame. If you click this button while parked on the first frame, you step to the last frame of the previous clip in the play list.



Go to the end of the clip. If you click this button while parked on the last frame, you go to the first frame of the next clip.



Go to the beginning of the clip. If you click this button while parked on the first frame, you go to the last frame of the previous clip.

Player Hot Keys

Hot Key:	Use:
ENTER	Play the clip.
-(minus)	Play the clip in reverse.
SPACEBAR	Stop playing the clip.

The Reference Buffer

The reference buffer is a buffer that holds a reference clip. The reference clip is a clip you refer to as you work. For example, in the Colour Corrector the reference clip might contain colours you want to use in a clip.

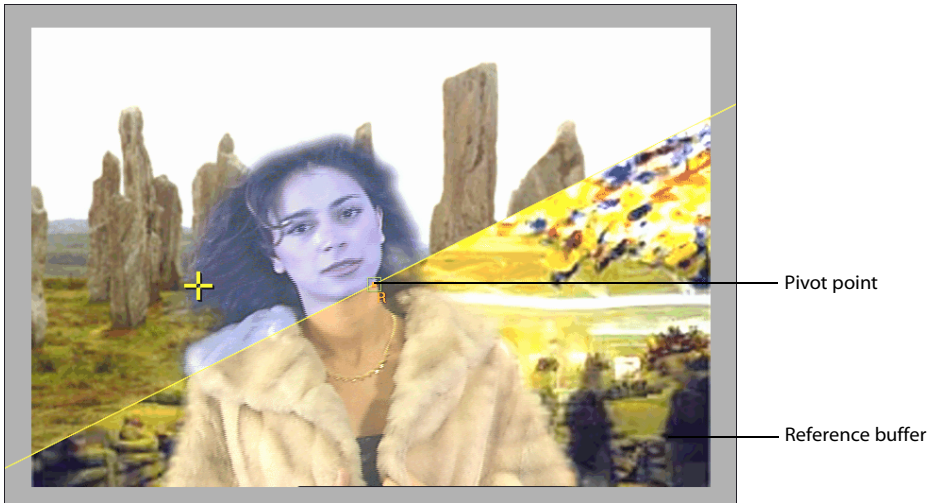
The reference buffer is available only in the Colour Corrector and Garbage Masks; in the Colour Warper node in both Batch and the Modular Keyer; and in all render-only nodes in Batch.

Turning the Reference Buffer On/Off

Toggle the reference buffer on or off by clicking Split On/Off in the reference buffer controls in the Setup menu.



When the reference buffer is on, it occupies the same region of the screen as the clip. A bar separates the reference clip from the clip. The letter R indicates which side of the bar displays the reference clip. The square with the centre dot is the pivot point of the bar.



You can adjust the position, rotation, and appearance of the bar as you work on a clip.

To:	Do this:
Rotate the bar	Click and drag on the bar.
Move the bar	Click and drag on the pivot point.
Reset the bar	CTRL-click on the pivot point.
Show/hide the bar	CTRL-click on the bar.

Loading and Flushing the Reference Buffer

You load and flush the reference buffer using the reference buffer controls.

To load the reference buffer:

1. Enter the Setup menu.
2. If the reference buffer is not visible, click Split On/Off in the reference buffer controls.
The reference buffer appears. If the reference buffer covers the entire clip, you may want to adjust the bar to see the clip as well.
3. To select the reference clip, click one of the clip view buttons (Front, Back, Matte, etc.).
The reference clip appears in the clip area. The reference buffer remains empty.
4. To transfer the clip to the reference buffer, click Grab.
The reference clip appears in the reference buffer.
5. To select the clip you want to work on, click one of the clip view buttons.
The clip appears in the clip area.

To flush the reference buffer:

1. Enter the Setup menu.
2. In the reference buffer controls, click Flush.

Reference Buffer Hot Keys

Hot Key:	Use:
CTRL+G	Transfer the clip to the reference buffer (same as Grab).
CTRL+F	Flush the reference buffer.
CTRL+B	Toggle the bar on and off.

System-Wide Hot Keys

Hot Key:	Use:
ALT-	To suppress a confirmation request. When you perform an action which requires confirmation, and simultaneously press the ALT key, flame carries out the action without confirmation.
ENTER	To confirm a confirmation message.
SPACEBAR	To abort an operation, such as clip processing, playing, etc.
CTRL-	When selecting items, use CTRL-<SELECT> for multiple selections.
CTRL+Z	Undo the last operation.

Hot Key:	Use:
CTRL+C	Copy.
CTRL+V	Paste.

The framestore is the high-capacity disk storage array on which your clips are stored. You divide your framestore into soft partitions, usually based on frame resolution. You can create, delete, and modify soft partitions.

Summary

In this chapter, you learn about:

- “Accessing the Framestore Setup Menu” on page 86
- “The Partition List” on page 88
- “Creating a Discreet Filesystem Soft Partition” on page 89
- “Adding a Partition” on page 90
- “Changing a Partition” on page 91
- “Changing the Partition Parameters” on page 91
- “Enabling a New Framestore Setup” on page 95

flame’s framestore uses an array of hard drives (usually a Discreet **stone**® disk array) that are separate from your internal system hard drive. The fast access and large volume hard drives in these units provide optimal performance.

Your **flame** system can be configured to use a framestore based on the Discreet Filesystem running on a **stone** disk array.

About Framestore Management Systems

The traditional logical volume method has become relatively obsolete with the development of the Discreet Filesystem, which provides a much more efficient and versatile system of framestore management.

The logical volume method of framestore management requires you to pre-allocate frame storage space within each partition that you create on your framestore. The Discreet Filesystem creates dynamic partitions that increase or decrease as material is added or removed. The space that each partition occupies on the framestore is determined by the amount of storage space that is actually used by stored clips.

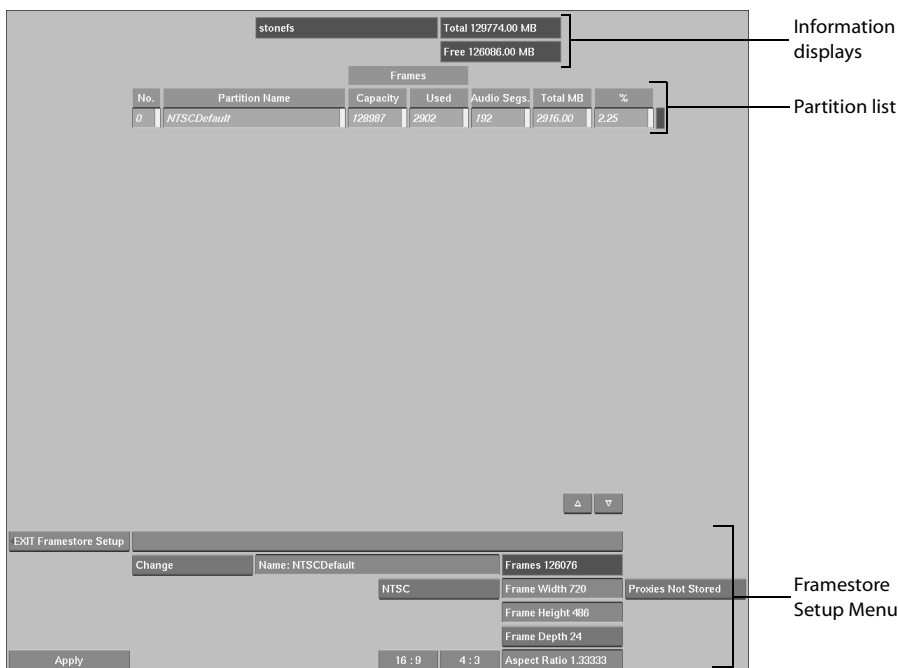
Using the logical volume method, you create two equal partitions (one to store video resolution clips, and the other for Film or HDTV resolution) and the partitions are static. For example, if you run out of space on your NTSC partition, you must delete clips before adding new clips even if there is space left in your Film partition. With the Discreet Filesystem, the entire framestore is treated as one storage unit, and the remaining amount of space on the framestore determines how many clips can be added to any partition.

Accessing the Framestore Setup Menu

You can access the Framestore Setup menu two ways:

- Select New in the Project box when you first start **flame** to enter the Project Management menu. Click New in the Partition box to enter the Framestore Setup menu. In this menu, you can create partitions to be used by specific projects created with **flame**.
- Click System in the Main menu, and then click Framestore Setup to enter the Framestore Setup menu.

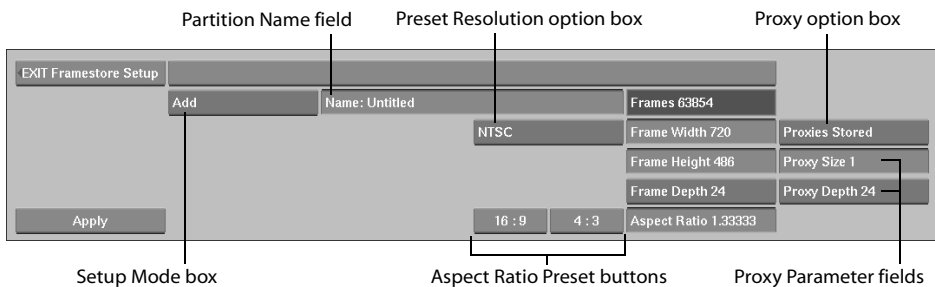
With the Discreet Filesystem, you configure your dynamic soft partitions in the Framestore Setup menu.



The displays at the top of the screen contain information about the present structure of your framestore, including the device type, the amount of storage space being used, and the amount of available storage space.



Use controls in the Framestore Setup menu to modify the existing structure of your framestore.



The Partition List

The current partitions in the framestore are listed in the Partition list.

stonefs

Total 129774.00 MB

Free 126086.00 MB

Frames

No.	Partition Name	Capacity	Used	Audio Segs.	Total MB	%
0	NTSCDefault	128987	2902	192	2916.00	2.25

Partition ratio gauge

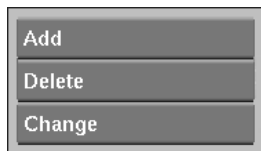
Each entry in the list contains:

- No.: the soft partition number. The number may change as you add and delete partitions. For this reason, a soft partition is always identified by its name rather than its number.
- Partition Name: the name of the soft partition.
- Frames Capacity/Used: the total number of frames that can be stored in the partition, including the frames already stored.
- Audio Segs: the audio segments (the amount of space taken up by audio data, expressed in frame units).
- Total MB: the total disk space used by the partition in MB.
- %: the percentage of the framestore space being used by the frames currently stored in the soft partition.

When you start **flame** for the first time, you specify a new project to work in and create a partition in the Framestore Setup menu in order to store any clips on your framestore.

You can add, delete, and change partitions. Each time you use one of these options and click Apply, the Partition list is updated. You can also use the Framestore Setup menu to change the size and aspect ratio of the frames and/or proxies in a new or existing partition.

Use the Setup Mode box to add, delete, and change partitions.



Select:

Add
Delete
Change

To:

Add a new partition to your framestore.
Remove an existing partition from your framestore.
Change the name, resolution, size and screen aspect ratio of the frames and proxies.



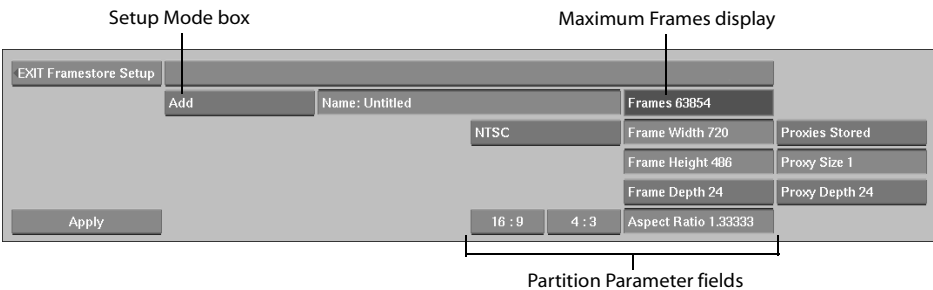
WARNING: Changing the resolution or frame depth on your framestore will cause all frames within the partition to be deleted.

Creating a Discreet Filesystem Soft Partition

With the Discreet Filesystem, you do not have to pre-allocate space for new partitions. The framestore is treated as one large storage area to which you can add soft partitions without deleting or modifying any of the existing partitions.

The number of frames that can be saved in any soft partition is limited only by the amount of space on the framestore. Removing clips from one soft partition frees up storage space for other soft partitions.

If you do not have space available and want to add a new partition, delete clips in one of the existing partitions.



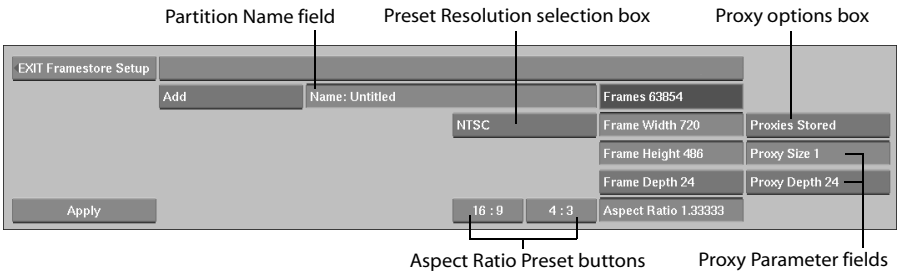
To add a Discreet Filesystem soft partition:

1. Go to the Framestore Setup menu.

NOTE: If this is the first time that you run **flame**, you will enter this menu from the Project Management menu. Otherwise, you will usually enter from the System menu.

2. Select Add from the Setup Mode box.

The Add Partition menu appears.



3. Enter a name for the new soft partition in the Partition Name field.

It is not possible to create a partition titled Untitled.

NOTE: When naming a partition, you must use alphanumeric characters. You can use spaces, but you cannot use any special characters such as punctuation marks.

4. Set the parameters for the new soft partition (frame width and height, aspect ratio, and so on). For more information on partition parameters, see “Changing the Partition Parameters” on page 91.
5. Click Apply.

The new partition is added to the Partition list.

Adding a Partition

You can create a new partition in your framestore using the Setup Mode box. When using a logical volume file system, your partitions occupy the entire capacity of the framestore. You must pre-allocate the size of the partition to have enough capacity for all clips of a specific resolution on that partition. If no space is currently available, you must delete an existing partition before adding a new one.

NOTE: When adding a partition, you must specify partition parameters such as the frame size, proxy size, and aspect ratio of the images in the partition you are modifying or creating. This affects the number of frames that can be stored in the partition.

To add a partition:

1. Make sure there is space for the new partition. If necessary, delete an existing partition.
2. Select Add from the Setup Mode box.

EXIT Framestore Setup			
Add	Name: Untitled	Frames 63854	
	NTSC	Frame Width 720	Proxies Stored
		Frame Height 496	Proxy Size 1
		Frame Depth 24	Proxy Depth 24
Apply	16 : 9	4 : 3	Aspect Ratio 1.33333

3. Click the Name field to access the keyboard. Enter a name for the partition you are adding.
4. Use the Parameter fields to specify the parameters of the new partition.
5. Click Apply.

The new partition appears in the Partition list.

6. After you finish setting up the framestore, enable the new setup. See “Enabling a New Framestore Setup” on page 95.

Changing a Partition

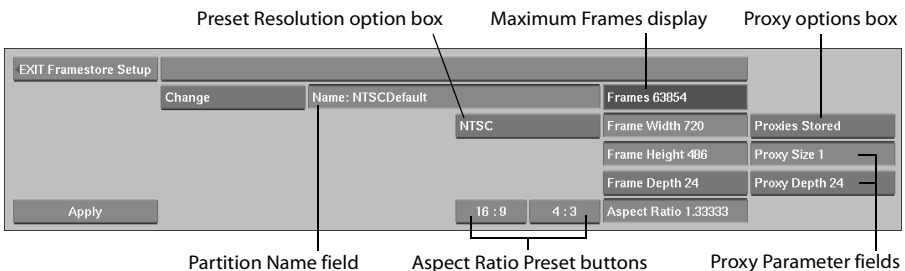
When changing a partition, you must specify partition parameters such as the frame size, proxy size, and aspect ratio of the images in the partition you are modifying or creating. This affects the number of frames that can be stored in the partition.



WARNING: Changing the resolution or frame depth parameters of a partition deletes all the frames it contains.

To change a partition:

1. Select the partition to change.
2. Select Change from the Setup Mode box.



3. To change the name, click the Partition Name field and enter a new name with the on-screen keyboard.
4. Use the Preset Resolution option box to select a set of parameters specific to a certain resolution. You can also change the resolution manually by clicking in the Frame Width and Frame Depth fields. The Resolution box will then display Custom Resolution.
5. Use the parameter fields to modify the partition parameters. You can use the Aspect Ratio Preset buttons to set the aspect ratio to 1.33333 (4:3) or 1.77778 (16:9).
6. Click Apply, then click Confirm. Click elsewhere to cancel.
7. After you finish setting up the framestore, enable the new setup. See “Enabling a New Framestore Setup” on page 95.

Changing the Partition Parameters

Use the appropriate fields in the Framestore Setup menu to specify the parameters for the selected partition. For information regarding image resolution, refer to “Maximum Resolution by Product and Frame Depth (FD)” on page 94.

Partition Name — Use the Partition Name field to enter the name of the new partition or the partition to be modified.

Frames — The Maximum Frames display shows the maximum number of frames that can be stored in the selected partition. When you change a partition parameter such as Frame Depth, Width, Height, or select a new Resolution Preset, this display is automatically adjusted.

Frame Width and Frame Height — Use these fields to specify the size of the images in the partition. The frame width is determined by the number of pixels on each scanline. For example, enter 720 for NTSC and PAL resolution. The frame height is determined by the number of scanlines in each image. For example, enter 486 for NTSC resolution, or 576 for PAL resolution.

Changing the frame width or frame height affects the number of frames that can be stored within the partition. As you change these values, the Maximum Frames display is updated simultaneously.

Proxy Size — Use this field to specify the scaling factor of the image proxies used in real-time playback. For more information, see “Changing the Proxy Size” on page 93.

Aspect Ratio — Use this field to specify the final aspect ratio of the images in the partition.

The final aspect ratio is the aspect ratio of the images as they will be shown on a television or film screen. This aspect ratio may differ from the one **flame** uses to display the images on the computer screen because pixels on the computer screen are square (aspect ratio of 1), whereas pixels in some film and video formats are rectangular (the width differs from the height).

The aspect ratio is expressed as a function of frame width divided by frame height.

Frame Depth and Proxy Depth — Use the Frame field to control the colour depth of your frames. You cannot change the Proxy Depth.

Changing the Frame Depth

The frame depth is the colour depth of your frames, and can be set as follows.

Frame Depth	Description
24	8-bit RGB
36	12-bit RGB packed. Saves I/O bandwidth and disk space, but needs more processing power. This is the recommended setting; however, you should still try each frame depth and compare the results to see which works best with your system.
48	12-bit RGB unpacked. Does not require CPU processing when reading or writing frames, but requires more I/O bandwidth and disk space.

The last two frame depths (36 and 48) store the same information: 12-bits per RGB channel, or 36-bits per pixel. The difference is between disk space, processing power, and I/O bandwidth:

- Processing power is needed to convert images between a packed framestore format and the internal memory format. Images in memory are stored in 48 bits, therefore, the CPU(s) must convert between the packed 36-bits disk format and the unpacked 48-bits in-memory format.
- Disk transfer time takes longer for images with a frame depth of 48, but no conversion is required when reading or writing images.
- Framestores using a frame depth of 48 occupy more disk space per image than framestores with a frame depth of 36.

Changing the Proxy Depth

The proxy depth is the colour depth of your on-disk proxies, and can be set as follows.

Proxy Depth	Description
24	8-bit RGB. This is the recommended proxy depth.
48	12-bit unpacked RGB. Provides full 12-bit per colour component; however, it requires more disk space and will lessen the size of proxies which can be played in real-time.

NOTE: In order to be able to change the proxy depth, you must choose the “Proxies Stored” option for that partition in the Framestore Setup menu.

Changing the Proxy Size

A proxy is a scaled-down version of a frame which is stored on the framestore. Proxies are used to speed up the interaction of the desktop as well as allow real-time playback when it is not possible to do so with the full frames. The Proxy Size control is used to specify the size of the image proxies that will reside in the partition. If you do not need to store proxies on the partition (for instance, if you are working in video resolution), select Proxies Not Stored.

NOTE: Selecting a Proxy size of 1 stores the proxy at the same size as your original.

For high-resolution images, you should select a proxy size that is close to the size of a video image. This will give the best possible image resolution for real-time playback.

You can select 8-bit proxies which have the same size as the 12-bit frames in order to get the largest possible image size for playback from disk. Select a Proxy Depth of 24 and a Proxy Size of 1. This will take up 40% of your framestore space for storing proxies (assuming frames are stored in the 36-bit packed format) and will slow down desktop interaction. For playing back high-resolution images, you may also want to investigate the memory playback options in the Play module.

To calculate the best proxy size:

1. Use the Test Disks command in the System menu to determine the number of MB read per second as well as the number of frames per second (see “The Test Disks Utility” on page 116).

NOTE: To stop the Test Disks utility, press on the tablet after letting the utility run for a minute or so.

2. Use the following equation to calculate the best proxy size.

Proxy Size =

$$\sqrt{\frac{\text{Test Disk MB per second} * 1048576}{\text{Frame Height} \times \text{Frame Width} \times \text{Frame Depth} \times \text{Desired playback in FPS speed}}}$$

Where:	Is:
Test Disk MB per second	The number of MB read per second from the Test Disk utility.
Frame Height	The Frame Height value from the Framestore Setup menu.
Frame Width	The Frame Width value from the Framestore Setup menu.
Frame Depth	The Frame Depth in bytes. For 8-bit proxies (Proxy Depth 24), this value is 3 bytes per pixel. For 12-bit proxies (Proxy Depth 48), this value is 6 bytes per pixel. For more information about frame depths in flame , see “Changing the Frame Depth” on page 92.
Desired playback speed in FPS	The desired playback speed in frames per second.

FILM and DTV Parameters

In addition to the NTSC and PAL default buttons, there are various other preset parameter settings for film and DTV (Digital Television) resolutions. Scroll through the Preset Resolution selection box to determine which parameter settings are the most appropriate for your work. The settings will appear in their respective fields, but will not take effect on your partition until you click the Apply button.

Maximum Resolution by Product and Frame Depth (FD)

The following table outlines the maximum image resolution possible on the current suite of Discreet Effects products.

Product	8-bit (FD 24)	12-bit (FD 36/48)
inferno 4.0	Double the 12-bit maximum in one dimension (such as 6224 x 4096)	4096 x 3112

Product	8-bit (FD 24)	12-bit (FD 36/48)
flame 7.0	2048 x 2048	2048 x 1744
effect 7.0	2048 x 2048	N/A
flint 7.0	2048 x 2048	N/A

NOTE: These values represent upper limits. You must have sufficient RAM to achieve these limits.

Things to keep in mind:

- An important factor is the number of pixels. For example, a frame with a resolution of 2048 x 1744 has about 3.5 million pixels, so that system could also support a custom resolution such as 1000 x 3500.
- 12-bit pixels consume twice as much memory as 8-bit pixels.
- Image width must be a multiple of 4. There are no such restrictions on image height.

Enabling a New Framestore Setup

Modifications to the partition structure of your framestore take effect only after you exit and restart **flame**. Once you start **flame** again, you must specify a project that will use the new or modified partition.

To enable a new setup:

1. After you are finished setting up the framestore, click Apply, then click Confirm.
2. Exit **flame**.
3. To make use of a new partition, restart **flame**, and specify a project which uses your new or modified partition.

For more information on creating projects, see “Creating a Project” on page 25.

This image shows a full page of white paper with horizontal dashed lines. The lines are evenly spaced and run across the width of the page, providing a guide for handwriting practice. There are no margins, text, or other markings on the page.

*Hot key short cuts exist for many of the functions within **flame**. The Hot Key Editor allows you to view listings of the current hot keys for these functions, and to modify and create your own custom hot keys.*

Summary

In this chapter, you learn about:

- “Editing Local Hot Keys” on page 100
- “Keystroke Conventions” on page 99
- “Creating New Local Hot Keys” on page 100
- “Managing Your Hot Keys” on page 101
- “Editing Global and Shared Hot Keys” on page 102
- “JLCooper MCS-3800 Interface” on page 102

About the Hot Key Editor

You can use the Hot Key Editor to view, modify, and create hot keys. If you are using an optional JLCooper MCS-3800™ media command station, the Hot Key Editor also lets you map buttons and functions to the controls on this device (see “JLCooper MCS-3800 Interface” on page 102 for more information).

flame 7.0 features a Hot Key Editor, which you can use to customize the hot key configurations on the desktop and within the modules.

To access the Hot Key Editor:

- Click the Hot Keys button in the System menu
or
press **F8**. (This is a toggle—if you enter the Hot Key Editor by pressing **F8**, you must also press **F8** to exit the Hot Key Editor.

NOTE: Hot key lists appear at the end of many of the chapters in this user’s guide.

Determining a Button’s Current Hot Key

Use this procedure to find out which hot key has been assigned to a button (if any).

To determine the hot key for a button from anywhere within the application:

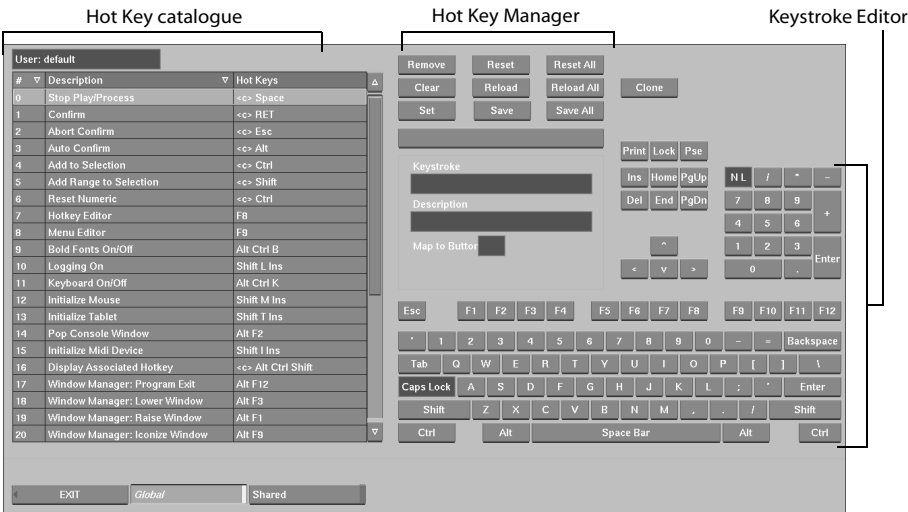
1. Press and hold **SHIFT-CTRL-ALT**.
2. Click the button for which you want to determine the assigned hot key. Current hot key messages that begin with “MIDI” refer to controls on the optional JLCoooper MIDI device.
- The current hot key message appears in the status area.

Current hot key message



Hot Key Editor

Use the Hot Key Editor to remap default hot keys and to create new ones. There are two types of hot keys, *Map-to-Button* hot keys and *functional* hot keys. To access the Hot Key Editor, press **F8**.



Map-to-Button Hot Keys

Map-to-Button hot keys are keystroke sequences that are mapped to a button or a field on the current menu. You can create and edit as many Map-to-Button hot keys as you like.

If you are using a JLCoper command station, you can map controls on this device to buttons or fields instead of keystroke sequences. You can even map a button or field to your system keyboard and the command station using the Clone feature (see “Clone” on page 101).

Functional Hot Keys

Functional hot keys are predefined hot keys for specific functions. You cannot create new functional hot keys, but you can redefine and remap the ones that are provided with the system. You can map functional hot keys to the controls on a JLCoper command station.

Hot Key Domains

There are three hot key domains: global, shared, and local.

Domain:	Used for:
Local	Hot keys that are available only within a specific component.
Shared	Hot keys that are available from within several components.
Global	Hot keys that are available from anywhere within the system.

Keystroke Conventions

There are a number of conventions you should be aware of when creating and editing hot keys.

- You are limited to four keystrokes.
- You cannot use the same keystroke twice in succession in the same keystroke sequence (e.g., you cannot use something like “TT”).

Logical Ordering

Logical ordering means that certain keystrokes are forced to the beginning or end of the sequence, to conform to conventions. Normally, the keystrokes are entered in the same sequence in which you choose them in the Keystroke Editor, but keys such as **CTRL**, **SHIFT**, and **ALT** are forced into conventional order:

- **ALT**, **CTRL**, **SHIFT** are always ordered in that sequence, regardless of the sequence in which you enter them.
- **ALT**, **CTRL**, **SHIFT** are always forced to the beginning of a keystroke sequence.
- **DEL**, **INS**, **HOME**, **PGUP**, **END**, **PGDN** are always forced to the end of a keystroke sequence.

Editing Local Hot Keys

When a menu is open, enter the Hot Key Editor by pressing **F8**. The Hot Key catalogue that appears shows the global, shared, and local hot keys for the current menu. Virtually every menu has an editable local hot key catalogue.

To edit the hot keys for a different menu, exit the Hot Key Editor (by pressing **F8** again), go to the menu where you want to edit hot keys, and bring up the Hot Key Editor from there.

To edit a local hot key:

1. Select the hot key in the hot key catalogue.
The keystroke sequence and its description appear in the Keystroke Editor fields. If you select a Map-to-Button hot key, the word “Yes” appears in the Map-to-Button field.
2. Click Clear in the Hot Key Manager area to clear the existing keystroke sequence.
3. Enter the new keystroke sequence by clicking keys in the Keystroke Editor or by pressing keys on your computer’s keyboard. If you are using a JLCopier command station, you can also press a key on the JLCopier instead of a keystroke from your system keyboard.
4. Click Set in the Hot Key Manager area of the Hot Key Editor.
5. Click Save to save the changes to the current user catalogue.

NOTE: If the keystroke sequence you enter is already in use, the change is not made and an error message is displayed. To determine which hot key is associated to a particular button (anywhere in the software), press CTRL-ALT-SHIFT and click on the button. The status bar will show whether a hot key is associated, and if so, show it.

Creating New Local Hot Keys

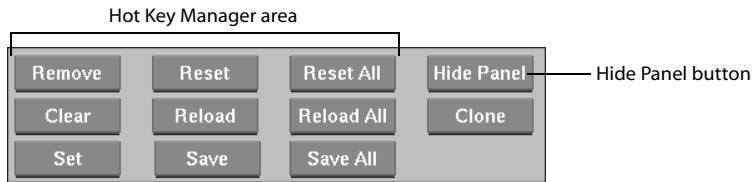
You can only create new Map-to-Button hot keys. You cannot create new functional hot keys.

To create a new Map-to-Button hot key:

1. With the Hot Key Editor open, click the menu button for which you want to create the hot key.
A description field appears in the Keystroke Editor.
2. Enter the new keystroke sequence by clicking keys in the Keystroke Editor (or your computer’s keyboard).
3. Click Set in the Hot Key Manager area of the Hot Key Editor.
4. Click Save to save the changes to the current user catalogue.

NOTE: Use the Hide Panel button to temporarily hide the Hot Key Editor panel if you want to select a menu button that is otherwise hidden by the panel. The panel returns after you select

a menu button. Alternatively, click in an open area to return to the Hot Key Editor panel without selecting anything.



Managing Your Hot Keys

The Hot Key Manager area of the Hot Key Editor contains items for managing the Hot Key catalogues.

Remove — Removes a hot key from the current Hot Key catalogue. Select the hot key in the catalogue and click the Remove button. You are not prompted for a confirmation. You must use Save to make the change permanent.

Clear — Clears the contents of the Keystroke field in the Keystroke Editor before you enter a new keystroke sequence.

Set — Is the second-to-final step when assigning a keystroke sequence to a button or function. You must use Save as the final step to make the change permanent. For more information, see “Editing Local Hot Keys” on page 100 and “Creating New Local Hot Keys” on page 100.

Reset — Resets the current catalogue’s editable hot keys (the ones in black) to their default settings.

Reset All — Resets all hot keys to their default settings.

Reload — Reloads the current catalogue of hot keys. This is useful if you have made a change but have not yet saved it, and you wish to discard the change.

Reload All — Reloads all hot keys from the specified user catalogue.

Save — Saves the current catalogue’s editable hot keys (the ones in black) to the specified user catalogue.

Save All — Saves all hot keys to the specified user catalogue.

Clone — Lets you assign a regular keyboard keystroke sequence and a JLCoooper control to a single button, field, or function. Although Clone is primarily intended for use with JLCoooper MIDI devices, you can also use it to map multiple keystroke sequences to a single button, field

or function using the regular system keyboard. This feature does not provide macro functionality.

To clone a button, field, or function:

1. Select a button, field or function in the Hot Key catalogue.
2. Click Clone in the Hot Key Manager area to create a second entry for this button, field or function.



3. Activate a control or enter a keystroke sequence on the JLCopper command station (or the regular system keyboard).
4. Click Set in the Hot Key Manager area of the Hot Key Editor.
5. Click Save to save the changes to the current user catalogue.

Editing Global and Shared Hot Keys

To edit global and shared hot keys, enter the Hot Key Editor through the System menu.

The Global/Shared Hot Key Editor looks and functions the same as the Local Hot Key Editor. However, the Global/Shared Hot Key Editor contains two buttons that allow you to view and edit either global or shared hot keys (you cannot view and edit both at the same time).

When you click Global, the global hot keys appear in the Hot Key catalogue. Similarly, when you click Shared, the shared hot keys appear in the Hot Key catalogue.

Global and shared hot keys can also be edited in the modules or wherever the Hot Key editor is available. You are warned that a global/shared hot key change affects all modules, and asked to confirm the change.

JLCooper MCS-3800 Interface

If you are using an optional JLCooper MCS-3800™ media command station, the Hot Key Editor lets you map a button or a function to the controls on this device.

To create or edit a hot key assigned to a control on the JLCooper MIDI device, follow the regular hot key assignment procedure and press a single key, keystroke sequence, or activate a fader or knob on the JLCooper, instead of the regular system keyboard.

Many JLCooper controls are mapped automatically to buttons, fields, and audio faders. You can edit these default mappings to suit your needs.

Fader and Knobs

Faders and knobs require that you set the Value Type using a pop-up menu. This setting determines the behavior of the knob or fader according to the type of button, field, or function being assigned to the control.

JLCooper Controls

The command station includes the following controls.

Section	Name
Mixer	<ul style="list-style-type: none"> • Motorized, touch-sensitive faders • Fader buttons • Rotary encoders • Page and Bank switches
Function Buttons	<ul style="list-style-type: none"> • F1 to F8 function buttons • Shift button • W1 to W5 buttons • M1 to M5 buttons • Cursor buttons
Transport	<ul style="list-style-type: none"> • Rewind • Fast Forward • Stop • Play • Record • Jog/Shuttle mechanism • Numerical keypad
System	<ul style="list-style-type: none"> • Cursor buttons • Rotary encoders • Assign button • LCD display • LED display

This image shows a full page of white paper with horizontal dashed lines. The lines are evenly spaced and run across the width of the page, providing a guide for handwriting practice. There are no margins, text, or other markings on the page.

Saving Setups and Preferences

Play it again, Sam

A setup is a file containing all the changes made to a clip within a module. A

preference is a setting made to customize the way a module works. You can save

setups and sets of preferences, then retrieve and use them in a later work session.

Summary

In this chapter, you learn about:

- “Using Directories” on page 106
- “Setting the Default Preferences” on page 111
- “Saving, Loading, and Removing Items” on page 107
- “Setup Compatibility between Discreet Products” on page 112
- “Restoring a Backup Copy of a Setup” on page 111

About Setups and Preferences

You can save setups in all of **flame**’s modules and preferences in most modules. You can also archive setups; for information, see “Archiving Setups” on page 375.

Setups

A setup is a file that contains a record of all changes you make to a clip in a particular module. This record includes references to clips used. Setups let you save your work separately from the clips, so you can load and work on the setup anytime, or apply the setup to other clips. You can create a number of different setups during a single work session, including text setups, key setups, custom filters, and images. For example, when you save an Action setup, all menu parameters, layers, clip references—everything you need to rebuild the scene—are saved in the setup.

By default, each type of setup is stored in a separate directory. For example, the *key* directory stores key setups, the *picture* directory in Paint stores single images, and the *correct* directory stores colour correction setups.

Preferences

Preferences are settings that let you customize the display or functioning of some module elements, hot keys, the desktop, pen and tablet, and audio. You can save preferences to a file and retrieve them at a later time. You set module preferences in the Setup menu of each module, hot key preferences in the Hot Key Editor, and system-level preferences in the Preferences menu. Examples of preferences are the Auto Key setting in many modules and the Texture option in the Action Setup menu. Like setups, each type of preference is stored in a separate directory.

System-level preferences (desktop, pen and tablet, and audio) are saved for each user when you exit **flame**, and are retained for future **flame** sessions. Module preferences are auto-saved when you exit the module, and retained even if you exit **flame**. The module preference auto-save file is retained until it is replaced with another or if the user is changed.

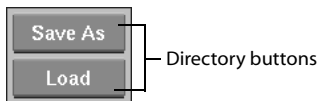
Using Directories

When you save a setup or preference in a directory, it is saved as a file in a directory in the UNIX file system. Each file and subdirectory in the file system has a pathname—a series of parent directory names separated by slashes (/)—that UNIX uses to locate the file or subdirectory. Each type of setup and preference has a different default directory, and therefore a different pathname.

The default directory is the one accessed by default when you save or load setups and preferences. You specify the default location for your setup directories when you create your project, and the default location for your preference directories when you create a new user. For more information on creating projects and users, see “Project Management” on page 23. The default pathnames for setups are listed in the Environment Directory Pathnames section of the project configuration file.

Accessing a Directory

All modules contain two directory buttons—a Save As button and a Load button, found in the module’s Setup menu.



Use these two buttons to save, load, and remove setups and preferences from the directories. For example, in the Colour Corrector Setup menu, you can use the Save As button to save setups to the Colour Corrector setup directory and the Load button to load a previously saved setup.



When you open a directory, all the setups or preferences of the item type selected in the Load option box appear in the file browser. Use the file browser to select the setup or preference you want to load.

NOTE: Only items of the current item type are displayed in the directory. For example, only Text setups are displayed when the Text directory is open, even if other files are stored in that directory.

Saving, Loading, and Removing Items

This section describes how to save, load, and remove items from directories.

When you load setups that were created in a different resolution, they can be scaled to conform to the resolution of the current partition. For information, see “Automatic Scaling of Setups” on page 120.

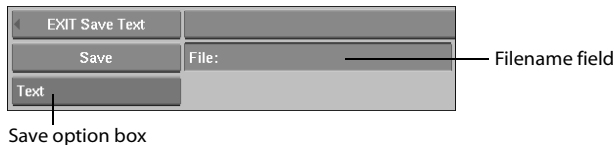
NOTE: The Paint module has additional features you can use when saving setups. See “Paint: Creating, Loading, and Saving Setups” on page 1157.

To save an item to a directory:

1. Click the Save As button in the Setup menu of a module. (In the Paint module, the Save As button is in the Library menu.)

The file browser appears in the top part of the screen and the Save menu appears in the lower area. The file browser points to the default directory for the item type currently selected in the Save option box. (If there is no Save option box, you can only save one type of item, a setup file for the module.)

Save menu



2. If there is a Save option box, select from it the type of item you want to save. (If there is no Save option box, you can only save one type of item, the setup file for the module.)
3. In the Filename field, enter a name for the item you are saving. Press the **ENTER** key. When re-saving an item, simply click the Save button.

The item is saved and you are returned to the previous menu.

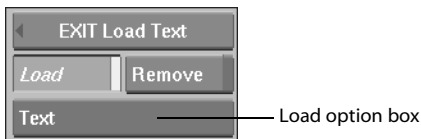
NOTE: If the directory already contains an item with the same name, a warning message appears in the message bar, and a Confirm button. Click the Confirm button to overwrite the file, or click elsewhere on the desktop to cancel the command.

To load an item from a directory:

1. Click the Load button in the Setup menu of a module.

The file browser appears in the top part of the screen, and the Load menu appears in the lower left corner. The file browser points to the default directory for the option currently selected in the Load option box.

Load menu



2. If there is a Load option box, select from it the type of item you want to load. (If there is no Load option box, you can only load one type of item, a setup file for the module.) The directory for the selected option appears.
3. If the setup you want to load is located in a different directory, use the file browser to change directories. To load a setup from a different project, select the project from the Project box.

For more information on using the file browser, see “The File Browser” on page 61.

4. Select the title (or proxy) of the item you want to load.

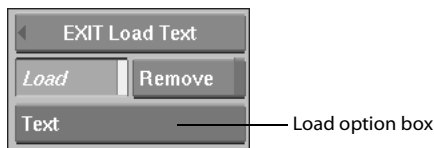
The item is loaded, and you are returned to the previous menu.

To remove an item from a directory:

1. Click the Load button in the Setup menu.

The file browser appears in the top part of the screen, and the Load menu appears in the lower left corner. The file browser points to the default directory for the option currently selected in the Load option box.

Load menu



2. From the Load option box, select the type of item you want to remove.

The directory for the selected item type appears.

3. If the setup you want to remove is located in a different directory, use the file browser to change directories. To remove a setup from a different project, select the project from the Project box.

For more information on using the file browser, see “The File Browser” on page 61.

4. Click the Remove button.
5. In the file browser, select the title (or proxy) of the item you want to remove.

A Confirm button appears in the menu.

6. Click the Confirm button to confirm your selection. The selected item is removed from the directory and you are returned to the previous menu. To cancel the command, click elsewhere on the desktop.

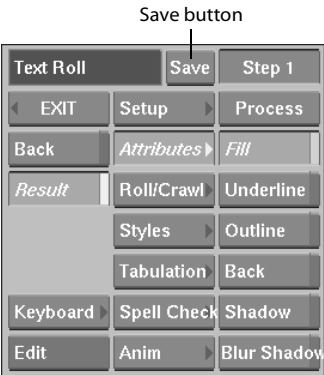
HINT: To bypass the Confirm button when saving items, press Alt-Enter to save. When removing items, press Alt as you select an item for removal.

Removing Multiple Items

To remove more than one item at a time, press and hold the **CTRL** key when you click the Confirm button. The selected item is deleted, and you remain in the Load menu. Click the Exit Load button to return to the previous menu.

Re-saving Setups the Fast Way

Use the Save button to quickly re-save a setup.



When you first start a module work session, the Save button is greyed out. Before using this button, you must name and save the setup as described in “To save an item to a directory:” on page 108, or load a previously saved setup (see “To load an item from a directory:” on page 108).

To perform a quick re-save:

- 1. Name and save the setup using the Save As button, or load an existing setup.
- 2. After making your changes, click the Save button.

The Confirm button and a message appear. If the setup is from the current project, the message asks you to confirm that you want to overwrite the previously saved setup. If the setup is from another project, the message asks you to confirm saving the setup *to the current project directory* (rather than overwriting the setup in the other project). This prevents the possibility of inadvertently overwriting another project’s setup.

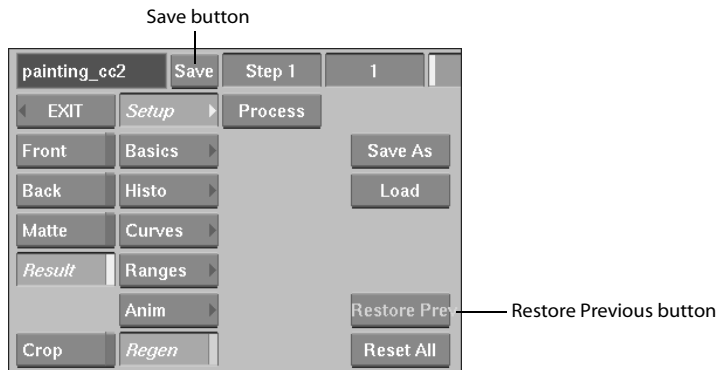
NOTE: If you want to save a setup to another project, use the Save As button and select the project from the Project box. After this, the Save button will save the setup to the project you specified.

- 3. Click the Confirm button to save the setup, or click elsewhere to cancel.

HINT: To override Confirm, press Alt and click the Save button.

Restoring a Backup Copy of a Setup

When you save a setup with the Save button, the new version overwrites the old version, and the old version is copied to a backup file. You can use the Restore Previous button to restore the backed up version.



To restore a backed up setup:

1. Save the setup at least once using the Save button (see “To perform a quick re-save:” on page 110).
2. Click the Restore Prev button.

The version saved prior to the last saved version is restored.

NOTE: To restore the last saved version, simply load it using the Load button.

Setting the Default Preferences

In any module where preferences can be saved, you can save your preferences as the default, so that they will be loaded each time you use the module. Each defined user can save their own default preferences. You can also return the default preferences to those that came with **flame**.

To save your preferences as the default:

1. Set the preferences the way you want.
2. Open the Save menu.
3. Select Defaults from the Save option box. If the Confirm prompt appears, confirm the save.
The defaults are saved and you are returned to the previous menu.

To reset the defaults to the factory setting:

1. Open the Load menu.
2. Select Factory Defaults from the Load option box and Confirm.

The factory defaults are loaded, and you are returned to the previous menu.

NOTE: When you load the factory defaults, the current default preferences are saved to a backup file that you can re-load from the user's preference directory if needed.

Setup Compatibility between Discreet Products

Some setups created in other Discreet products can be loaded into **flame**, and vice versa.

Compatibility between Effects and Editing Products

The following setups created in 3.x and 4.0 Editing products can be loaded into 3.x/6.x and 4.0/7.0 Effects products and vice versa:

- Keyer (but not garbage mask setups)
- Text
- Paint
- Stabilizer
- Filter
- Optics
- Regrain/Degrain
- Colour Correction

NOTE: Colour Correction setups are compatible between Effects and Editing products only as of version 4.0/7.0.

Compatibility with Combustion

The following setups created in **combustion**[™] can be loaded into Effects and Editing products and vice versa:

- Colour Correction
- Stabilizer
- Keyer

To copy the setups directly to the **flame** setup directories, the currently logged user must have write access.

NOTE: Garbage mask setups are not transferable between **combustion** and Effects/Editing products. Also, certain parameters in **combustion** setups may not transfer to Effects/Editing products. For details, refer to the *combustion User's Guide*.

System Utilities and Preferences

The way you want it

Use the System menu to set up and recover framestores, access the Hot Key Editor, work with monitor calibration and project management, and access the Preferences menu.

Summary

In this chapter, you learn about:

- “The System Menu” on page 113
- “The Preferences Menu” on page 118
- “Desktop Preferences” on page 118
- “Pointer Preferences” on page 124
- “Using Custom Menus” on page 125

For information on audio preferences, see Chapter 24, “Using Audio.”

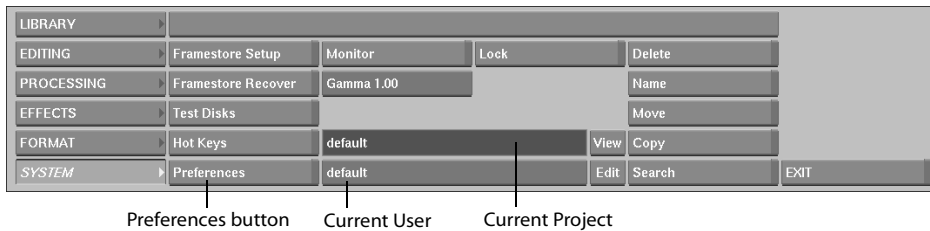
The System Menu

This section describes how to use most of the items in the System menu. Some of the items are discussed in other chapters of this user’s guide:

- For information on framestore setup, see Chapter 5, “Framestore Setup.”
- For information on monitor calibration, see Chapter 10, “Monitor Calibration.”
- For information on project management, see Chapter 3, “Project Management.”
- For information on the Hot Key Editor, see Chapter 6, “Hot Keys.”

Access the System menu from the Main menu.

System Menu



Current Project and User

The names of the current project and user appear in the System menu. You can:

- View information on the current project by clicking the View button.
- Change to a different user by selecting one from the User box.
- Define new users by selecting <New> from the User box.
- Modify user settings. Click the Edit button to modify the name or set a different directory for the user's preferences.

For more information, see Chapter 3, "Project Management."

Locking and Unlocking a Clip

You can lock a clip to prevent it from being deleted accidentally.

To lock a clip:

1. Click the Lock button in the System menu.
2. Select the clip.

The frame numbers (or timecode) on the locked clip appear in red. You will not be able to delete or cut this clip until it is unlocked.

To unlock a clip:

1. Click the Lock button in the System menu.
2. Select the locked clip.

The frame numbers (or timecode) on the clip appear in white. The clip can now be deleted or cut using the Delete or Cut command.

Recovering Clips from the Framestore

The Framestore Recover command reads all the frames on a partition and creates one large clip containing all the frames that it finds. This command can be useful if:

- You have a system disk failure and after replacing the disk you want to recover clips on the framestore.
- You accidentally delete a clip library and want to recover the clips.

Once you have the single clip, you have to identify the frames within it that you want to keep and manually separate them into individual clips using the Cut command. If effects were applied to the clips and were not processed, and you want to reapply them (given that you had backed up the setups), you must reapply them manually and then reprocess them, which can be time-consuming. Additionally, editing information such as timewarps, cuts, and dissolves are not retained, and must be reapplied. For these reasons, it is recommended to recapture your clips if you have an EDL, and only use this command as a last resort.

NOTE: If you start **flame** using the **-v** option, the partition table is erased, and you cannot recover the clips with Framestore Recover.

To recover clips after a system disk failure:

1. After installing your new system disk and installing **flame**, start **flame** and create a new project. The partitions in the framestore appear in the Framestore Setup menu. Select the partition containing the clips you want to recover.
2. Click Framestore Recover in the System menu.

The Recover Mode box appears.



3. Select an option from the Recover Mode box.

Select:	To:
Creation Order	Restore "linked" frames (frames still referenced on the framestore) and sort them according to creation order. This is the recommended option.
Disk Order	Restore "linked" frames in the order in which they appear in the framestore's frame table. This is not true disk order, but is the closest possible with the information available at this level.

Select:	To:
All	Restore all "linked" and "unlinked" frames (frames that are not referenced but are still valid on the framestore).
Deleted	Restore only "unlinked" frames.

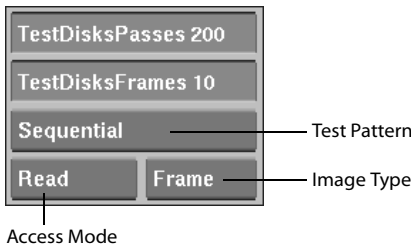
- Select a destination.
A single clip, comprised of all recovered frames in specified order, appears on destination reel.

The Test Disks Utility

Use the Test Disks utility to verify the performance of the disks on the framestore.

To verify disk performance:

- Click the Test Disks button.
Options appear to the right of the System menu.



- Create a profile for the test by setting the following options.

Use:	To Set:
Test Disk Passes	The number of times the test will access the disks.
Test Disk Frames	The number of images that will be accessed for each disk pass.
Test Pattern	The method used to access images. Choices are: <ul style="list-style-type: none"> • Sequential: Adjacent images are accessed (one image after another). • Butterfly: Access simulates a dissolve (Alternately accesses two different places on the framestore). • Random: Images are accessed randomly.
Access Mode	The mode of access: Read or Write.
Image Type	The type of image accessed: Frame or Proxy.

- Click in the desktop area.
The raw performance of the disk array appears in the message bar. The display includes:

- The number of frames read per second
- The number of seconds it takes to read one frame
- The number of MB read per second

For example:

```
TEST DISKS: Sequential Frame Read.289.33 fps 0.14 spf. 289.16
MBps.
```

NOTE: To abort the process, press on the tablet or click the mouse.

Regenerating Clips

The Regenerate button only appears when proxies are stored on the current partition. Use the Regenerate command to regenerate proxies on the desktop and in clip libraries. You only need to regenerate proxies if one or more of the following proxy parameters have changed:

- The field dominance (see “The Field Dominance Box” on page 622)
- The field display (see “Interlace Display Options” on page 122)

NOTE: You do not have to regenerate clips when you turn Proxy Aspect on or off, as this only affects the display of proxies, not how they are generated.

To regenerate a clip:

1. After changing a field parameter, click the Regenerate button in the System menu.
2. Select the source clip.

The selected clip is regenerated.

For more information on storing proxies, see Chapter 5, “Framestore Setup.”

Switching Display LUTs

When you start **flame**, the Display LUT is read from the configuration file. The Display LUT can be a System Gamma value, Custom LUT, or monitor calibration file. You can change Display LUTs from **flame**.

To change the Display LUT, click the Gamma box. The options that appear are those that appear in the *init.cfg* configuration file under the Gamma keyword. See “Monitor LUTs” on page 182.

You can change the Display LUT from anywhere in **flame** using hot keys. The first three Display LUTs in the *init.cfg* configuration file are assigned to the following hot keys:

Use this hot key:	To use:
CTRL+SHIFT+1	The first Display LUT defined in the configuration file.
CTRL+SHIFT+2	The second Display LUT defined in the configuration file.

Use this hot key: **To use:**
CTRL+SHIFT+3 The third Display LUT defined in the configuration file.

NOTE: You can define more than three Display LUTs in the *init.cfg* configuration file, but only the first three are assigned to the hot keys. To access all available Display LUTs, use the Gamma box.

Exit

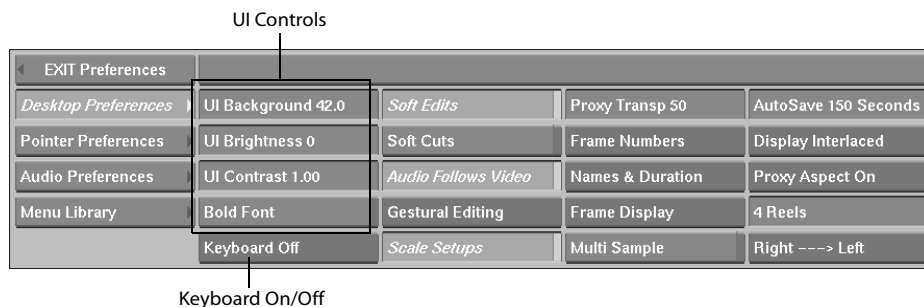
Use the Exit command to exit from the current work session and save the current status of the reels and menu setups.

The Preferences Menu

The Preferences menu contains menus for desktop, pointer, and audio preferences, as well as the Menu Library.

Desktop Preferences

Desktop preferences affect how your desktop looks and how you interact with it.



Changing UI Settings

Use the UI controls to adjust how the user interface appears.

Use:	To:
UI Background	Adjust the background brightness. 0 = black and 100 = white. The default value is 22. A darker background is suggested for film work, and a lighter background is suggested for video work.
UI Brightness	Adjust the brightness of interface elements such as buttons and fields. The default value is 0.
UI Contrast	Adjust the contrast of interface elements such as buttons and fields. The default value is 1.0.
Bold Font	Toggle bold font for text on buttons on and off.

NOTE: The image window, image proxies, and background are not affected by the UI Brightness and UI Contrast settings.

Setting the Keyboard On or Off

When you enter text in a field, you can opt to have the on-screen keyboard appear or just have editing access to the field. Set the Keyboard button to On or Off.

HINT: You can also turn the keyboard on or off from anywhere in **flame** with the hot key **CTRL-ALT-K**.

Enabling Soft Edits

Enable the Soft Edits button to enable the soft edit features of **flame**. When the Soft Edits button is disabled, heads and tails are not retained when editing, and the EditReel is not available in the Player, unless a previously made soft clip exists.

For more information, see “Soft Edits” on page 389 and “The EditReel” on page 386.

Soft Cuts

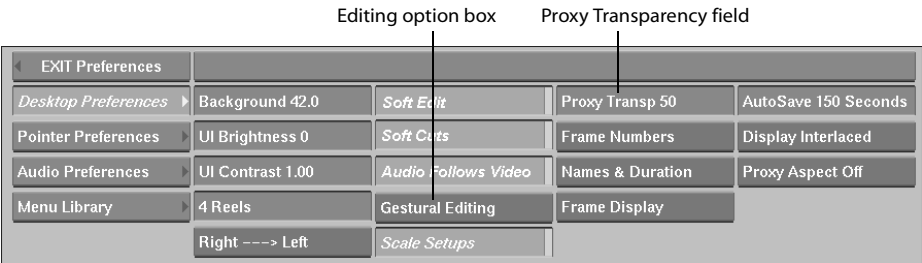
See “Soft Cuts” on page 389 for a description of this option.

Setting the Audio Option for Gestural Editing

Use the Audio Follows Video button to specify how audio is edited when you gesturally edit clips. When the Audio Follows Video button is enabled, gestural edits such as cuts and splices are applied to both video and audio tracks. When it is disabled, gestural edits such as cuts and splices are applied to video elements only. Many gestural editing options are affected by the status of this button. For more information, see “Editing Video and Audio Simultaneously” on page 383.

Gestural Editing Options

To set the mode for editing clips, select an option from the Editing option box.



Select:	To:
Drag & Drop	Enable drag and drop operations only (moving, copying, and deleting clips).
Gestural Editing	Enable drag and drop operations and gestural editing operations (cutting, copying, insert edits, replace edits).
Gest. Editing Off	Disable gestural editing and drag and drop operations.

NOTE: Drag and drop operations are available in clip libraries even when gestural editing is disabled.

Adjusting the Proxy Transparency

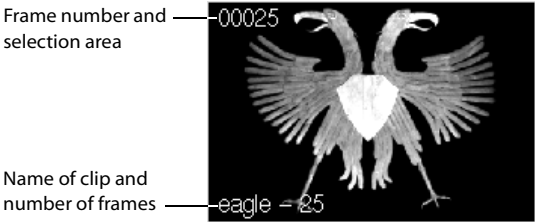
Use the Proxy Transparency field to adjust the level of transparency on proxies during drag and drop operations on the desktop and in the clip library. A setting of 0 means the proxy has no transparency when it is being dragged. A setting of 100 means the proxy is completely transparent when it is being dragged. The default setting is 50.

Automatic Scaling of Setups

You can load setups (such as animation setups, keyer setups, and colour correction setups) that were created in a different resolution from that of your current project. To work correctly in your current project, the setups must be scaled when they are loaded. To scale setups when loading them, enable the Scale Setups button.

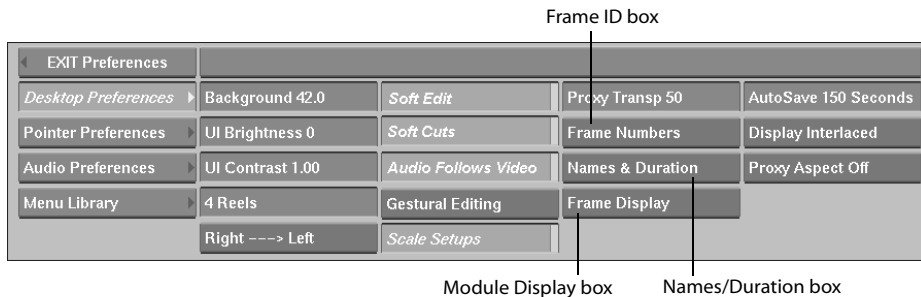
Displaying Timecodes, Frame Numbers, and Names

The frames in a clip are numbered sequentially using either frame numbers or timecodes. These identification numbers appear in the upper-left corner of the frame. By default, the first frame in a clip has a frame number of 00001 or timecode of 00:00:00:00. Timecodes are always read in hours, minutes, seconds, and frames.



Timecodes and Frame Numbers

Use the Frame ID box to select the frame identification mode for all clips on the desktop reels.



Select:	To:
Src and Rec TC	Display the source and record timecodes.
Record TC	Display the record timecode and frame numbers.
Source TC	Display the source timecode and frame numbers.
Frame Numbers	Display the frame number in the clip.
No Frame Display	Disable the display of both timecodes and frame numbers.

The source timecode (on the left) is the timecode on a source clip that you load from a VTR or create in **flame**. For more information, see “Editing Timecodes” on page 390.

NOTE: You can change the timecode of a clip using the Change Timecode command in the Format menu. For more information, see “Changing Timecode” on page 617.

Displaying Frame Numbers or Timecodes in Modules

Use the Module Display box to display either frame numbers or timecodes in the image window play controls in the Processing and Effects modules.

Clip Names and Duration

Use the Names/Duration box to enable or disable the display of clip names or clip durations for all clips.

Select:	To:
Names & Duration	Display both the clip name and the duration for each clip.
Duration	Display the clip durations only.
Names	Display the clip names only.
No Name Display	Disable the display of clip names and clip durations.

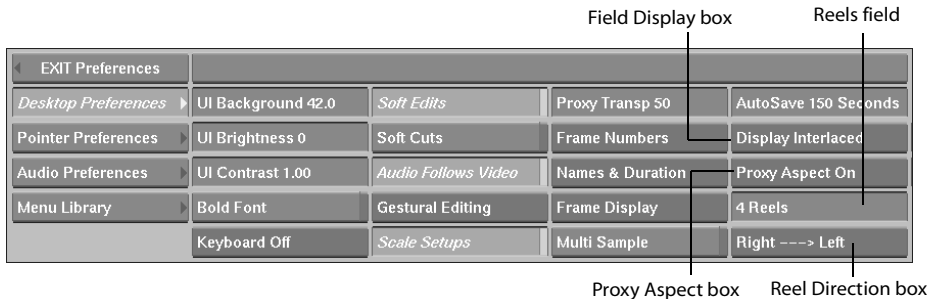
You can change the name of a clip using the Name command. For more information, see Chapter 20, “Basic Editing.”

Setting the AutoSave Timing

Use the AutoSave field to specify how often **flame** automatically backs up the desktop reels. A message briefly appears in the desktop message bar and the cursor changes to the wait cursor during an autosave.

Interlace Display Options

Use the Field Display box to select the field displayed for all clips.



Select:

Display First Field

Display Second Field

Display Interlaced

Display NonInterlaced

To Display:

The first field only for all clips.

The second field only for all clips.

Both fields as interlaced fields for all clips.

Both fields as non-interlaced fields for all clips.

If the frame proxy size has the maximum value of 1, all clips on the desktop are automatically updated when you change the status of the Field Display box.

If the frame proxy size is less than 1, you can use the Regenerate command to regenerate selected clips according to the current status of the Field Display box. For more information, see “Regenerating Clips” on page 117.

For more information on setting the proxy size, see Chapter 5, “Framestore Setup.”

Proxy Aspect Display Options

You can use the Proxy Aspect box to display the clips on the desktop according to either of the following:

- Their actual frame aspect ratio.
- The aspect ratio that you specify using the Aspect Ratio box in the Framestore Setup menu.
For more information about using the Aspect Ratio box, see Chapter 5, “Framestore Setup.”

Select:	To Display:
Proxy Aspect On	Clips according to the aspect ratio specified in the Aspect Ratio box of the Framestore Setup menu.
Proxy Aspect Off	Clips according to their actual frame size as specified in the Frame Height and Frame Width fields in the Framestore Setup menu.

Changing the Number of Reels on the Desktop

Use the Reels field to change the number of reels that are visible on the desktop. You can choose to view between four and eight reels. The reels automatically scale to fit the desktop as you change the number of reels.

NOTE: If you reduce the number of reels that are visible on the desktop when there are clips on all reels, the clips remain on the hidden reels. If you save the desktop, all reels including the hidden ones are saved.

Changing the Orientation and Direction of Reels

Use the Reel Direction box to change the orientation (horizontal or vertical) and direction of the reels on the desktop. The following options are available:

- Right ---> Left
- Left ---> Right
- Bottom ---> Top
- Top ---> Bottom

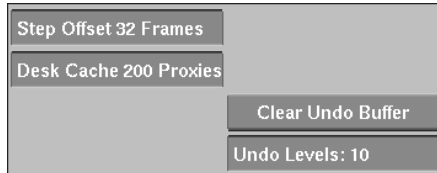
NOTE: The “traditional” **flame** orientation and direction is Right ---> Left.

Step Offset

Use Step Offset to set an offset value, in number of frames, for moving locked reels to the left or right. You move the clips on locked reels by the specified offset by pressing the **LEFT ARROW** and **RIGHT ARROW** hot keys (it does not matter where the cursor is located). For details on locking reels, see “Locking Reels Together” on page 44.

Desk Cache

Use this field to set the number of proxies you want to be able to store in RAM. Higher values improve desktop performance since it reduces the need to access the proxies on the framestore.



Clear Undo Buffer

Use the Clear Undo Buffer to clear the undo buffer for desktop operations. Clearing the undo buffer can make more room on the framestore. For example, if the Undo Levels value is set at 50 and you have deleted 50 clips from the desktop, the space on the framestore does not increase because the clips are still stored in the undo buffer. Click Clear Undo Buffer to clear the 50 clips from the undo buffer and free up the space on the framestore.

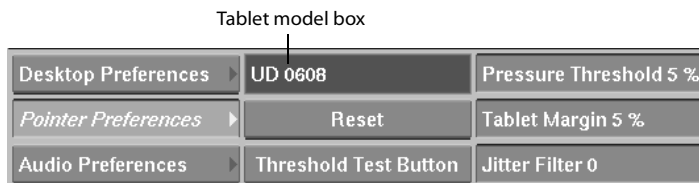
NOTE: Once you use Clear Undo Buffer, you cannot undo any of your previous actions.

Undo Levels

Use the Undo Levels field to specify how many levels (0-50) of information **flame** saves to the desktop undo buffer. This value determines the maximum number of actions you can undo when working on the desktop. For example, if the Undo Levels value is set at 5 and you delete six clips from the desktop, you can click Undo once for every clip you want to restore to the desktop, up to the fifth one. The sixth deleted file cannot be restored.

Pointer Preferences

You can adjust the sensitivity of the stylus using the Pressure Threshold and Tablet Margin fields in the Pointer Preferences menu.



Tablet Model box — This box displays the model of the currently installed tablet.

Pressure Threshold field — Use this field to set the amount of pressure that you want to apply when using the stylus. Use a higher value to decrease the sensitivity of the stylus (more pressure required). A value of 5 works well.

Threshold Test button — Use to interactively assess the sensitivity of the stylus. Press the stylus on the button and a sliding grey bar in the message bar shows the response to the amount of pressure applied.

Tablet Margin — Use this field to set the sample rate per second for the stylus. Use a lower value for smoother cursor action, which may be especially useful in Paint. A value of 2 works well.

Reset button — Use Reset to return all values to their default settings.

HINT: If the pen stops functioning, click the Reset button with the mouse to re-activate it.

Jitter Filter — If you are using an Intuos tablet (Wacom GD- series), the cursor may jitter when the tablet is close to a source of magnetic fields, such as your monitor. You can adjust the Jitter Filter setting of the stylus to compensate for this unwanted jitter.

Modify the Jitter Filter value until you get the result you want.

This value represents the minimal amount of stylus movement needed to report a new coordinate on the screen.

Audio Preferences

The available audio preferences vary depending on the audio system you are using and your platform. For information on setting the audio preferences for your system, see Chapter 24, “Using Audio.”

Using Custom Menus

You can create a custom menu setup by changing the size or location of buttons, fields, and boxes on the interface.

All interface items are aligned on a grid. Press the **F9** key to display the grid on the screen. Press **F9** a second time to hide the grid.

To select an interface item, click it using the left mouse button. To select a group of interface items, press and hold the **CTRL** key while clicking on each. Drag the control handles that appear on the selected items to change their size. To move a selected item, press and hold the item and drag it to the desired location. To reposition the button title text, click the button with the centre mouse button and drag the text to the desired location.



WARNING: There are a number of hidden buttons in the menus. Do not alter these buttons, as you may affect the functionality of **flame**.

Saving and Removing a Custom Menu Setup

After modifying the menu setup, you can save your custom setup for later work sessions.

To save a custom menu setup:

1. Make the desired menu changes.
2. In the Preferences menu, click Menu Library.
The Menu Library menu appears.
3. Click Save Menu.
The on-screen keyboard and file browser appear.
4. Enter a name for your menu setup files and click Enter. (It is not necessary to specify a filename extension.)
The custom menu setup is saved.

To remove a custom menu setup:

1. In the Menu Library menu, click Remove Menu.
The file browser appears.
2. Select the menu setup you want to remove.
The Confirm button appears.
3. Click Confirm to remove the menu and return to the Menu Library menu.

Loading a Custom Menu Setup

If you want to load a custom menu setup, you must do so when starting **flame**.

To start flame with a custom menu setup:

At the command line type:

```
flame -f <filename>
```

where **<filename>** is the name of the custom menu setup. (It is not necessary to specify a filename extension.)

flame starts, using the custom menu setup.

Animation uses successive positions of images and objects to create the illusion of movement. For example, in Action, you can set the position of objects, light sources, and cameras for an animated result.

Summary

In this chapter, you learn about:

- “Using the Channel Editor” on page 129
- “Editing Animation Curves and Keyframes” on page 134
- “Understanding Motion Path Animation” on page 141
- “Using Explicit Animation” on page 148
- “Using Curve Functions and the Curve Value Field” on page 157
- “Changing the Timing of an Animation” on page 163
- “Working with Expressions in the Channel Editor” on page 165

About Animation

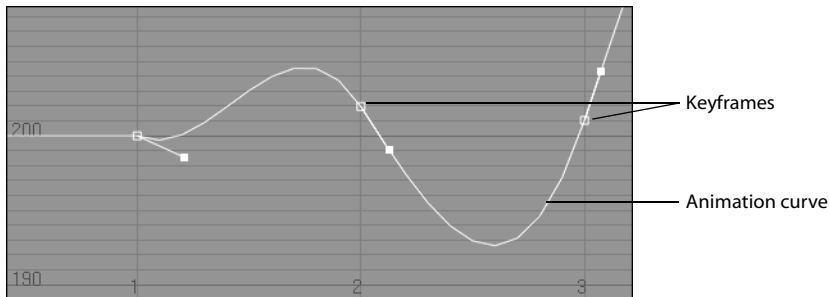
In **flame**, *animation* refers to a value that changes over a number of frames. A value can be anything from the position, rotation, or scaling of an object, to the gamma, gain, or offset in a colour correction. Values that can be used to create animation are called *channels*. These channels are contained in the Channel Editor and organized in a channel hierarchy of folders.

Almost all **flame** modules contain the Channel Editor and its interface and functionality remain the same in each module, while the specific channels correspond to the module itself as follows:

- Action—channels include the position, rotation, and scaling of an object. For example, you can use the Channel Editor to animate the position and colour of a light source.

- Paint—channels include the shape, scale, and rotation of geometry, as well as brush size.
- Keyer—channels include matte transparency and position of a garbage mask.
- Colour Corrector—channels include highlights, midtones, and shadows as well as hue, saturation, and contrast.

To create an animation, you set a different value for the same channel at two or more different frames. After you set a value for a channel at a particular frame, the value at that frame is called a *keyframe*.

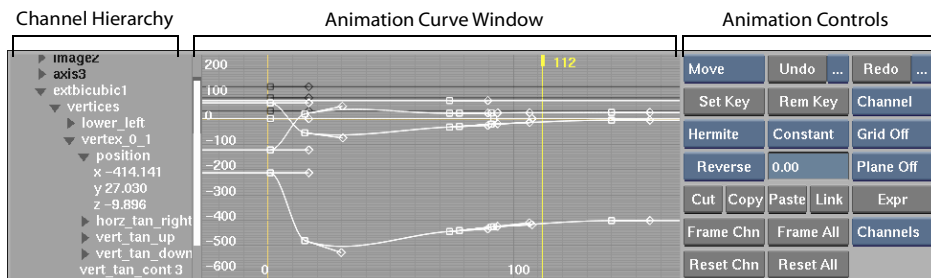


Keyframes are plotted on a graph in the Channel Editor. This graph maps frame numbers on the horizontal axis and channel values on the vertical axis. When you set more than one keyframe, the values between the keyframes are automatically calculated, creating an animation curve. The automatic calculation between keyframes is called interpolation. See “Setting Interpolation” on page 152.

For example, in the Colour Corrector, to reduce the saturation of a 30-frame clip from 200 to 0 over the entire clip, you would set saturation to 200 at frame 1 (first keyframe) and then set saturation to 0 at frame 30 (second keyframe). For detailed steps on how to do this, see “Working with Keyframes” on page 149.

Using the Channel Editor

You use the Channel Editor to create and modify animation curves. The Channel Editor consists of the channel hierarchy, the Animation Curve window, and a number of animation controls for modifying keyframes and animation curves. While you work with material in a module such as the Keyer, Action, the Compositor, or the Colour Corrector, you can access the Channel Editor by clicking the Animation button found in the menu of these modules. The Animation module consists of the channel hierarchy, the animation curve window, and the animation controls.



To access the Channel Editor:

1. Click Anim.



Animation button in Action

2. To view the Channel Editor in the image window, select Channels in the Channel View box and swipe the bar at the left or right side of the menu.

The contents of the image window appear in the menu and the Anim button changes to the Viewer button.

3. Click Viewer to toggle between the menu and the image window.

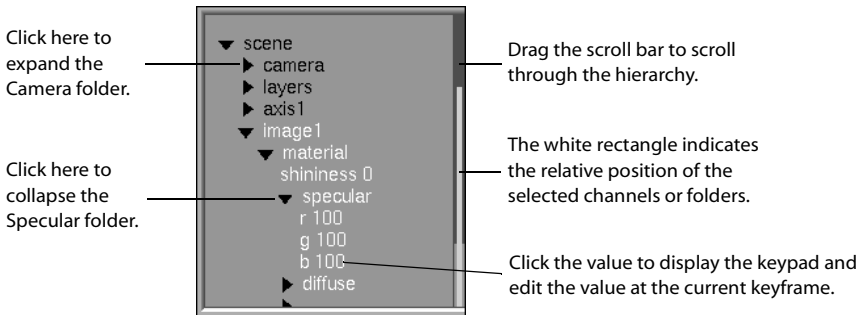
For example, you can view the Axis menu in Action while setting keyframes using the Channel Editor.

4. Swipe the bar again to view the clip in the image window.

The Channel Hierarchy

The channel hierarchy organizes the channels of a particular module in a hierarchy of folders. The folders contain channels that are grouped based on the properties they animate.

When the channel hierarchy first appears, only the top-level folders are shown. You can expand these folders to set values for specific channels. For example, the Scale folder contains channels for scaling the object on X and Y plane. Channels are indented and appear below their folders.



In the channel hierarchy, the keyframe value for the current frame is shown beside the channel name. The current keyframe position is indicated in the Animation Curve window when you select the channel name.

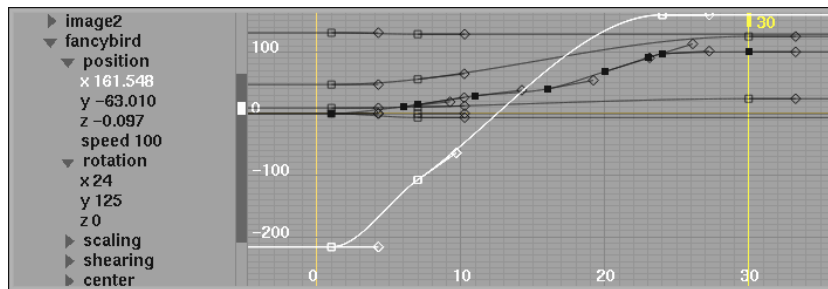
To set a value for a channel:

1. Click ► beside the folder name to expand it.

When several folders are expanded at once, it can be hard to match animation curves with their corresponding channels in the channel hierarchy. Collapse folders you are not using to isolate the channel you want to edit by clicking ▼.

2. Select the channel whose value you want to set by clicking its name.

The channel name and curve are highlighted.



NOTE: If you accidentally click the value instead of the channel name, the keypad appears, prompting you to enter a value. Click outside the keypad to hide it.

3. Edit the keyframe value by:
- Clicking the keyframe value and dragging left or right to decrease or increase the value.
 - Clicking the keyframe value to display the keypad and enter the new value.
 - Moving the keyframe on the animation curve.

Viewing Channels

You can also view the Channel Editor in a tabular format, making it easier to navigate the channel hierarchy and locate information. Using the Info channel view, you can sort by channel value, keyframe, and expression. See “Working with Expressions in the Channel Editor” on page 165.

Click the column heading to sort in ascending or descending order

Click to view channels in dot property format

Channel ▾	Component	Value	# Keys	Expression ▲
texture1	texture1	0	0	
poly1.subdivide	subdivide	5	0	
poly1.scaling	scaling	100	0	
poly1.material.transparency	transparency	0	0	
poly1.material.specular.r	r	100	0	
poly1.material.specular.g	g	100	0	
poly1.material.specular.b	b	100	0	
poly1.material.specular	specular			

Move

Undo ...

Redo ...

Set Key

Rem Key

Persp

Hermite

Constant

Grid Off

Reverse

0.00

Plane Off

Cut

Copy

Paste

Link

Expr

Frame Chn

Frame All

Info

Reset Chn

Reset All

Channel View box

To view or browse data in the Channel Editor:

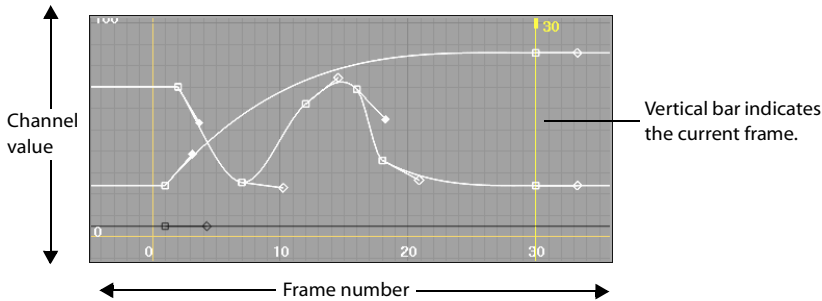
1. Click Anim to display the animation controls.
2. In the Channel View box, select a view as follows.

Select:	To:
Info	View channel information in tabular format and display channels that have expressions associated with them. With Info view, you can click a column heading to sort by Channel, Value, Number of Keys, or Expressions.
Tracks	Access the Track Editor and view channels as tracks, allowing you to adjust the timing of the animation.
Channels	View the animation curves and channel hierarchy in the Channel Editor. This is the default setting.

Animation Curve Window

Animation curves for channels are plotted on the graph in the Animation Curve window. An animation curve appears for channels that contain one or more keyframes. When you select a channel, its curve is highlighted in the window. When you select a folder, the animation curves for all channels in the folder are highlighted.

When you set a value for a channel at a particular frame, it becomes a keyframe in the animation. A keyframe has two components: a value for the channel and the frame where the value is set.

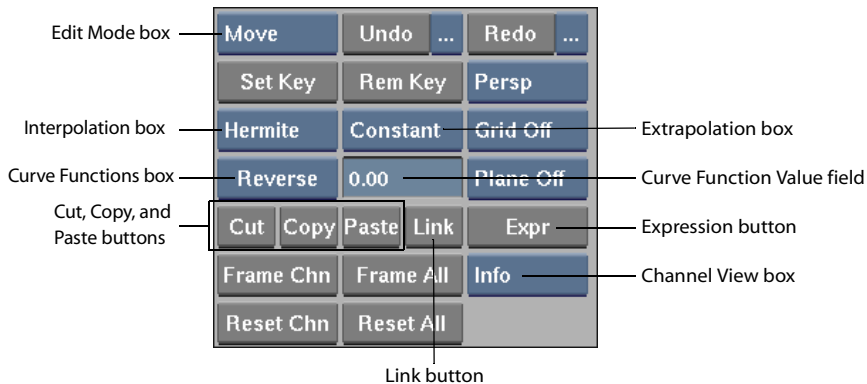


The frame number in the animation is mapped along the horizontal axis, ranging from 0 at the origin to the total number of frames (shown in the Total Frames field) at the right. The vertical bar indicates the frame currently appearing in the image window.

The channel value is mapped on the vertical axis, increasing upwards and decreasing downwards. The range depends on the portion of the curve being framed.

Animation Controls

The animation controls include a number of operations for modifying keyframes and animation curves. The Animation menu is shown in the following figure.



The animation controls are described in the following table.

Select:	To:
Edit Mode	Select, add, move, and modify animation curves and keyframes. See “Editing Animation Curves and Keyframes” on page 134.
Set Key and Rem Key buttons	Set and remove keyframes. See “Setting and Removing Keyframes” on page 151.
Interpolation and Extrapolation boxes	Use Interpolation to set how an animation curve is interpolated between keyframes. Use Extrapolation to select the behaviour of the animation before the first keyframe and after the last keyframe. See “Setting Interpolation” on page 152 or “Setting Extrapolation” on page 154.
Curve Functions box	Negate, reverse, or simplify animation curves or keyframes. See “Using Curve Functions and the Curve Value Field” on page 157.
Curve Function Value field	Provide a value when the selected Curve function requires a value, for example, the simplification factor when using Simplify.
Cut, Copy, Paste, and buttons	Cut and copy animation curves or keyframes. Paste pastes copied or cut animation curves or keyframe values to the selected channel. See “Copying and Pasting Channels or Keyframes” on page 156.
Link button	Link the animation expression of one channel to another. See “Linking Expressions to Other Channels” on page 168.
Frame Chn button	Display the selected channels in the Animation Curve window.
Frame All button	Display the animation curves for all expanded channels in the Animation Curve window.
Channel View box	Select a specific view for the Channel Editor: Channels, Tracks, or Info. See “Changing the Timing of an Animation” on page 163 and “Working with Expressions in the Channel Editor” on page 165.
Expression button	Enter expressions or mathematical formulas to modify the behavior of a selected channel. Using expressions, you can apply animation dynamically to various channels. See “Working with Expressions in the Channel Editor” on page 165.
Reset Chn	Reset selected channels or keyframes. If one or more channels are selected, their keyframes are removed. If one or more keyframes are selected, their tangent handles are reset.
Reset All	Reset all animation curves and remove all keyframes.
Undo and Redo buttons	Reverse or repeat the most recent animation actions. You set the number of Undo levels in the Setup menu of the module.

The Edit Mode box and Curve Functions box each contain several special animation options that are described in their own sections in this chapter: “Editing Animation Curves and Keyframes” on page 134 and “Using Curve Functions and the Curve Value Field” on page 157.

Editing Animation Curves and Keyframes

You can edit animation curves and keyframes using the Edit Mode box. You can also use this box to pan or zoom the Animation Curve window. The selected mode remains in effect until you select a different one.

Edit Mode box



The animation edit modes are described in the sections that follow.

Adding Keyframes

Use Add mode to add a keyframe to a channel's animation curve. Adding a keyframe to the animation curve does not add a control point to the motion path. Added points are *unlocked* and appear as square outlines on the curve. See “Modifying a Motion Path Using Edit Modes” on page 145 for details about using Add mode when working with motion paths.

To add a keyframe:

1. Select the channel.
2. In the Edit Mode box, select Add.
3. Position the cursor at the coordinates (frame number and value) where you want to add the new keyframe and click.

The new keyframe is added at the specified coordinates for all selected channels. You can also move the new keyframe after it has been added. The shape of the animation curve and the slope of the new keyframe are generated automatically.

Using Auto Mode

Use Auto mode to reset a keyframe after its tangent handle has been moved. You can use Auto mode with Break mode to reset a split slope handle. You can also reset a split tangent handle by selecting the keyframe and clicking the Reset button.

To reset a tangent handle:

1. In the Edit Mode box, select Auto.
2. Click the tangent handle.

It resets to its default value. The slope of the animation curve is updated automatically.

To reset a split tangent handle:

1. In the Edit Mode box, select Break.
2. Click the keyframe that you want to reset.
One of the two tangent handles is removed.

NOTE: To perform this operation, the keyframe must already have two tangent handles.

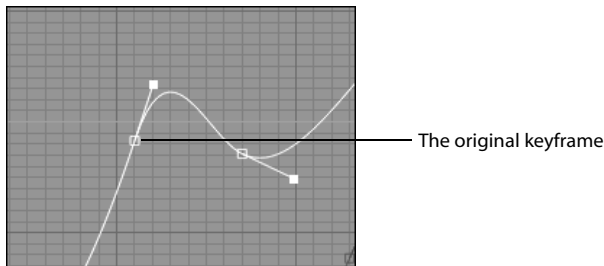
3. In the Edit Mode box, select Auto.
4. Click the remaining tangent handle.
The tangent handle is reset to its default value.

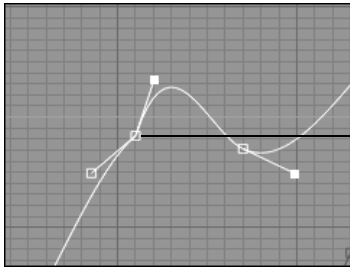
Notice that any custom curve is reset to the curve that existed before breaking the keyframe.

Breaking Keyframe Tangent Handles

Use Break mode to split a keyframe's tangent handle into two tangent handles. You can then move each tangent handle independently and have more control over the slope of an animation curve.

For example, the following figure shows a keyframe before and after using Break mode.





Break was used to split the keyframe's tangent handle. The two tangent handles were then moved. Notice the slope has changed on either side of the keyframe.

NOTE: Only animation curves that use Hermite or Natural interpolation have keyframes with slope handles. See “Setting Interpolation” on page 152.

Using Break mode is also convenient for tweaking the timing of a motion path. See “Changing Motion Path Timing” on page 147 for details.

To break a slope handle into two tangent handles:

1. In the Edit Mode box, select Break.
2. Click the keyframe you want to split.

Two tangent handles appear for the keyframe. Each tangent handle appears as an outlined square and controls the shape of the curve on its respective side. Click the keyframe again to recombine the handles into a slope handle.

NOTE: If two channels overlap, you may have to select the channel first before using Break.

3. Select Move mode and move one or both tangent handles.
As you move the handles, the slope on each side of the keyframe is modified independently.

Deleting Keyframes

Use Delete mode to delete one or more keyframes from an animation curve. You can also delete keyframes for more than one channel using the Rem button. For more information see “Setting and Removing Keyframes” on page 151.

To delete a keyframe:

1. In the Edit Mode box, select Delete.
2. Click the keyframe.

The keyframe is removed from the curve. The shape of the animation curve is updated automatically.

To delete a range of keyframes:

1. In the Edit Mode box, select Delete.
2. Drag a selection box around the keyframes.

When you release, all keyframes inside the selection box for the selected channel(s) are deleted. Animation curves are updated automatically. Unselected keyframes and keyframes in collapsed folders are unaffected.

Moving Keyframes

Use Move mode to change the position of a keyframe or slope handle on an animation curve.

If you move a keyframe, the frame number (horizontal) and keyframe value (vertical) coordinates of the keyframe are updated and displayed. The shape of the curve is updated automatically.

If you move a slope handle, the value of the slope is updated and displayed. After a slope handle is moved, it appears as a solid square.

While moving a keyframe, hold down **ALT** to snap the frame value (vertical axis) to the nearest integer. Press **CTRL+ALT** to snap to the nearest multiple of ten.

To move a keyframe or slope handle:

1. Select the channel with the keyframe or slope handle that you want to move. You may have to click the Frame Chn button to view the curve.
2. In the Edit Mode box, select Move.
3. Drag the keyframe or the slope handle to a new position.

Using Pan Mode

Use Pan mode to scroll through the Animation Curve window.

To use Pan:

1. In the Edit Mode box, select Pan.
2. Position the cursor in the Animation Curve window and drag in any direction.
You can also pan in any mode by pressing **SPACEBAR** and dragging in the image window.

Using Rectangular Zoom

Use Rectangular Zoom mode to zoom in on a particular region of the Animation Curve window.

To use Rectangular Zoom:

1. In the Edit Mode box, select Rect Zoom.
2. Position the cursor in the Animation Curve window and drag diagonally to draw a rectangle over the region to enlarge.
The selected region is enlarged.
3. To zoom out, click the Frame All or Frame Chn button:

- To frame all the curves in the Animation Curve window, click Frame All.
- To frame the selected curve in the Animation Curve window, click Frame Chn.

Selecting Multiple Keyframes and Channels

Use Select mode to select multiple keyframes or channels, for example, to modify every second keyframe in an animation curve or apply the same modification to two or more channels from different folders.

NOTE: You do not have to be in Select mode to select a channel; you can select a channel in any edit mode by clicking its name in the channel hierarchy.

To select multiple channels:

1. In the Edit Mode box, select Select.
2. Click a channel to select it.

You can click a channel's name in the hierarchy, or its animation curve. When a channel is selected, its name and animation curve are highlighted.

3. **CTRL**-click another channel to add it to the selection. **CTRL**-click a selected channel to deselect it.

To select multiple keyframes:

1. In the Edit Mode box, select Select.
2. Click a keyframe to select it.

Hold down **CTRL** and drag in the Animation Curve window to draw a selection box. All keyframes inside the selection box are selected.

3. **CTRL**-click another keyframe to add it to the selection. **CTRL**-click a selected keyframe to deselect it.

Hold down **ALT+CTRL** and drag in the Animation Curve window to add all keyframes inside a selection box to the selection.

To select multiple keyframes from a selected channel:

1. Select one or more channels.
2. Hold down **CTRL+F** and drag along the horizontal axis.

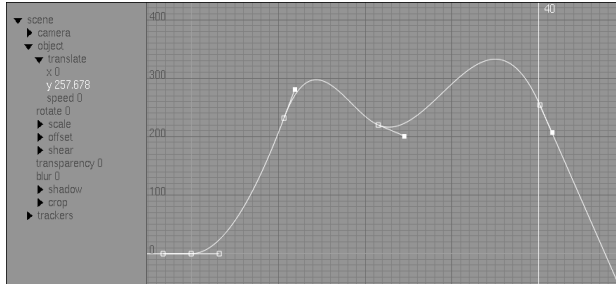
The keyframes of the channels within the drag range are selected.

For example, dragging the selection box over the first 10 frames of an animation selects all keyframes that fall within those 10 frames.

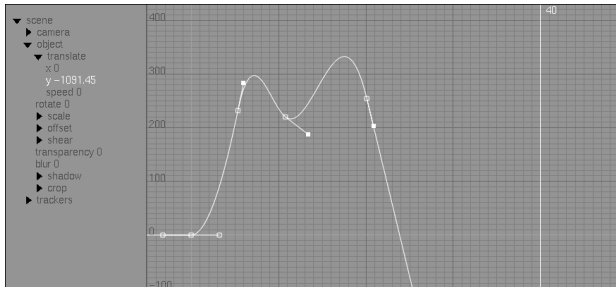
You can also **CTRL**-click to add keyframes to the selection, or hold down **ALT+CTRL+F** and drag to add a range of keyframes to the selection.

Using XScale Mode

Use XScale mode to change the horizontal scale of an animation curve. This lets you change the number of frames used by an animation without moving each keyframe manually.



The original animation curve



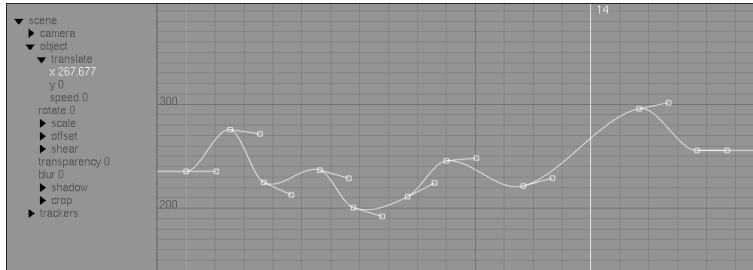
XScale used to compress a curve from 40 to 20 frames

To use XScale:

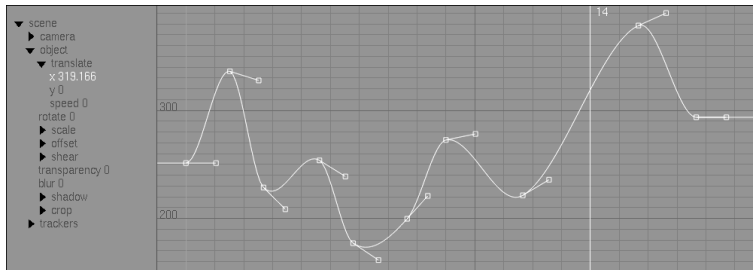
1. Select the channel.
You may have to click Frame Chn to view the curve.
2. In the Edit Mode box, select XScale.
3. Press the first or last keyframe and drag left or right:
 - To compress the animation curve, drag the first keyframe right.
 - To expand the animation curve, drag the first keyframe left.
 - To expand the animation curve, drag the last keyframe right.
 - To compress the animation curve, drag the last keyframe left.

Using YScale Mode

Use YScale mode to change the vertical scale of an animation curve or a selection of keyframes. This lets you change the keyframe values for a channel or a group of keyframes while maintaining each keyframe's relative frame position.



The original animation curve



YScale used to scale the curve based on a selected keyframe

To use YScale on a channel or group of keyframes:

1. Select a channel or group of keyframes.
You can select more than one channel or a group of keyframes from more than one animation curve.
2. In the Edit Mode box, select YScale.
3. Click the keyframe that will act as the centre of the YScale and drag up or down to increase or decrease the value of the other keyframes relative to the selected keyframe.
YScale works differently if you select more than one curve. For the other selected curves, the number of the frame that you select is verified in all animation curves. If the animation curve has a keyframe at the selected frame, this keyframe is used as the centre of the Yscale for that curve. If the animation curve does not have a keyframe at the selected frame, the curve is deselected and remains unscaled.

Using Translate Mode

Use Translate mode to move an animation curve or a selection of keyframes horizontal and/or vertical.

To translate a channel or a group of keyframes:

1. Select a channel or group of keyframes.
2. In the Edit Mode box, select Translate.
3. Click in the Animation Curve window and drag in any direction.

If you are translating a channel, its animation curve is moved. The relative positions of the animation curve's keyframes remain unchanged.

If you are translating a group of keyframes, each keyframe is moved. The relative positions of the selected keyframes remain unchanged.

Using Zoom Mode

Use Zoom mode to enlarge or reduce the display in the Animation Curve window.

To use Zoom:

1. In the Edit Mode box, select Zoom.
2. Click the cursor in the Animation Curve window.
The cursor changes to a magnifying glass.
3. To zoom in, drag right. To zoom out, drag left.

Understanding Motion Path Animation

With motion path animation, object positions follow a spline (curved line) that you draw in the image window. Using this method, you have a visual reference for the object's position and you can change the speed of an object without affecting its position.

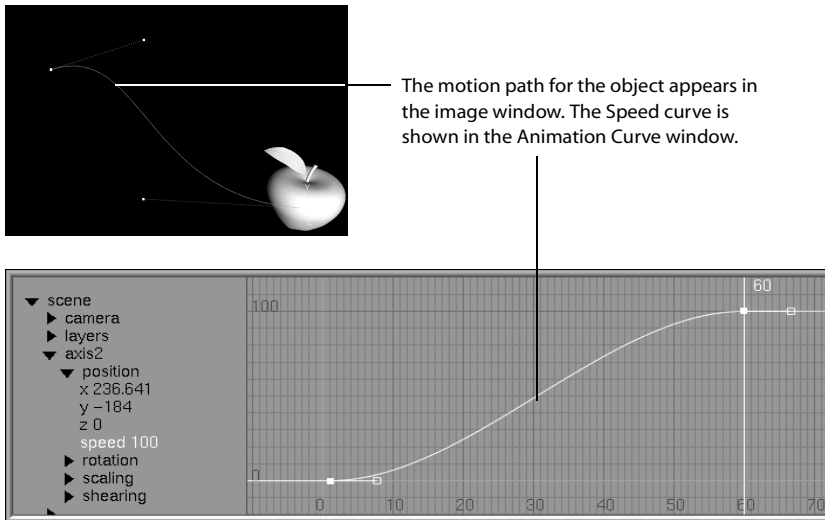
If you are animating the position of axes, lights, particle manipulators, and the camera in Action, a garbage mask in the Keyer, or an object in the Compositor, you should use a motion path.

Use motion paths to animate the position of any of the following:

- Axes, light sources, the camera eye, the camera point of interest, or particle manipulators in Action. The Motion Path buttons in the Axis and Light menus enable or disable motion path animation for the selected axis or light. The Motion Path button in the Camera menu enables or disables motion path animation for both the camera eye and point of interest.

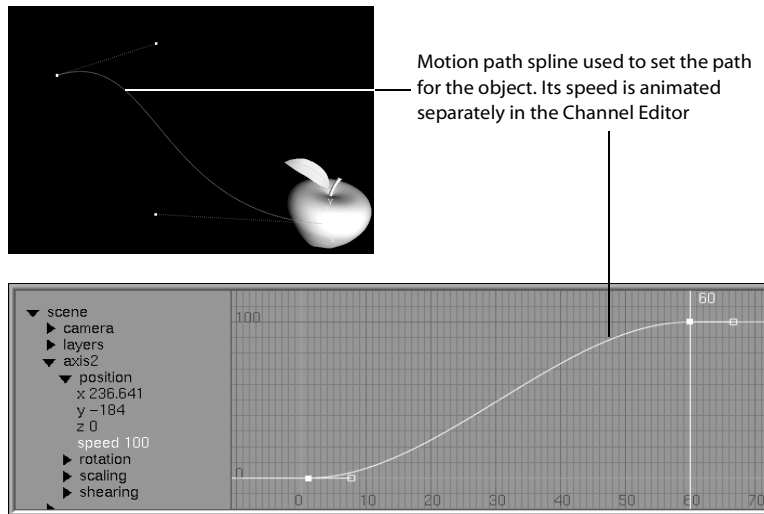
- The camera or the object in the Compositor. The Setup menu contains the Object and Camera buttons. The Object button enables or disables motion path animation for the object. The Camera button enables or disables motion path animation for the camera.
- Keyer masks in the Keyer. In the Axis menu (in the Mask menu), the Motion Path button enables or disables motion path animation for the selected keyer mask.

Motion path animation uses a spline drawn in the image window as the path for the object. The timing or speed of the object is animated separately using a speed curve, which is found in the Channel Editor.



In Action and the Keyer, motion paths are shown regardless of whether an axis, light source, or camera folder is expanded or collapsed in the Channel Editor. In the Compositor, motion paths are only visible when the object or the camera's translate folder is expanded.

For example, the following figure shows the motion path and the speed curve for an object animated in the Compositor.



The Spline

The spline represents the path of the object, camera, light source, for example, and consists of alternating coloured segments. Each segment represents one frame and indicates the object's speed: the longer the segment, the faster the object is travelling. When the motion path is selected, the segments of the spline are red and blue. When unselected, the segments are blue and green.

The spline is drawn using control points. Each control point has a tangent handle that changes the shape of the spline. Changing the shape of the spline between control points also affects the speed of the animation.

The Speed Curve

The speed curve represents the percentage of the path travelled by the animation in time. In other words, the keyframes on the speed curve correspond to the timing values for the keyframes.

The speed curve is created automatically when you draw the motion path. The speed curve only controls the timing of the animation. When you create the motion path, a keyframe is added to the speed curve for each control point on the spline. You can edit the speed curve and adjust the timing of the animation by moving, adding, and deleting control points from the spline or keyframes in the speed curve. Modifying keyframes on the speed curve does not affect the shape of the motion path spline.

For example, a speed curve keyframe with a value of 0% means the animation is at the beginning of the spline. A speed curve keyframe with a value of 100% means that the animation is at the end of the spline. When you first create or add control points to the spline, an equal number of keyframes are added to the speed curve. Each control point on the spline corresponds to a keyframe in the speed curve, letting you change timing of the animation between control points.

If Auto Key is enabled, the second control point is set when you move the object. If Auto Key is disabled, you must click the Set key button after moving the object to its new position.

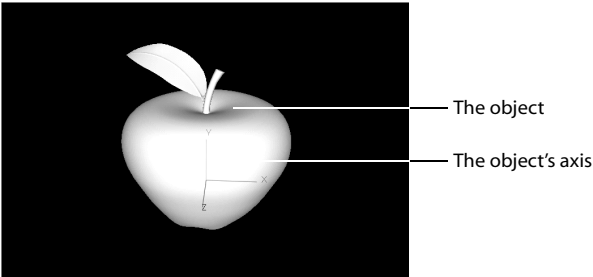
To create a motion path:

1. Enable the motion path setting according to the module you are using as follows.

Module	Description
Action	To animate the position of an axis, enable Motion Path in the Axis menu. To animate the position of a light, enable Motion Path in the Light menu.
Compositor	To animate an object's position, enable Object in the Setup menu.
Keyer	To animate a garbage mask, enable Motion Path in the Axis menu by choosing Key and Mask.

2. Set control points either manually or automatically when the shape of the spline changes.
3. Create the first control point on the motion path spline by moving the object in the image window to a specific location.

For example, if you have a 30-frame clip in Action and you want a 3D object to move from the upper-left corner in frame 1 to the lower-right corner in frame 30, go to frame 1 and move the object to the upper-left corner.



There are two ways to move an object:

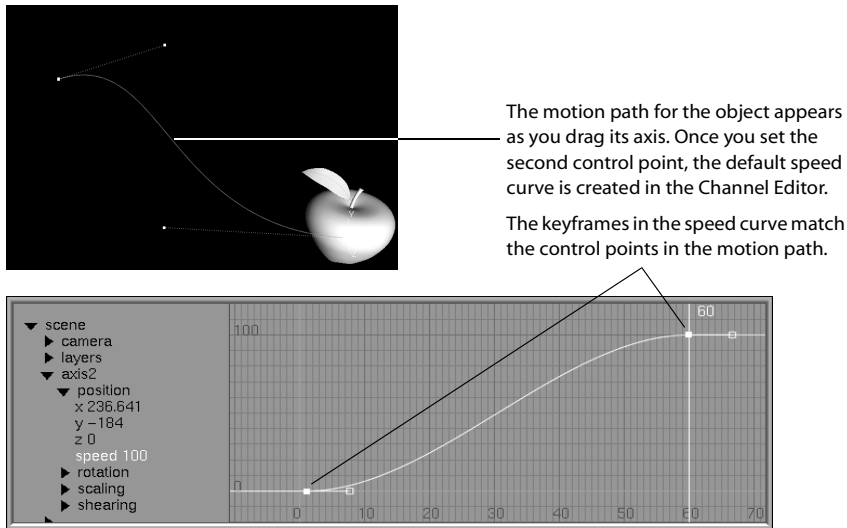
- Change the values of the position fields in the appropriate menu. For example, position fields for a light in Action are found in the Light menu; the position of an axis in Action is set in the Axis menu.
- Drag the object by its axis in the image window.

If Auto Key is enabled (set in step 2), a control point is automatically set for the motion path spline. If Auto Key is disabled, you must click the Set key button to manually set control point.

4. Set the second control point on the motion path spline.

To complete the motion path, you must set at least one other control point. Go to frame 30 and move the object to the lower-right corner.

NOTE: If you drag the object to set its second control point, make sure you click either the X, Y, or Z axis and not on the centre point where the three axes meet. Dragging from the centre may change the beginning keyframe of the motion path instead of setting a new keyframe.



Modifying a Motion Path Using Edit Modes

Use the options in the Edit Mode box to change the amount of control you have over the timing of the animation. For example, you can delete all but the first and last keyframes from the speed curve to control the timing over the entire animation. Or, you can add keyframes to the speed curve to change the speed of an animation between control points on the motion path. For details about using the options in the Edit Mode box, see “Editing Animation Curves and Keyframes” on page 134.

You can also change the timing of a motion path by changing the speed curve’s interpolation. See “Setting the Interpolation” on page 135.

Use this mode: To control the motion path as follows:

Add

Add a keyframe to the speed curve.

Adding a keyframe to the speed curve does not add a control point to the motion path. The added point is *unlocked* and appears as a square outline. An unlocked keyframe in the speed curve lets you change the timing at that frame without changing the shape of the spline.

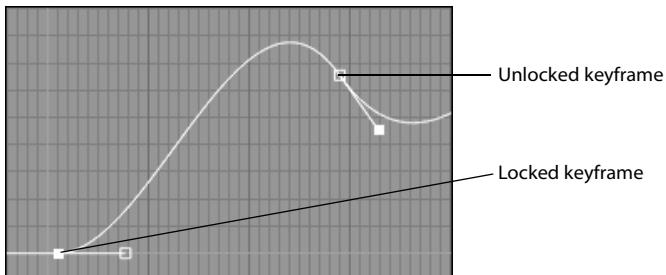
Use this mode: To control the motion path as follows:

Auto	Reset the tangent handle of a control point on the spline or the slope of a keyframe in the speed curve.
Break	Break a control point's tangent handle into two tangent handles. You can then move each tangent handle independently.
Delete	<p>Delete a control point on the motion path or delete a keyframe from the speed curve.</p> <p>Deleting a control point from the spline deletes its associated keyframe from the speed curve, affecting both the shape and speed of the animation.</p> <p>Deleting a locked keyframe from the speed curve unlocks its associated control point on the spline. You can use the unlocked point to change the shape of the spline without affecting the timing of the animation.</p>
Move	Move a control point on the motion path or a keyframe on the speed curve.
Select	Select and edit a particular spline when more than one motion path appears in the image window.
Translate	<p>Reposition the speed curve in the animation curve window by dragging a control point on the curve. The first and last keyframes do not move.</p> <p>You cannot move or translate these keyframes vertically because the timing of the animation must begin at 0% and end at 100%.</p>

Working with Motion Path Control Points

Because motion paths are drawn in the scene and not in the Channel Editor, you do not have to be positioned at a specific keyframe to move the control points of a motion path. You can move control points and tangent handles for any frame in the animation.

You can also add control points to a motion path. Adding control points to a motion path also adds a keyframe to the speed curve. The control point and keyframe are *locked*, which means that moving the control point also affects the keyframe and vice versa. A locked keyframe appears as a solid square.

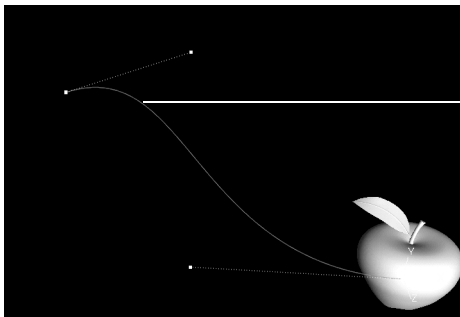


Moving a locked control point on the spline changes the position of the object at that frame. A locked keyframe on the speed curve can only be moved along the horizontal axis (frame number). This changes the frame number at which the animation reaches a particular point on the spline, but it does not affect the shape of the spline. This is referred to as changing the timing of the animation.

Changing Motion Path Timing

You can also change the timing of the animation by moving the tangent handles of any keyframe on the speed curve. The steeper the slope of the curve, the faster the animation will move. You can also reverse the animation by creating a curve with a negative slope.

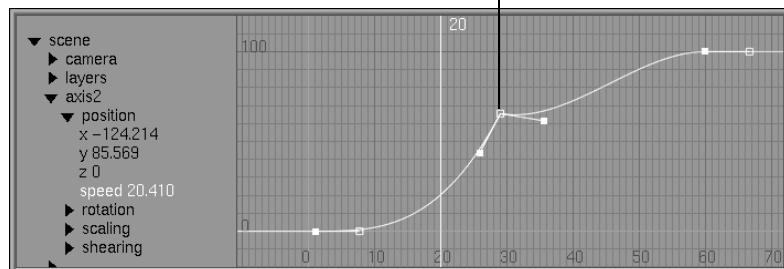
Breaking the slope of a keyframe on the speed curve introduces a discontinuity in the timing of an animation. For example, you can have the animation accelerate until it reaches a given keyframe then have it continue slowly. This change in timing occurs between control points.



You can change the timing without changing the spline by adding a keyframe to the speed curve.

You can then break the tangents of the keyframe and move each one separately.

The speed curve makes the apple accelerate rapidly until it reaches frame 29, where it stops and continues slowly.

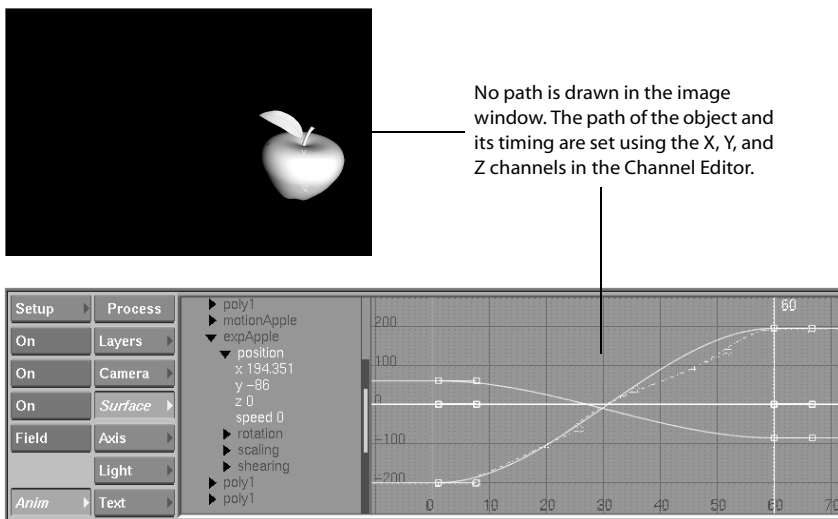


Using Explicit Animation

Explicit animation uses the X, Y, and Z position curves in the Channel Editor to animate an object, giving you greater control over an object's position. Unlike motion path animation, you do not use a separate curve to control the timing or speed of the object. The X, Y, and Z position curves determine the speed at which the object moves. If you change the speed of an object using this method, its position changes.

For example, if you are animating saturation in the Colour Corrector and you want the position of the camera and the object to remain constant, you should use explicit animation.

The following figure shows the explicit animation of an object animated in Action.



Working with Keyframes

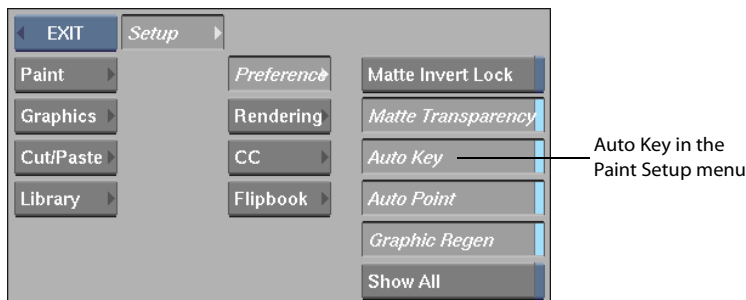
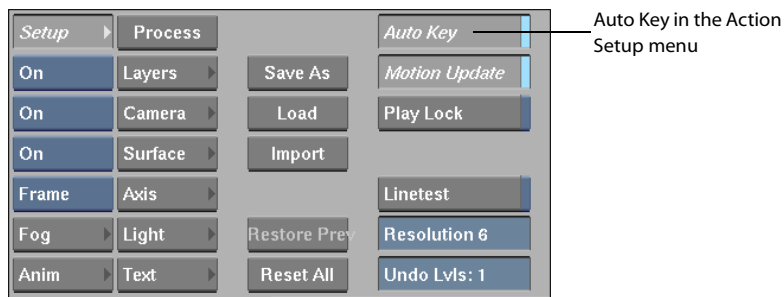
Changing the value of a channel creates a keyframe. A keyframe is expressed using x and y coordinates on the animation curve, where x represents the frame number at which a channel value is set and y represents the channel value.

You can animate the X, Y, and Z positions for object properties such as scale, rotation, transparency, or colour. As you change channel values, you can add keyframes manually or automatically. You must set at least two keyframes to create an animation. The values between the first and second keyframes are calculated by interpolation and plotted on the animation curve. The values that lie outside the animation curve are calculated by extrapolation. See “Setting Interpolation” on page 152 and “Setting Extrapolation” on page 154 for more information.

To set keyframes explicitly using Auto Key:

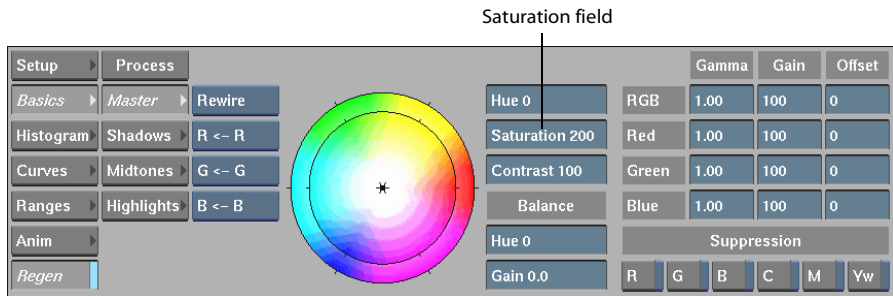
1. Enable Auto Key.

The location of the Auto Key button may vary from module to module. The following figures show the location of the Auto Key button in Action and in Paint.



2. At the first keyframe, frame 1, change the saturation to 200.

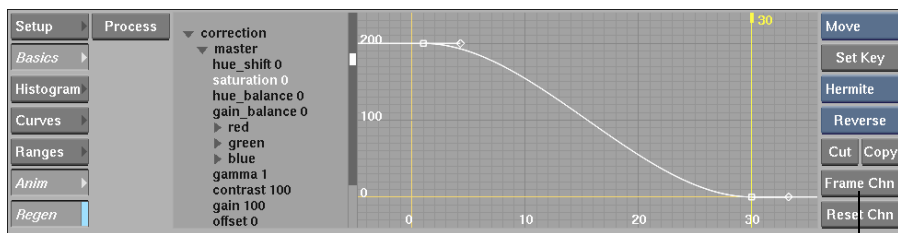
You can animate the saturation using either the Saturation field or its associated channel in the Channel Editor.



3. Set the second keyframe at frame 30 and change the saturation to 0.

Once the second keyframe is set, the values between the two keyframes are interpolated and plotted on a curve

4. Click Frame Chn to display the curve and its keyframes as shown in the following figure.



Setting and Removing Keyframes

The Set Key and Rem Key buttons are used for setting and removing keyframes explicitly. Keyframes can be set either automatically or you can set them explicitly.

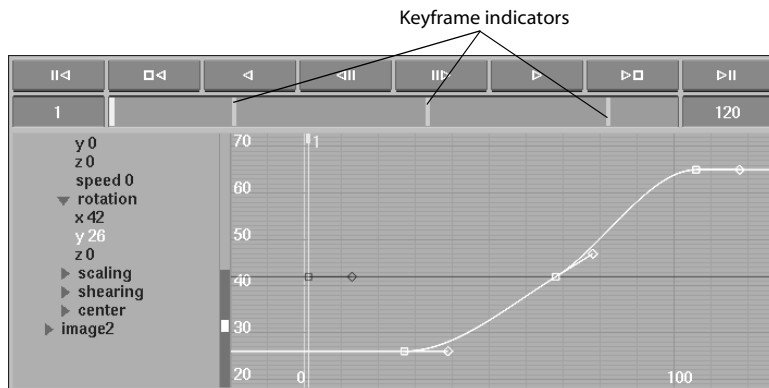
Keyframes are set automatically when the Auto Key button in the Setup menu is enabled. When the value of a channel is set in a frame, a keyframe is automatically created for that channel. When the Auto Key button is disabled, you must set the keyframes explicitly using the Set Key button in the Channel Editor. The Set Key button is particularly useful if you want to:

- Set a keyframe in the first frame of the clip without changing the default channel values.
- Copy channel values from one frame to another.

To set keyframes explicitly:

1. In the timeline, specify the frame where you want to set the keyframe.
2. In the Channel Editor, select a channel.
3. Set a value, and click Set Key.

The values for the selected channels are set in the current frame and the position of keyframes that you set is indicated in the timeline. Keyframe indicators are displayed as blue bars in the timeline.



To copy keyframe values:

1. In the Setup menu, disable Motion Update.
2. In the timeline, specify the frame for the keyframes you want to copy.
3. Select the channels containing the keyframes and click Copy.
4. In the timeline, specify the new frame.
5. Click Set Key.

The keyframe values for the selected channels are pasted and set in the current frame.

To remove keyframes:

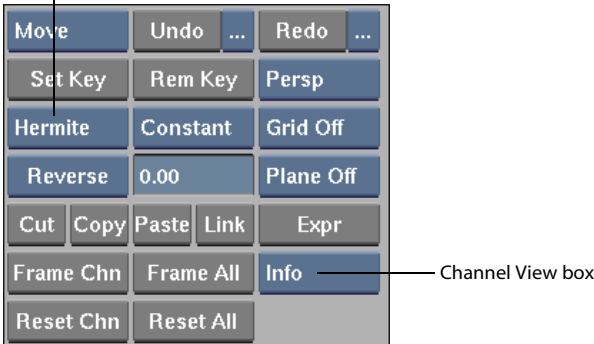
1. In the timeline, specify the frame for the keyframes you want to remove.
2. Select the channels containing the keyframes.
3. Click Rem Key.

The keyframes for the selected channels are deleted.

Setting Interpolation

Interpolation defines the shape of an animation curve based on the range of keyframe values. You set the interpolation mode to determine the smoothness of the transitions for an entire curve or in between keyframes. The smoother the transition between each keyframe, the less jitter that appears in your result.

Interpolation box



To change the interpolation for an entire curve:

1. In the Channel View box, select Channel.
2. Select the animation curve and click Frame Chn.
3. In the Interpolation box, select an interpolation mode.

As soon as you select the interpolation, it is applied to the selected curve.

Understanding Interpolation Modes

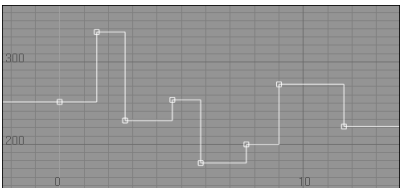
The following table describes the interpolation modes and illustrates the resulting curves.

Interpolation mode

Constant

Produces a square curve. In a square curve, the value of one keyframe is held constant until the next keyframe.

Resulting curve

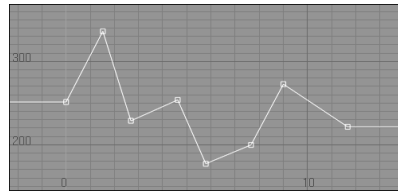


Interpolation mode

Linear

Joins keyframes using straight lines.

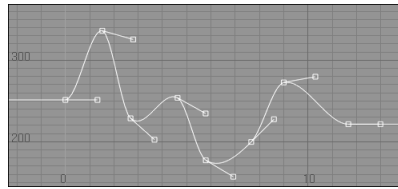
Resulting curve



Hermite

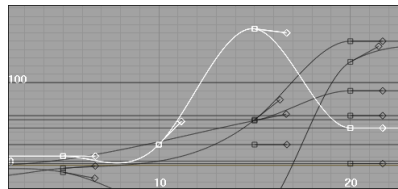
Produces a smooth curve with the smoothest transitions between keyframes. This is the default setting. In Hermite interpolation mode, each keyframe on the animation curve has an associated slope represented by a tangent handle, ending in a slope handle. By default, the slope is managed automatically by **flame**. You can change the shape of the animation curve by dragging the slope handle. When you move a slope handle, it changes to a solid square.

You can also introduce a discontinuity in an animation curve by breaking the slope handle into two tangent handles and moving each tangent handle independently. See “Breaking Keyframe Tangent Handles” on page 135.



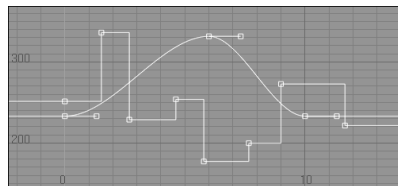
Natural

Produces a smoother, more continuous curve than Hermite. The Natural animation curve has natural cubic splines, similar to Hermite curves, but the tangents of the spline adjust themselves automatically to match the first and second derivatives at each keyframe. This results in a very smooth curve. Each tangent is re-evaluated when you move a point on the curve.



Mixed

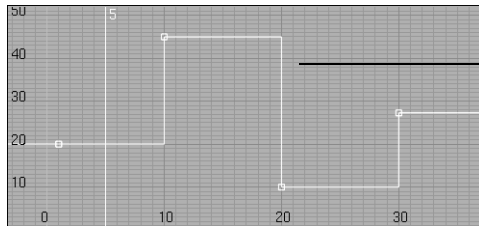
You cannot select Mixed. This option is shown automatically when you selected two or more channels that have different interpolation modes, or when an animation curve contains different interpolation modes between keyframes.



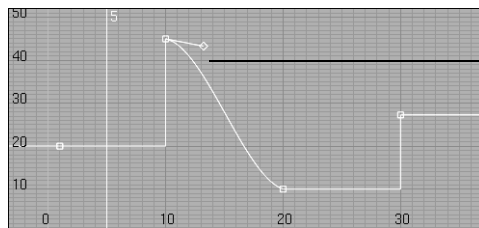
To change the interpolation mode between keyframes:

1. Select the keyframe before the section of the animation curve you want to change.
2. In the Interpolation box, select an interpolation mode.

The interpolation between the selected keyframe and the next keyframe is changed. For example, the following figure shows the results of changing a keyframe's interpolation from Constant to Hermite in an animation curve that uses Constant interpolation.



The original animation curve using Constant interpolation.



The same curve after changing the second keyframe to Hermite interpolation.

Setting Extrapolation

Extrapolation defines the shape of an animation curve before the first keyframe and after the last keyframe of the curve. You will see the effect only if there are frames before and after the first and last keyframes.



Extrapolation box

To change the extrapolation:

- 1. Select the animation curve.
- 2. In the Extrapolation box, select an extrapolation mode.

As soon as you select the extrapolation, it is applied to the selected curve.

Understanding Extrapolation Modes

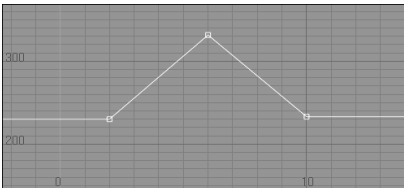
The following table describes the extrapolation modes and illustrates the resulting curves.

Extrapolation mode

Constant

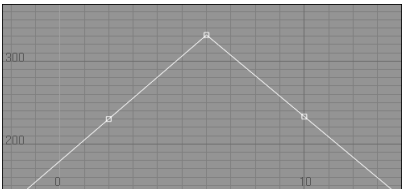
Applies the channel value in the first keyframe to all frames before the first keyframe. The value of the last keyframe is applied to all frames after the last keyframe. This is the default setting.

Resulting curve



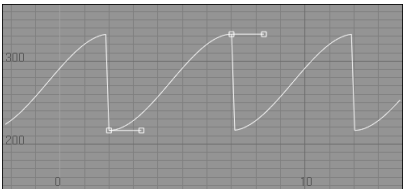
Linear

Continues the curve in linear fashion before the first keyframe and after the last keyframe.



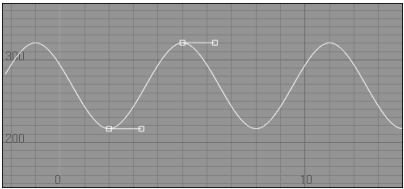
Cycle

Creates cycles in the curve. The period of each cycle is determined by the first and last keyframe.



Reverse and Cycle

Repeats, reverses, then cycles the curve.

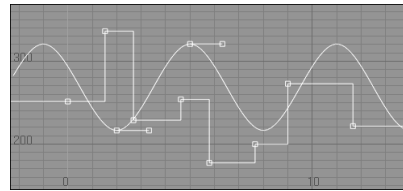


Extrapolation mode

Mixed

You cannot select Mixed specifically. This option is shown automatically when you select two or more channels that have different extrapolations.

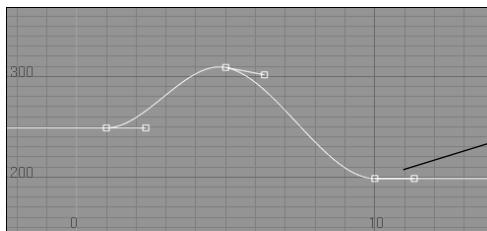
Resulting curve



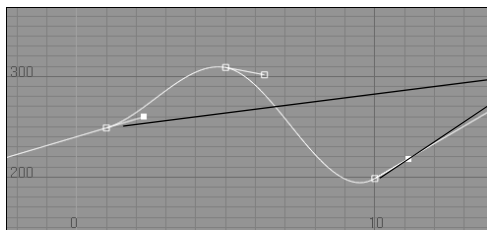
Using Hermite Interpolation with Linear Extrapolation

Using Hermite interpolation with Linear extrapolation affects the animation differently from using Linear extrapolation with Linear interpolation.

When both interpolation and extrapolation modes are set to Linear, the animation continues diagonally. Using Hermite interpolation with Linear extrapolation appears to be the same as using it with Constant extrapolation until you move the tangent handle of the first or last keyframe.



Although the extrapolation is set to Linear, the animation seems to end but the animation actually continues.



Linear extrapolation is not evident until you move the first or last tangent handle.

Copying and Pasting Channels or Keyframes

Use the Copy and Paste buttons to copy and paste animation curves or keyframes between channels. The Chn button must be enabled for copy and paste operations.

To copy and paste a curve:

1. In the Channel View box, select Channel.
2. Click Frame Chn.
3. Select the channel containing the animation curve.

You can also select a folder to copy all the channels in a folder.

4. Click Copy.

The selection is copied to the copy buffer.

5. Select the channel where you want to paste the copied curve.

6. Click Paste.

The channel values you copied are applied to the selected channel or folder.

To copy and paste a selection of keyframes:

1. Select the keyframes you want to copy.

2. Click Copy.

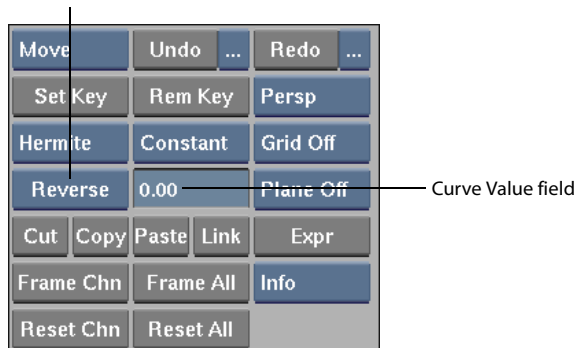
3. Paste the copied keyframes. Use the following methods when pasting keyframes:

- Select the channel where the keyframes will be pasted, then click Paste. This pastes the keyframes at the current frame.
- In Select mode **SHIFT**-click a channel to paste starting from the location of the cursor. For example, if you have keyframes in the copy buffer, **SHIFT**-click frame 10 on a channel to paste the copied keyframes into the channel starting at frame 10.
- You can copy and paste folders provided the folder to which you are pasting has the same channels as the folder you copied. For example, in Action, you can copy the Scale folder from an axis and paste it into the Scale folder of another axis because they have the same channels (X scale, Y scale, and so on). Trying to copy and paste the Camera folder to the Scale folder will not work because the two folders do not contain the same channels.

Using Curve Functions and the Curve Value Field

Use the Curve Functions box to negate, reverse, simplify, and remove jitter from animation curves or a group of keyframes. You can also colour curves and folders, offset curves or keyframes, and lock keyframes placed between frames.

Curve Functions box



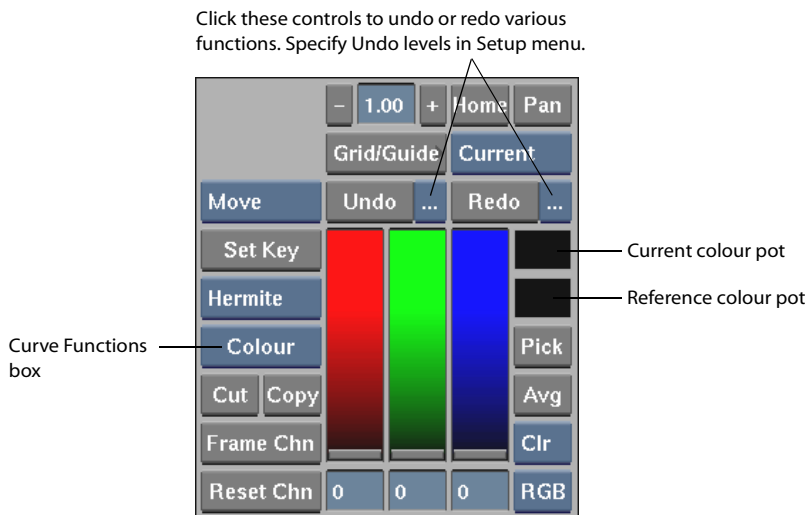
Some curve functions work only on curves and some only on keyframes. For certain curve functions, you must enter a numerical value in the Curve Value field.

Colouring Channels and Folders

Use the Colour option to change the colour of channels or folders. This is useful when you want to differentiate animation curves.

To colour channels or folders:

1. Select the folder(s) or channel(s) to colour.
See “Selecting Multiple Keyframes and Channels” on page 138.
2. In the Curve Functions box, select Colour.
The colour picker appears.



3. Use the colour picker to select a colour.
4. Click the current colour pot.
The colour of the selected channel(s), or all channels within the selected folder, changes. The new colour also appears in the Track Editor.

To cancel without colouring a channel, click in the empty portion of the menu, above the colour picker.

Rotating Tangent Handles

Use the tangent options—Tangent R and Tangent L—to rotate the left or right tangent handle of a keyframe. You can also rotate all the tangent handles on an animation curve.

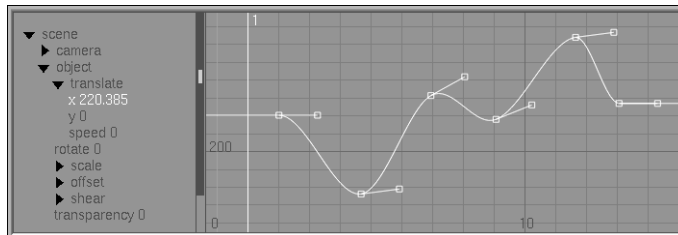
NOTE: Tangent R and Tangent L only work on curves and keyframes that use Hermite or Natural interpolation.

To rotate a keyframe's tangent handles:

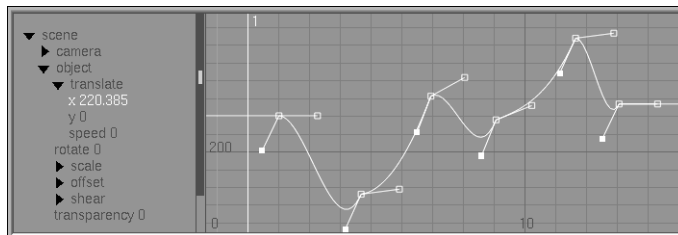
1. Select a channel or group of keyframes.
2. In the Curve Value field, enter a value for the rotation.
A negative value rotates each tangent counter-clockwise. A positive value rotates each tangent clockwise.
3. In the Curve Functions box, select Tangent R to rotate each right tangent handle or select Tangent L to rotate each left handle.

The slope handle breaks into two tangent handles first, and then rotates the handles.

For example, the following figure shows the results of applying Tangent L with a rotation value of 36 to the keyframes of an entire curve. No keyframes were broken before Tangent L was applied.



The curve before applying Tangent L. Note that none of the keyframes are broken.



The curve after applying Tangent L of 36. A break is applied to all keyframes before the tangents are rotated.

Offsetting Channels

Use Translate X and Translate Y to offset a curve or a group of keyframes on either the horizontal or vertical axis. The offset value is set using the Curve Value field.

To offset a channel:

1. Select the channel or keyframes to offset.
2. Specify the offset value in the Curve Value field.
A negative value offsets the channel or group of keyframes to the left (Translate X) or down (Translate Y). A positive value offsets to the right (Translate X) or up (Translate Y).
3. In the Curve Functions box, select Translate X to offset the channel horizontally or Translate Y to offset vertically.
The selected channel or keyframes are offset. The coordinates of each keyframe, relative to each other, remain unchanged.

Swapping Animation Curves

Use Swap to swap animation curves between two channels. Swap works in conjunction with Copy or Cut. You can also swap single keyframes, but not a group of keyframes.

To swap two channels:

1. Select the first channel to swap.
2. Click Cut.
The channel is cut to the copy buffer.
3. Select the second channel to swap.
4. In the Curve Functions box, select Swap.
The selected channel is swapped with the channel in the copy buffer.
5. Select the first channel and click Paste.
The two channels are swapped.

Flipping Channels and Keyframes

Use Negate to flip a channel or a group of keyframes vertically. Use Reverse to flip a channel or a group of keyframes horizontally.

To negate or reverse a channel or group of keyframes:

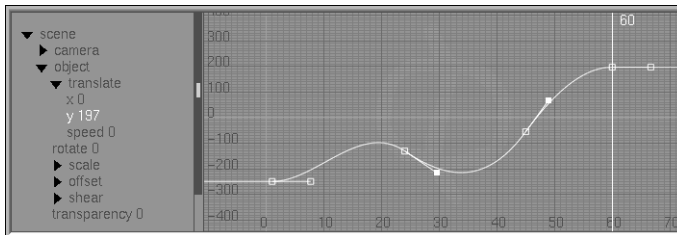
1. Select a channel or group of keyframes.
2. In the Curve Functions box, select Negate or Reverse.
The selection is flipped either horizontally (negate) or vertically (reverse).

Simplifying Keyframes

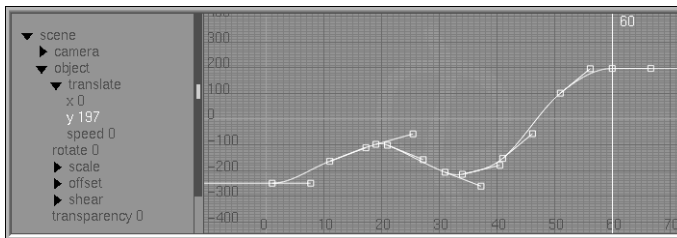
Use Simplify to change the number of keyframes in your animation curve or within a group of keyframes.

Simplify analyses the peaks and valleys in the selection and tries to generate keyframes with duration between them as specified in the Curve Value field.

For example, the following figure shows an animation curve with four keyframes. Simplify is applied with a simplification value of 2. The result is an animation curve with nine keyframes.



The curve before applying Simplify. Note that the curve only has four keyframes.



The curve after applying Simplify with a simplification value of 2.

To simplify a channel or a group of keyframes:

1. Select a channel or group of keyframes.
2. In the Curve Value field, enter the simplification value.

Use a large number to simplify the curve or group of keyframes. A small number increases the number of keyframes.

3. In the Curve Functions box, select Simplify.

Removing Jitter

Use Jitter to remove jitter from an animation curve or within a group of keyframes. Use the Curve Value field to specify the Over value. The Jitter option is typically used on shift data in the Stabilizer to remove jitter while keeping camera movement. See “Removing Jitter while Keeping Overall Motion” on page 809.

To remove jitter:

1. Select the channel or group of keyframes.
2. In the Curve Value field, specify the Over value.
3. In the Curve Functions box, select Jitter.

Applying an Average

Use Average to apply a weighted average to the selected channel or a group of keyframes. Enter the number of keyframes used to calculate the average in the Curve Value field. Average is typically used on shift data in the Stabilizer to remove camera movement while keeping jitter. See “Removing Jitter while Keeping Overall Motion” on page 809.

To average a channel or group of keyframes:

1. Select a channel or group of keyframes.
2. Specify the divisor in the Curve Value field.
3. Select Average in the Curve Functions box.

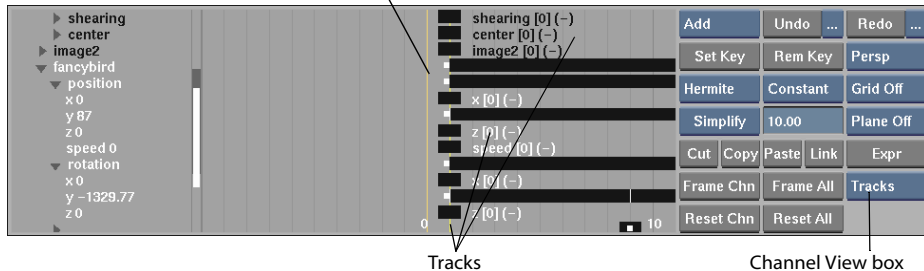
Using FrmSnap

Use FrmSnap to fix keyframes that fall between frames. This may occur after using other curve functions such as XScale, or when using the Track Editor to change the timing of an animation. To fix keyframes that fall between frames, select the channel or group of keyframes and select FrmSnap in the Curve Functions box.

Changing the Timing of an Animation

Use the Track Editor to change the global timing of an animation. You can stretch, compress, and/or offset individual channels or groups of channels interactively. The Track Editor is extremely useful for changing the length of an animation. For example, you can compress a 300-frame animation into a 280-frame animation without moving a single keyframe.

The vertical axis in the Track Editor represents the frame number in the animation sequence.



In the Track Editor, channels and folders are represented by horizontal bars called tracks. The channel name appears to the right of each track. The tracks appear in the same order as the channels in the channel hierarchy.

The position and length of a track is determined by the first and last keyframes for the channel it represents. The track extends from the frame number of the first keyframe to the frame number of the last keyframe. The frame numbers for the first and last keyframes appear to the right of the track and are shown in square brackets [1, 32], and the total number of frames is shown in parentheses (32). If a channel contains no keyframes, the number zero appears in square brackets [0] and a dash appears in parentheses (–) beside the track.

When you select a track, three handles appear as white squares on the track.



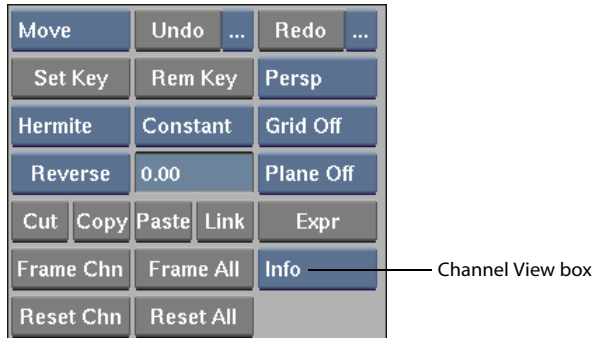
You adjust the timing and offset of an animation by stretching, compressing, and moving these tracks. After you change the timing of the animation, you may need to change the number of frames for the final clip in the Total Frames field. This is especially important when you want to increase the length of the animation.

Moving the handle of a folder affects all channels in the folder. For example, moving a handle of the Scene folder (the top level folder) causes all of the folders and channels in the hierarchy to be offset or scaled by the same amount. Changing the timing of the scene changes the timing of the whole animation.

You can use the Select, Pan, Zoom, and Rectangular Zoom edit modes in the Track Editor. Note that Add, Delete, Translate, Break, and Auto cannot be used.

To modify animation timing:

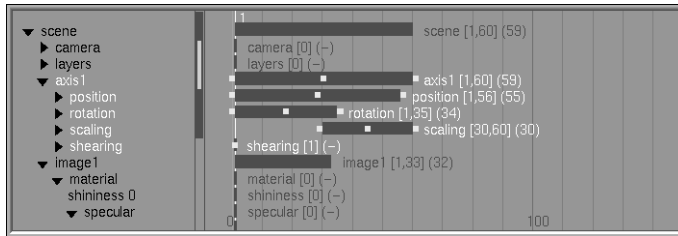
1. In the Channel View box, select Tracks.



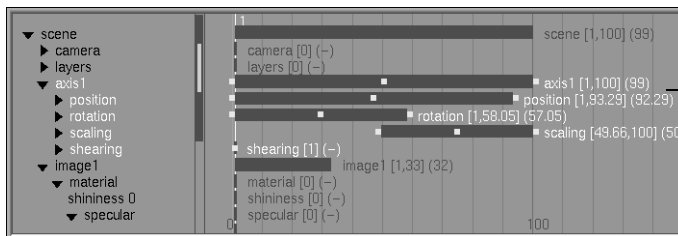
The Track Editor appears and the animation curves appear as tracks.

2. In the Edit Mode box, select Move.
3. In the Track Editor, select the track whose timing you want to modify.
Selection handles appear on the selected track.
4. To stretch, compress, or offset a track, drag one of the track handles as follows:
 - Drag the middle handle to offset a track. The frame numbers of the first and last keyframes are changed. The length of the track is unchanged.
 - Drag the left or right handle to scale the animation.
 - Drag the left handle to change the frame number of the first keyframe.
 - Drag the right handle to change the frame number of the last keyframe.

5. If you increase the length of the animation, change the number of frames in the Total Frames field to match the new number of frames in the animation.



The animation is initially 59 frames long, from frame 1 to 60.



Decimal value
indicates
keyframe is
between frames

The length of the animation is increased to 99 frames, frame 1 to 100.

Sometimes increasing or decreasing length of an animation results in keyframes between frames. When this happens to first or last keyframe in an animation, a decimal value is given to keyframe value. To snap a keyframe between frames, see “Using FrmSnap” on page 162.

Working with Expressions in the Channel Editor

You use expressions to apply animation dynamically across channels. For example, you can have a layered image behave as a function of another image. Expressions are available through all modules that use the Channel Editor.

Using expressions saves time since you can apply animation to one channel and have other channels behave in a similar way or out of phase automatically. Before writing your own expressions, you should understand the following:

- Rotation, Scale, and Shear parameters are composed of three vectors that are written using the convention (x, y, z) where x, y, and z are separate values. Position includes the speed parameter and is composed of four vector elements. For example, the Position parameters are expressed as follows:

- axis1.position.x
- axis1.position.y
- axis1.position.z
- axis1.position.speed

- For arithmetic operations and functions, be sure you specify values and vector values that are contained within an acceptable range. For example:

- For the channel `image1.material.transparency`, a valid expression would be `axis1.position.x/5`

- Some functions return values and other functions return vectors. When you use a function, you should verify that the returned value is in an acceptable range.

NOTE: Expressions override interpolation modes and previously set keyframes for a selected channel.

Consider the following examples:

- The position of `axis1` and `axis2` is the same and are aligned.

Channel:	Expression:
<code>axis2</code>	<code>align(axis1.position)</code>

- The transparency of `image1` is twice as much as `image2` across all keyframes.

Channel:	Expression:
<code>image2.material.transparency</code>	<code>image1.material.transparency*2</code>

- The position of `axis8` and `axis9` is the same, but delayed by 10 frames.

Channel:	Expression:
<code>axis9.position</code>	<code>eval(axis8.position, frame-10)</code>

- The animation of `axis2` and `axis1` is the same, but `axis2` is half the speed of `axis1`, making it twice as slow.

Channel:	Expression:
<code>axis2</code>	<code>eval(axis1, frame / 2)</code>

- A random positioning effect is created for `axis 1`.

Channel:	Expression:
<code>axis1.position.x</code>	<code>noise(frame) * 5</code>

- The X-axis follows animation position of the object. This function only works while the object is moving. When the object stops, the object snaps to the specified axis as shown for `axis18`.

Channel:	Expression:
<code>axis17</code>	<code>align(position)</code>

Channel:

axis18

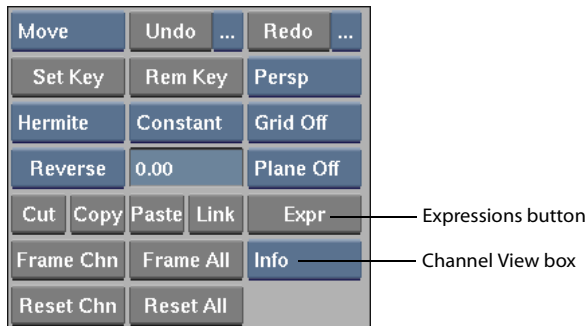
Expression:

align(position, (0, 1, 0))

You can also use arithmetic operators, conventions, constants, and functions in your expressions. See “Expression Reference” on page 169.

To apply an expression to a channel:

1. Click Anim to access the animation controls.



2. In the Channel View box, select Info.

The channels in the Channel Editor appear in a tabular format.

3. Select the channel to which you want to apply an expression.

Channel	Component	Value	# Keys	Expression
image13.material.transparency	transparency	0	0	eval(image1.material.transparency.frame-axis17.position.x*12)
image14.material.transparency	transparency	0	0	eval(image1.material.transparency.frame-axis17.position.x*13)
image15.material.transparency	transparency	0	0	eval(image1.material.transparency.frame-axis17.position.x*14)
image3.material.transparency	transparency	0	0	eval(image1.material.transparency.frame-axis17.position.x*2)
image4.material.transparency	transparency	0	0	eval(image1.material.transparency.frame-axis17.position.x*3)
image5.material.transparency	transparency	0	0	eval(image1.material.transparency.frame-axis17.position.x*4)
image6.material.transparency	transparency	0	0	eval(image1.material.transparency.frame-axis17.position.x*5)
image7.material.transparency	transparency	0	0	eval(image1.material.transparency.frame-axis17.position.x*6)
image8.material.transparency	transparency	0	0	eval(image1.material.transparency.frame-axis17.position.x*7)
image9.material.transparency	transparency	0	0	eval(image1.material.transparency.frame-axis17.position.x*8)
image10.material.transparency	transparency	0	0	eval(image1.material.transparency.frame-axis17.position.x*9)
axis1	axis1			
axis1.center	center			
axis1.center.x	x	0	0	
axis1.center.y	y	0	0	
axis1.center.z	z	0	0	
axis1.position	position			
axis1.position.speed	speed	0	0	
axis1.position.x	x	-212.401	2	
axis1.position.y	y	132	2	
axis1.position.z	z	0	2	

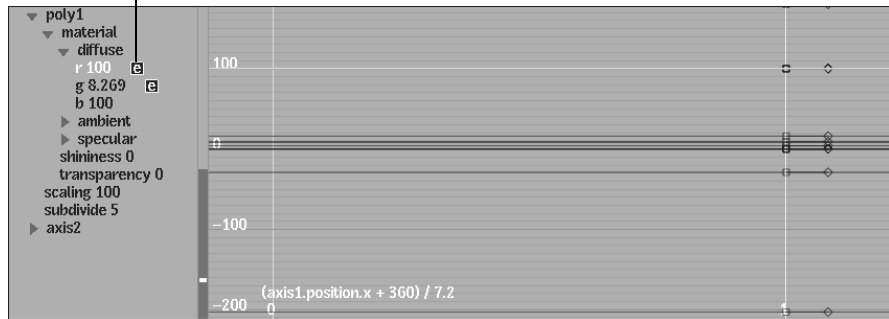
4. In the Animation menu, click Expr or press the expression hot key = .

The Expression field appears below the Channel Editor in the menu.

5. Enter an expression according to guidelines described in “Expression Reference” on page 169.

When you apply an expression to a channel, the letter 'e' appears next to the channel in the channel hierarchy.

Expression indicator



To modify an expression:

1. Click Anim and in the animation controls, select Info from the Channel View box.
The animation curve window changes to a tabular format.
2. Click the Expressions column heading to sort by expressions and view them more easily.
3. Select the expression you want to edit.
4. Click Expr.
The Expression field appears at the bottom of the animation controls.
5. Modify the expression and press **ENTER**.


The modified expression appears in the table.

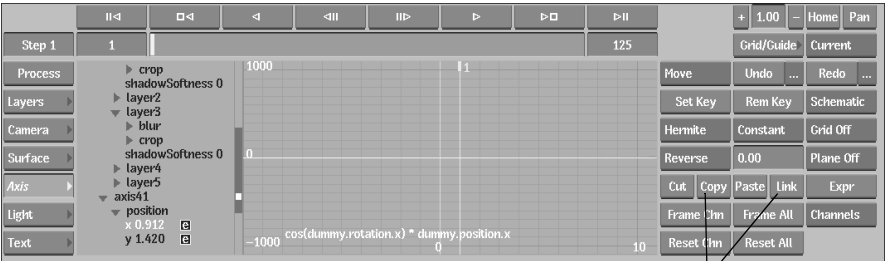
Linking Expressions to Other Channels

You can link the expression behaviour of one channel to another using Copy and Link. You can link different types of channels together. For example, the scaling of one layer can affect the rotation of another layer. Also, you can link layer blur to the position of an axis as shown in the following procedure. In this way, any change that occurs in the position of the axis is reflected dynamically in the layer blur.

To create an expression by linking one channel to another:

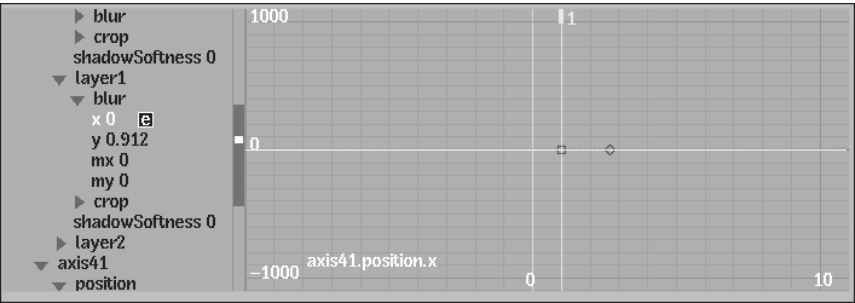
1. Click Anim to display the animation controls.
2. In the Channel View box, select Info.

3. To understand the syntax of the expression more clearly, click  to see the dot property format of the channel.



Copy and Link buttons

4. In the channel hierarchy, select a channel and click Copy.
For example, you can copy the x position of axis41 and link it to the X blur value of a layer.
5. Select the destination channel and click Link.
The following example illustrates the result of the copied channel. The X blur value of layer1 is linked to the X position value of axis 41.



Expression Reference

Arithmetic operators you can use in expressions are listed in the following table.

Operators:	Description:
*	Multiplication
/	Division
%	Modulus
(x,y,z)	Vector where x,y,z may also be the results of functions

Arithmetic and vector functions you can use in expressions are listed in the following table.

Expression:	Description:
<code>abs(x)</code>	Calculates the absolute value of x
<code>acos(a)</code>	Calculates the arccosine of a
<code>align(a)</code>	Generates a rotation vector based on an object's position to align the object with an axis
<code>asin(a)</code>	Calculates the arcsine of a
<code>atan(a)</code>	Calculates the arctangent of a
<code>ceil(x)</code>	Returns the largest integer greater than or equal to x
<code>cos(a)</code>	Calculates the cosine of a
<code>cross(v1,v2)</code>	Calculates the cross product of two vectors; returns a vector
<code>degrees(r)</code>	Converts r to degrees
<code>dot(v1,v2)</code>	Calculates the dot product of two vectors; returns a vector
<code>eval</code>	Evaluates another channel at a different point in time
<code>exp(x)</code>	Calculates the exponential function of x
<code>expm1(x)</code>	Calculates the equivalent to $\exp(x)-1$
<code>floor(x)</code>	Returns the smallest integer greater than or equal to x
<code>fnoise(v)</code>	Calculates the fractal noise vector v , returns a float
<code>length(p)</code>	Calculates the euclidean length of point p
<code>log(x)</code>	Calculates the natural logarithm of x
<code>log10(x)</code>	Calculates the base 10 logarithm of x
<code>log1p(x)</code>	Calculates the equivalent to $\log(1+x)$
<code>mod(x,y)</code>	Returns the remainder of dividing x by y
<code>noise(v)</code>	Calculates the noise of vector v , returns a scalar
<code>noise3(v)</code>	Calculates the noise of vector v , returns a scalar
<code>radians(a)</code>	Converts a converted to radians
<code>round(x)</code>	x rounded to the nearest integer
<code>sign(x)</code>	Returns +1 or -1 depending on the sign of x
<code>sin(a)</code>	Calculates the sine of a
<code>sqrt(x)</code>	Calculates the square root of x
<code>tan(a)</code>	Calculates the tangent of a
<code>trunc(x)</code>	Returns the integer value of x
<code>turbulence(v,o)</code>	Calculates the turbulence of vector v and octave o ; returns a float

Expression:	Description:
<code>turbulence3(v,o)</code>	Calculates the turbulence of vector <i>v</i> and octave <i>o</i> ; returns a vector

Sample Expression Setups

A few Action setups with sample expressions are provided in the directory `/usr/discreet/[product name]/example_setups`. To load one of the setup files, open Action with front, back, and matte clips and load one of the setup files. You may add or change clips according to the effect you want to produce. See “Saving, Loading, and Importing Setups” on page 899.

[illegible]

10

Monitor Calibration

So much gamma, so little time

Accurate colour reproduction is everything. If you want to see the real deal, it is essential that your monitor is calibrated correctly. Follow all the steps here to ensure that what you see is what you get.

Summary

In this chapter, you learn about:

“About Calibration” on page 174

“Black Level and Picture Level” on page 175

“Calibrating Your Monitor” on page 177

“Gamma” on page 179

“Monitor LUTs” on page 182

“Manual Calibration” on page 186

“Saving Your Setup” on page 191

“Customizing the Configuration File” on page 191

“Troubleshooting” on page 192

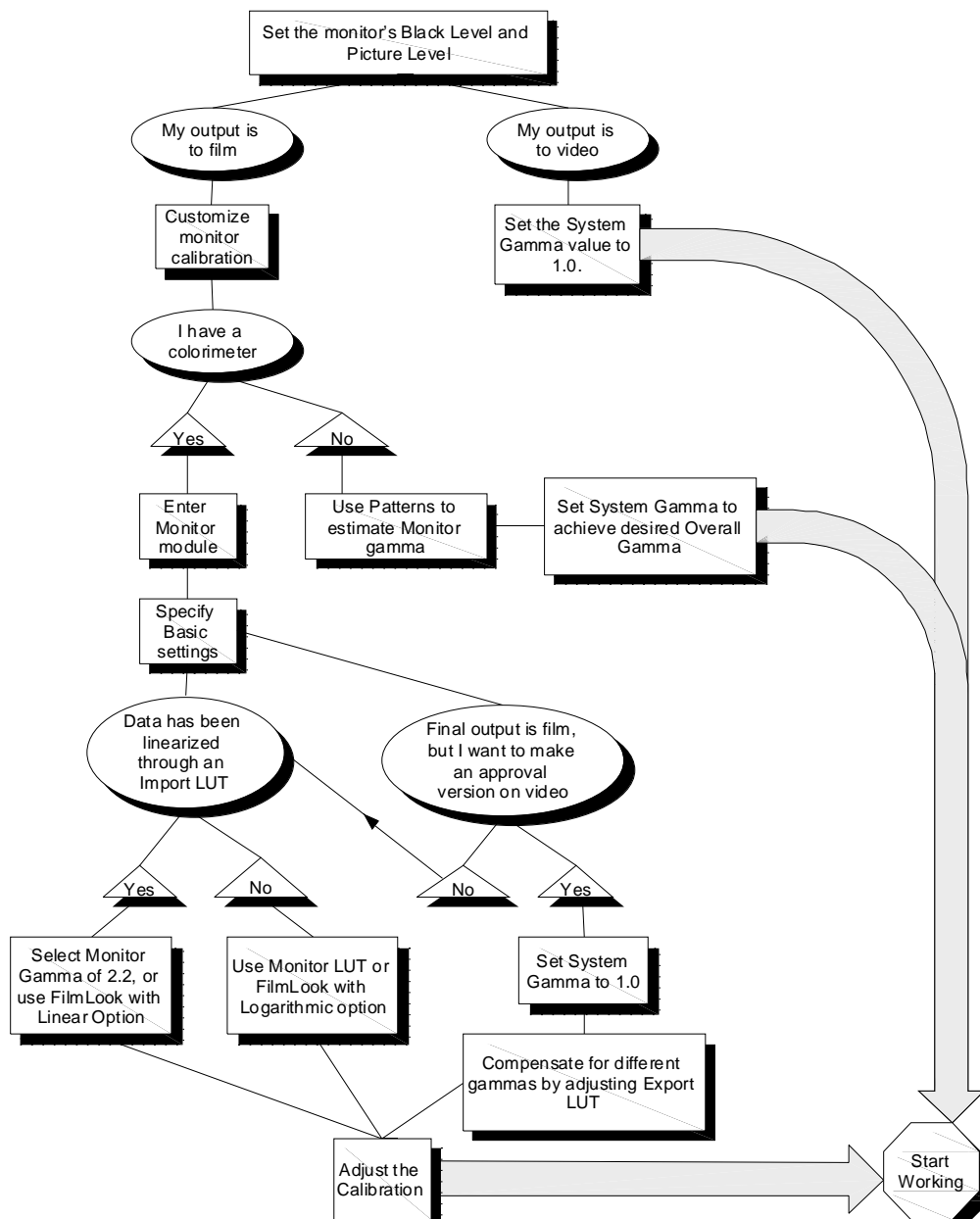
All monitors display colours differently. The colours that you see on your monitor change slightly over time because the intensity of light generated by Cathode Ray Tubes (CRTs) for a given input signal can drift. By calibrating the monitor, you ensure that the image on the screen is consistent from day to day. Proper calibration is also important if you want to work with the same project on different monitors.

If you are using a Barco monitor, you are prompted to recalibrate it after approximately 400 working hours. If you are using an SGI monitor, you will need to calibrate it more frequently because there is no internal feedback loop to maintain colour accuracy.

NOTE: flame processes an Internal Calibration on Barco monitors which means that the colour patches are generated within the display.

About Calibration

Before displaying your images, you should ensure that the monitor is properly calibrated to simulate the final viewing environment..



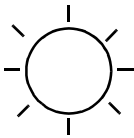
Black Level and Picture Level

The first step in calibrating your monitor is setting the correct Black Level and Picture Level. The Black Level controls the Black Level range, and the Picture Level controls the White Level range, or brightness of your monitor. When both Black Level and Picture Level are set correctly, you will achieve the most accurate colour reproduction and the optimum contrast ratio for the display.

Typically, you should only need to set these levels once. Once you have determined the correct Black Level and Picture Level, make sure the levels are not readjusted.

Black Level

Setting the proper Black Level for your monitor is important to ensure that true black at the monitor's input is accurately reproduced by your display. If the Black Level on your monitor is set too high, true black will appear slightly grey on your display. If the Black Level is set too low, the monitor will not be able to display all black levels in the lower range (such as shadows). On many monitors, the control for Black level has the following icon:

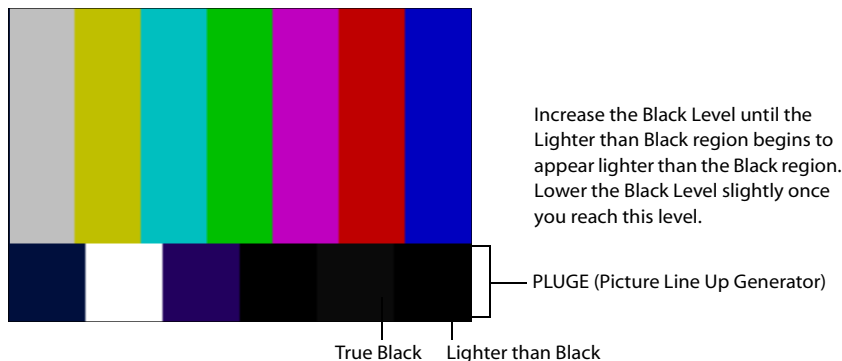


NOTE: This control is often labelled, "Brightness"; however, it does not actually modify the brightness at all. This control adjusts the contrast, or Black Level, of the display.

To set the Black Level:

1. Make sure your work environment is dark enough.
2. Set the Picture Level to the minimum value. See "Picture Level" on page 176.
3. Send a test bar pattern that contains a PLUGE or true black signal to your monitor.
4. Set the Black Level on the monitor to the minimum value.
5. Increase the Black Level until the true black signal starts to appear grey. This is the Black Level limit.

SMPTE Colour Bars (75%)



NOTE: If you are working with PAL, send a PLUGE only pattern to the monitor.

6. Set the Black Level slightly below the Black Level limit.
7. The Lighter than Black portion of the colour bar should be slightly lighter than the True Black region.

Picture Level

Setting the proper Picture Level for your monitor is important to ensure that true white at the monitor's input is accurately reproduced by your display. If the Picture Level is set too low, true white will appear off-white on your display. If the Picture Level is set too high, the monitor will not be able to display all white levels in the higher range (such as highlights). On many monitors, the control for Picture Level has the following icon:

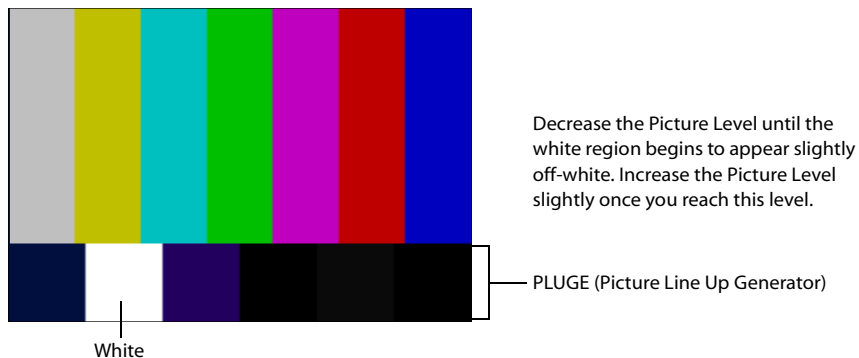


NOTE: This control is often labelled "Contrast"; however it does not actually modify the contrast at all. This control adjusts the brightness, or Picture Level of the display.

To set the Picture Level:

1. Make sure your work environment is dark enough.
2. Make sure the Black Level is set properly. If the Black Level is not set properly, see "Black Level" on page 175.
3. Send a test bar pattern that contains a PLUGE to your monitor.
4. Set the Picture Level to the maximum value.
5. Decrease the Picture Level until the true white signal starts to appear off-white. This is the White Level limit.

SMPTE Colour Bars (75%)



NOTE: If you are working with PAL, send a PLUGE only pattern to the monitor.

6. Set the Picture Level slightly above the White Level limit.

NOTE: Typically, changing the Picture Level should not affect the Black Level setting. If you notice that the Black Level has changed, you might need to slightly readjust the Black Level once you have set the appropriate Picture Level.

A more detailed explanation of Black Level and Picture Level appears in a paper written by Charles Poynton. You can locate his article at:

http://Home.InfoRamp.Net/~poynton/notes/black_and_picture/index.html

Colour Temperature

Colour temperature specifies certain sources of light energy, measured in Kelvin (K). A sunny day has a colour temperature of approximately 6500K. White light projected by a movie theatre projector has a colour temperature of approximately 5400K.

Choosing the correct colour temperature (or white point setting) for the monitor can help you emulate the final viewing conditions of a sequence of images. The way the human brain perceives a colour depends on the light source illuminating the image. For example, an object illuminated by daylight can look very different from the same object viewed in the dark conditions of a film theatre. By setting the colour temperature of the monitor, you can more closely align the viewing properties of the monitor to the final viewing conditions of the images with which you are working. Traditionally, images that will be viewed on film are best viewed on a monitor with a colour temperature setting of around 5400K because that is the approximate colour temperature of projected film in a dark theatre. For video footage or images that will be output to D1, monitor settings of around 6500K work well.

Calibrating Your Monitor

If you want to calibrate the monitor by having full control of the R, G and B channels, use the following procedures to perform an automatic calibration with the colorimeter, or manual calibration using the colour ramps.

You can calibrate your monitor the following two ways:

- Automatic calibration using the colorimeter.
- Manual calibration using the colour ramps and patterns.

To use automatic calibration, you must have either an Optisense device (built-in on most Barco monitors) or an X-Rite DTP 92 colorimeter.

When you calibrate the monitor manually, you apply a gamma correction based on your judgement. Manual mode does not make any hardware adjustments, even if you are using a Barco monitor.

It is highly recommended to use automatic calibration since human perception may vary from one person to the next. You should rely on objective measurements for monitor calibration.

Automatic Calibration with the Colorimeter

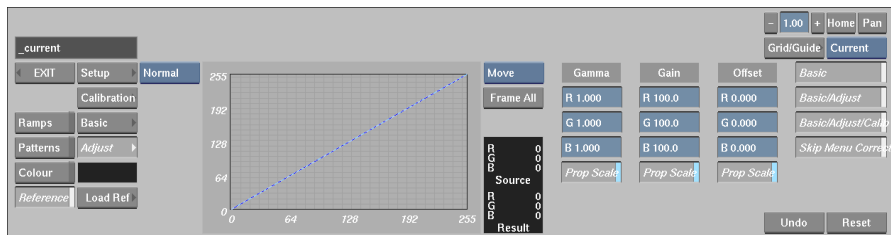
You must have an X-Rite DTP 92 colorimeter or a Barco monitor with the built-in Optisense device to use automatic calibration.

Automatic calibration is based on objective measurements from a sensitive hardware device.

To calibrate the monitor:

1. Click Monitor in the System menu.

The Monitor Calibration menu appears.



NOTE: If the message: “Moncal: No device available” appears, the configuration file must be updated to specify the colorimeter device. See “Customizing the Configuration File” on page 191.

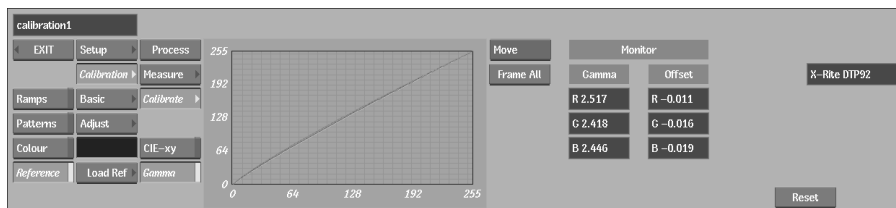
2. Click Calibration.
3. Click Calibrate.
4. Select the colorimeter you are using in the Colorimeter display.

If the colorimeter does not appear in the list, change the configuration file.

5. On Barco monitors, enter the appropriate colour temperature in the Temp display. The colour temperature should be in the 5400K° range for film and 6500K° for video.
6. Attach the colorimeter to the centre of the screen.
7. Click Process.

A series of colour patches appear on the screen. On Barco monitors, the cycle may be repeated several times and take a few minutes to complete.

The updated gamma curves appear on the graph.



8. If you want to see a CIE chromaticity diagram, click CIE-xy.

The spectral locus appears along with the chromaticity locations (R,G,B and W).

9. Load a reference image to see if you are satisfied with the calibration. See “Using a Reference Image” on page 189.

NOTE: Save the calibration setup if you want to use it in a later **flame** session. You can specify the calibration file to start up automatically in the *config* file, or you can load it the next time you start **flame**. See “Saving Setups and Preferences” on page 105.

Now that the monitor is calibrated, you can make adjustments to achieve the overall gamma of your final viewing environment. If your final output is to film, load a LUT or use FilmLook. If your final output is to video, you don not need to make any further adjustments.

Gamma

Once you calibrate your monitor, you need to match the display of the monitor to your final viewing environment, otherwise known as the Overall Gamma. You do this by using the tools in the Monitor module to adjust the calibration.

The Overall Gamma for video is 2.2. The Overall Gamma for film is approximately 1.5. It is important to set the correct overall gamma for viewing images. The Overall Gamma is the product of two Gammas:

- Monitor Gamma
- System Gamma

The gamma values have the following relationship:

$$OG = MG / SG$$

Where: **Is:**

OG Overall Gamma.

MG Monitor Gamma. This number represents the natural gamma response of the monitor’s RGB electron guns. This value is approximately equal to 2.2.

SG System Gamma. (Or the Gamma of the graphics card).

When comparing the same project on two monitors, the Overall Gamma on both displays must be equal.

If your final output will be on film, you want to achieve an Overall Gamma of approximately 1.5, assuming your working environment is dark enough. For film images, monitor calibration adjustments are therefore required. To achieve a specific Overall Gamma, **flame** modifies the System Gamma. Changing the System Gamma performs a gamma correction that compensates for the natural response of the monitor’s electron guns (Monitor Gamma). Once you are satisfied with the Overall Gamma, you can load logarithmic images (Cineon or DPX) using the

default LUT. See Chapter 16, “Image Import and Export.” If you are working with video, the images have already been gamma corrected at the camera and can be displayed on your monitor without any additional adjustments.

Working with Film Images

If your final output is to film, you can either load a Monitor LUT, or use the FilmLook tools to achieve the Overall Gamma of 1.5. Depending on your configuration and workflow, there are several methods you can use when working with film images.

You can load the logarithmic files with an Import LUT. Once the data has been linearized with the Import LUT, you can use the FilmLook tools to precisely match the output to the print.

If you do not use an Import LUT when loading the logarithmic images, you can use a Monitor LUT to view the images correctly or use the FilmLook tools.

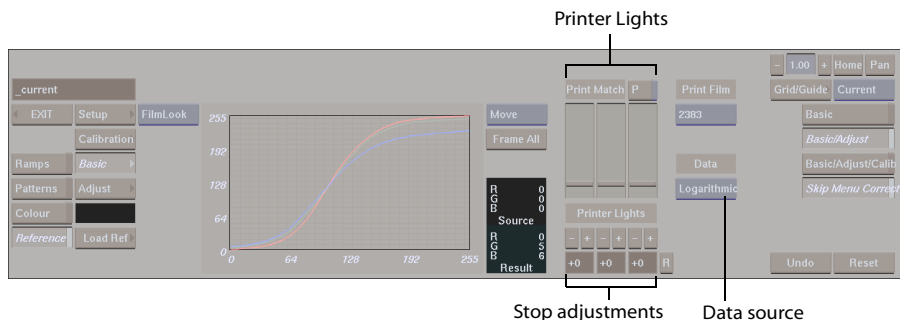
If your final is film, but you are making an intermediate version on video, you can set the System Gamma to 1.0 and modify the Export LUT to output the images correctly.

Using FilmLook

FilmLook is an advanced technique that allows you to precisely match the output to the final print based on predefined print film settings. You select the source of the image data and then select the final print type. You can then fine-tune the adjustment to suit your viewing environment. By selecting the final print type, you can accurately reproduce your final result. You can then use the printer lights to fine-tune the adjustments.

To use FilmLook:

1. Click Basic.
2. Select FilmLook.



3. Select the data source.

Select:	If the source data is:
Video	Video

Select:	If the source data is:
Linear	Linear data file or Log data loaded with Log to Lin Import LUT
Logarithmic	Log data loaded without an Import LUT

4. Select the final print type: (2383, 2393).
5. Adjust the R, G, and B, curves, by using the Printer Light controls, if necessary.
6. Save this setup so you can verify the print at a later time. See “Saving Your Setup” on page 191.

Once you are satisfied with the results of your FilmLook setup, use it as a reference when the print returns from the printer. You can then use the Stop Adjustment controls to verify the print with your display. If there is a discrepancy between your FilmLook setup and the print, you can use the Stop Adjustment controls to determine the adjustments the printer needs to make for the final output.

To use the Stop Adjustment controls:

1. Load the setup you used for the images you are verifying.
2. Load as a Reference the same image that appears on the print.
3. If there is a difference between the image on the monitor and the print, use the +/- Stop Adjustments to fine-tune the setup.

Each click of the + or - controls raises or lowers the Printer Light sliders by one point. Once you match the image on the monitor with the print, you can tell the printer by how many points each channel needs to be adjusted when making the final print.

Skip Menu Correct

Typically, you want to apply the gamma correction only to the image window, and not the entire interface. However, if you are making many modifications to the System Gamma, it takes longer for the system to calculate the data and then apply an inverse gamma correction to the rest of the interface to restore the buttons and background to their regular appearance. To facilitate fast changes to the System Gamma, enable Skip Menu Correct as you make the modifications. This applies the gamma correction to the entire interface. Once you are satisfied with the result, disable Skip Menu Correct so that the gamma correction is only applied to the image window.

Monitor LUTs

A Monitor LUT is a lookup table that is used for display purposes only and does not affect the image data during either import or export. Monitor LUTs are useful when you import a logarithmic image without an Import LUT. You can load a Monitor LUT to linearize the log data for the display and leave the original data unmodified. If you are using an Import LUT when you load the logarithmic image, you should not use a Monitor LUT. Using Import and Export LUTs is explained in Chapter 16, “Image Import and Export.”

Once your monitor is calibrated, you can load a Monitor LUT to approximate the final viewing conditions.

To load a Monitor LUT:

1. Click Basic.
2. Select LUT from the option box.
3. Click Load LUT.
4. Select the LUT from the file browser.



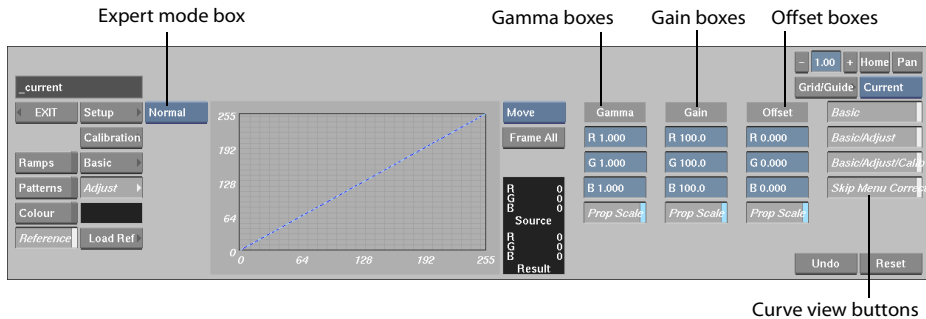
NOTE: Octane does not support Monitor LUTs with more than 256 entries. If you want to use a Monitor LUT that contains more than 256 entries such as the 10to8 LUT, use the following procedure to create a Monitor LUT that is recognized by the Octane.

To use a Monitor LUT with more than 256 entries:

1. In Batch, add a LUT Builder node.
2. Load your reference image.
3. In the Curves menu, change the LUT type to Log to Linear.
4. In the Export LUT menu, change the Source to 8 bit.
5. In the Export LUT menu, change the Destination to 8 bit.
6. In the Param menu, change the gamma correction type to Video.
7. Build your LUT to get the best results for the image.
8. Save the LUT.
9. You can now load your custom Monitor LUT into the Monitor module.

Adjusting the Monitor Calibration

Once you calibrate the monitor and specify the Basic settings such as FilmLook or loading a Monitor LUT, you may need to fine-tune the calibration. Use the Adjust menu to fine-tune the calibration.

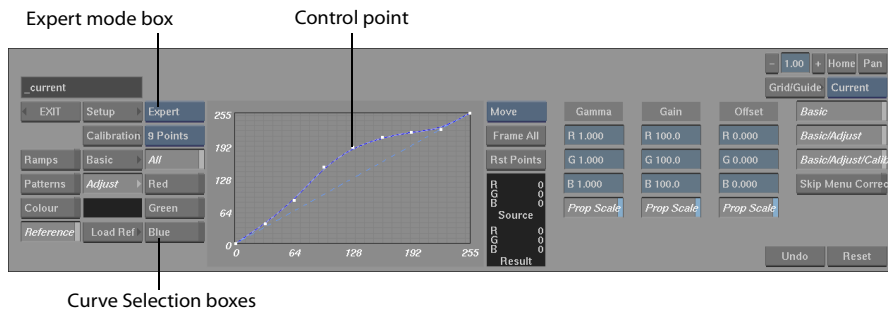


To adjust the calibration:

1. Click Load Ref and load a good reference image.
2. Enable the Curve view buttons for the curves you want to see.
Make sure either Basic/Adjust or Basic/Adjust/Calib is enabled if you want to see the Adjust curve change as you make modifications.
3. Modify the values in the Gamma, Gain and Offset boxes to get the result you want.

To use Expert Mode:

1. Select Expert in the Expert mode box.



2. Select either 9 Points or 3 Points to change the number of control points on the curve.
3. Enable the Curve selection box for the curve you want to modify.
4. Move the control points to change the shape of the curve.

Common Workflows

Because of different configurations and procedures within each facility, there are many different possibilities for adjusting your calibration and working with images for a film output. This section describes some typical workflows that you may use to achieve the best results. However, the Monitor module is designed so that you can customize it to your own needs. The following examples outline some procedures depending on your source material and the method you use for importing and exporting your image data. For detailed descriptions of each task, refer to the appropriate section in this chapter.

Logarithmic Data In, Film Out

If you are loading logarithmic images without an Import LUT to convert them to linear data and you want them to appear correctly on the monitor, you need to use a Monitor LUT, or FilmLook with the Logarithmic option.

To view log data with a Monitor LUT:

1. Load your log data without an Import LUT.
2. Enter the Monitor module.
3. Click Basic.
4. Select LUT.
5. Click Load LUT.
6. Select the LUT to load from the file browser.

NOTE: If you are loading a LUT that contains more than 256 entries, follow the workaround in “Monitor LUTs” on page 182.

7. Adjust the calibration in the Adjust menu, if necessary. See “Adjusting the Monitor Calibration” on page 183.

To view log data with FilmLook:

1. Load your log data without an Import LUT.
2. Enter the Monitor module.
3. Click Basic.
4. Select FilmLook.
5. Select Logarithmic.
6. Specify the FilmLook settings as described in “Using FilmLook” on page 180.
7. Adjust the calibration in the Adjust menu, if necessary. See “Adjusting the Monitor Calibration” on page 183.

Linear Data In, Film Out

If you are loading logarithmic images with an Import LUT to convert them to linear data and you want them to appear correctly on the monitor, you need to set a System Gamma of 2.2 or use FilmLook with the Linear option.

To view linear data by setting a System Gamma:

1. Load your linear data with an Import LUT. For example, if you are loading a 10 bit log file, use the 10to8 log file.
2. Click System on the desktop to view the System menu.
3. In the System Gamma box, select 2.2.
4. Click Monitor to enter the Monitor module.
5. Click Basic.
6. Make sure that Linear is selected.
7. Adjust the calibration in the Adjust menu, if necessary. See “Adjusting the Monitor Calibration” on page 183.

To view linear data with FilmLook:

1. Load your linear data with an Import LUT. For example, if you are loading a 10 bit log file, use the 10to8 log file.
2. Set the System Gamma to 1.0 in the System menu.
3. Enter the Monitor module.
4. Click Basic.
5. Select FilmLook.
6. Select Linear.
7. Specify the FilmLook settings as described in “Using FilmLook” on page 180.
8. Adjust the calibration in the Adjust menu, if necessary. See “Adjusting the Monitor Calibration” on page 183.

Logarithmic/Linear Data In, Video Out (Film Final)

It is possible to work with log images for a film output but create an intermediate copy of your work on video. In this case, there are some adjustments you need to make so that the images appear correctly on the monitor to simulate the final film look and are also output to the VTR correctly. To do this, you need to set the System Gamma to 1.0 as you normally would for output to video and then customize an Export LUT that converts the image data so that it appears correctly on the VTR. The custom Output LUT must compensate for the difference between the Overall Gamma for film (1.5) and the Overall Gamma for Video (1.0).

To use log or linear data for an intermediate video output:

1. Set the System Gamma to 1.0.
2. Linearize all Log data during import with an Import LUT.

NOTE: Make sure the Gamma Correction values in the Param menu of the Import LUT are set to 0.45.

3. Set the System Gamma value to 1.0 in the System menu.
4. Output your material to video.
5. Before going back to film, export the data with an Export LUT to compensate for the differences in gamma. Usually, the Export LUT is the inverse of the Import LUT you used to import the data.

HINT: It is recommended to create a few Export LUTs and prints to determine which one gives the best results.

6. Create a custom Output LUT to compensate for the difference in gamma.

Manual Calibration

If you do not have an X-Rite DTP 92 colorimeter or an Optisense device, you can use manual calibration. When you use manual calibration, you must first determine the Monitor Gamma of the CRT by using the Patterns in the Monitor module. Once you determine the MG, you can use the following equation to determine the value required for the System Gamma:

$$OG = MG / SG$$

There are two ways you can manually calibrate the monitor:

- Create a Monitor LUT in the Adjust menu of the Monitor module
- Assign a single value for the System Gamma and apply a gamma correction

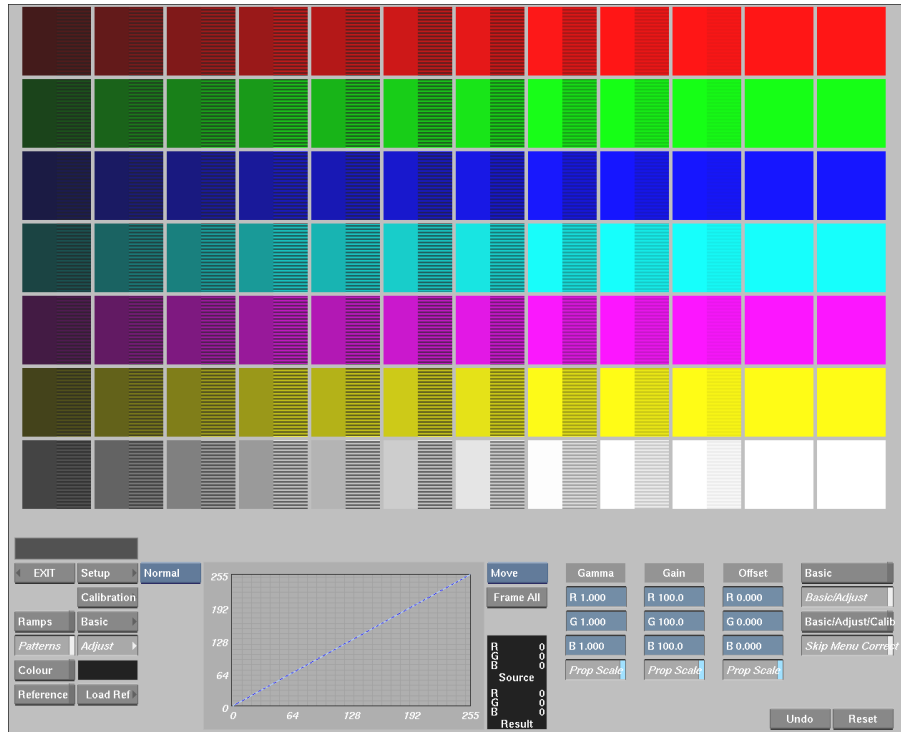
Determining Your Monitor Gamma

You estimate the Monitor Gamma of your monitor by using a series of reference patterns while adjusting the System Gamma. When the System Gamma is approximately equal to the Monitor Gamma there should be little or no change in brightness between the solid and striped regions of the reference patterns.

To calibrate manually:

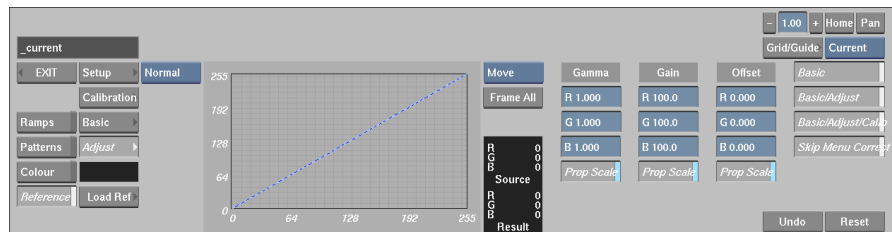
1. Click System on the desktop to enter the Monitor module.
2. Click Monitor.
The Monitor module appears.
3. Click Patterns.

A series of gradient coloured boxes appear. These are called Patterns. Each box is divided into two halves: a solid region and a striped region where the horizontal lines alternate between full intensity and black.



4. Click Adjust.

The Adjust menu appears.



5. While looking at the Patterns from an arms-length distance, adjust the values in the Gamma boxes until the two sides of the pattern have the same apparent brightness.

HINT: The gamma should be around 2.2 for a Barco monitor and about 2.4 for an SGI monitor.

You have now determined the Monitor Gamma. Use this value to either build a Monitor LUT or add a new System Gamma value for a gamma correction as explained in the following sections.

Creating a Monitor LUT for Manual Calibration

You can manually calibrate your monitor by creating a Monitor LUT with the tools in the Adjust menu.

If your final output is film, then you want to approximate an Overall Gamma of 1.5. If your estimated Monitor Gamma is 2.3 and you want an Overall Gamma of 1.5, then:

$$2.3/SG = 1.5$$

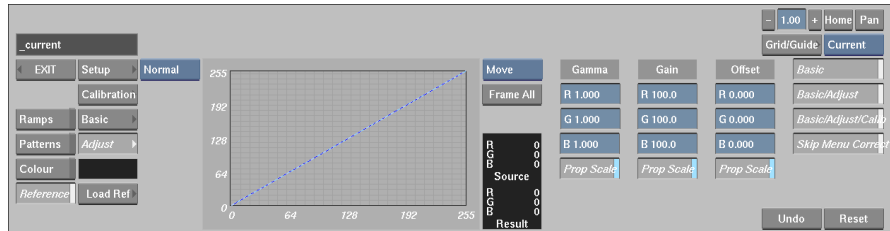
$$\therefore SG=1.53$$

Use the value 1.53 to create your Monitor LUT.

To create a monitor LUT:

1. Load a good reference image.
2. Click Adjust.

The Adjust menu appears.



3. Enable Prop Scale below the Gamma boxes.
4. Enter the value you calculated for the System Gamma in the Gamma boxes.
The image should appear correctly, although you may need to make some minor adjustments.
5. If you need to fine-tune one of the channels, disable Prop Scale below the Gamma box and adjust each gun individually.
6. You can further fine-tune the calibration by adjusting the values in the Gain and Offset boxes.

NOTE: If you modify the gain or offset values, you will clip the upper or lower RGB values.

7. Save the setup so you can use it at a later time. See "Saving Your Setup" on page 191.

Gamma Correcting the Monitor

If you do not want to use the Adjust menu to create a custom Monitor LUT for your monitor calibration, you can use a single System Gamma value to gamma correct the monitor. Gamma correcting the monitor in this way means that you can use the Adjust menu to fine-tune the calibration. However, a gamma correction uses the same value for each channel and may not provide the result you want.

If your estimated Monitor Gamma is 2.3 and you want an Overall Gamma of 1.5, then:

$$2.3/SG = 1.5$$

$$\therefore SG=1.53$$

The System Gamma value therefore equals 1.53. Once you determine this value, you can set the System Gamma value in the *config* file by adding it to the list.

If you are working with video, the images have already been gamma corrected at the camera and can be displayed on your monitor without any gamma correction.

Using a Reference Image

It is a good idea to check your gamma correction with a reference image. The image should be representative enough to judge the Overall Gamma.

To use a reference image:

1. Click Reference.
2. Click Load Ref.
The desktop appears.
3. Select the clip you want to use.

Displaying Ramps

To display ramps, click Ramp in the Calibration menu. Like Patterns, Ramps are useful to check the linear response of your monitor. If the System Gamma is equal to the Monitor Gamma (Overall Gamma of 1.0), the Ramps should appear as linear gradients.

Measuring a Colour

If you have a colorimeter, you can measure colours in Automatic Calibration mode. You can either measure a colour that you pick in a clip or one you define with the colour picker. You measure a colour to determine the standard CIE primaries (X,Y,Z), the chromaticity coordinates (x,y) and the colour temperature (T).

To measure a custom colour:

1. Click Measure in the Calibration menu.
2. Click in the colour patch next to the Colour button.
3. Use the colour picker to select a colour.
The colour appears in the image area.
4. Attach the colorimeter to the centre of the screen.
5. Click Process.

The X,Y,Z,x,y and T values appear.

NOTE: Sometimes when you measure a colour, the T field may say "error" instead of returning a colour temperature. This occurs when the (x,y) coordinates are too saturated to correlate and does not mean that the measurement did not work.

To measure a colour in a reference clip:

1. Click Measure in the Calibration menu.
2. Click Load Ref.
The desktop appears.
3. Select a clip from the desktop.
4. Click Reference in the Calibration menu.
The clip appears in the image area.
5. Click in the colour patch next to the Colour button.
6. Use the colour picker to select a colour from anywhere in the image.
The colour appears in the image area.
7. Attach the colorimeter to the centre of the screen.
8. Click Process.

Saving Your Setup

Save your setup if you have calibrated the monitor or created a custom Monitor LUT using the tools in the Basic and Adjust menus. You can load the same setup and continue working with the same project.

You can also share your setup with a different system to work with the same project. If you load a monitor setup from a different machine, you should recalibrate your monitor after loading setup.

To save your setup:

1. Click Setup.
2. Click Save As.
3. Type a name for the setup and save it in the file browser.

All information in the Monitor module is stored in a file with the extension `_current`.

To save a Monitor LUT:

1. Click Setup.
2. Click Export LUT.
3. Type a name for the Monitor LUT and save it in the file browser.

Only information in the Basic and Adjust menus are stored in the Monitor LUT.

Customizing the Configuration File

The following keyword appears in the configuration file:

- Gamma (For System Gamma, numerical value, LUT file, or monitor calibration setup file)
- MoncalDevice (For colorimeter device and port).

Setting the System Gamma

The System Gamma is used to apply a gamma correction through the graphics adapter of the SGI. If you have not yet determined the System Gamma, see “Creating a Monitor LUT for Manual Calibration” on page 188.

To set the System Gamma numerical value:

1. Search for the Gamma keyword in the *config* file.
2. Add the value of the System Gamma you want.

The new value is available in the System Gamma box in the System menu. The first three values are accessible with the hotkeys, **CTRL + SHIFT + 1**, **CTRL + SHIFT + 2**, and **CTRL + SHIFT + 3**.

Defining the Colorimeter

To use automatic calibration, you must define the colorimeter and its port in the configuration file. For detailed instructions on modifying the configuration file, see the *flame Installation Guide*.

To define the colorimeter and its port:

1. Search for the MoncalDevice keyword in the configuration file.
2. Uncomment the line that describes the device you want to use:

To use:	Remove the # sign in front of:
Optisense	MoncalDevice Barco Calibrator, /dev/ttyd2
X-Rite	MoncalDevice X-Rite DTP92, /dev/ttyd2

3. Change the port definition (dev/ttyd2) to match the port that you want to use.

Troubleshooting

The following tables discuss common problems and provide troubleshooting tips. If the problems persist, contact Customer Support.

Problem	I imported a Cineon image and it looks washed out. What’s going on?
Possible Cause	You need to use an Import LUT to convert image data from logarithmic to linear or use a Monitor LUT.
Solution	Try loading the image with one of the default LUTs. If you have an 8-bit partition, use the 10logto8 LUT. If you have a 12-bit partition, use the 10logto12 LUT.
Problem	I imported a Cineon image through the default LUT, and it looks too dark. Now what?
Possible Cause	You may not have set the correct System Gamma. To work with film images, you need to apply a gamma correction to your monitor in order to get an Overall Gamma of around 1.5.
Solution	Try a System Gamma value of 1.7. See “Setting the System Gamma” on page 191.



Section 2: Managing Clips

Organize your

clips with

libraries and

archives.

Transfer clips

to and from

devices.

Overview: The Library Menu

The big picture on the big pictures

*To work in **flame**, you need the means to bring media into **flame**, record it out again, store and organize your clips, and archive them at the end of a project. The Library menu provides the clip management tools to accomplish all of these essentials and more.*

Summary

In this chapter, you learn about:

- “The Library Menu” on page 195
- “Accessing The Library Menu” on page 196

The Library Menu

Use the Library menu to move media to and from the desktop. Clips can be loaded from an external device such as a VTR, processed within **flame**, and recorded out to an external device. The clips can be saved to a clip library, and can be saved as image files in the UNIX filesystem. The Library menu also includes facilities for archiving clips and linking audio tracks to clips.

The following chapters provide complete details on the modules of the Library menu:

Chapter 12, “Clip Libraries.”	Store desktops, reels, clips in clip libraries. Removing and loading entries from a clip library is also discussed, along with maintaining clip libraries. This chapter also shows you how to use wire to access clip libraries in other partitions or on remote systems.
Chapter 13, “Clip Input and Output.”	Record and load clips to and from an external device using the Input Clip, Output Clip, HDCAM Codec and DDR Library modules.
Chapter 16, “Image Import and Export.”	Transfer clips as image files to and from the file system using the Import Image and Export Image modules.
Chapter 17, “OMF Files.”	Use the Import OMF and Export OMF modules to transfer clips in OMF (Open Media Framework) format.
Chapter 18, “Archiving.”	Archive partitions, clip libraries, desktops, reels, and clips to a tape, VTR, or file. You can use Archive and Tape Library modules to archive material. This chapter also explains how to archive setups to a data tape or file.
Chapter 24, “Using Audio.”	Import audio files, record and play audio tracks, and how to link them to clips.

Accessing The Library Menu

To display the Library menu, click the Library button in the main menu.

LIBRARY				
EDITING	Load	Import Image	Input Clip	Delete
PROCESSING	Save	Export Image	Output Clip	Name
EFFECTS	Archive	Import OMF	DDR Library	Move
FORMAT	Network	Export OMF	Tape Library	Copy
SYSTEM		HDCAM Codec		Search

The following table briefly explains the purpose of the modules and commands accessed from the Library menu.

Use This Button: To:

Load	Load a previously saved desktop, reel, or clip from a clip library to the desktop.
Save	Save a desktop, reel, or clip to a clip library.
Archive	Archive partitions, clip libraries, desktops, reels, and clips to the filesystem or an external device, such as a VTR or DAT tape.
Network	Access clip libraries in other partitions or on remote systems (Discreet Filesystem and wire required).
Import Image	Load an image or a sequence of images from the file system onto the desktop.
Export Image	Transfer an image or a sequence of images from the desktop to the filesystem. The images can be saved in various file formats.
Import OMF	Load files in Open Media Framework (OMF) format from the filesystem to the desktop.
Export OMF	Export clips as OMF files from the desktop to the filesystem.
Input Clip	Load a clip from a VTR, VCR, camera, or other device.
Output Clip	Record a clip to a VTR, VCR, or other device.
DDR Library	Transfer clips to and from a DDR.
Tape Library	Archive clips to and from one or more DAT or Exabyte devices. Using this module, the tape device can run in the background.
Audio Library	Import audio files, and record, play, and link audio tracks to clips.
HDCAM Codec	Manually decode clips input using the HDCAM Codec option, and encode clips prior to output.

12

Clip Libraries

Great housekeeping

As you work on a project and clips accumulate on the desktop, you need a place to safely store your clips. Use clip libraries to store clips and keep them organized.

Summary

In this chapter, you learn about:

- “Framestore Basics” on page 198
- “The Clip Library Box” on page 199
- “Opening a Clip Library” on page 199
- “Saving Items to a Clip Library” on page 201
- “Clip Library Structure” on page 203
- “Displaying Entries in Clip Libraries” on page 205
- “Selecting Entries in a Clip Library” on page 212
- “Loading Entries from a Clip Library” on page 213
- “Transferring Clips Between Partitions” on page 214
- “Organizing Clips” on page 217
- “Clip Compatibility Between Products” on page 223
- “Using the Network Menu” on page 226
- “Hot Keys” on page 228

About Clip Libraries

Use clip libraries to store individual clips, all clips on a reel, or all clips on the desktop. Load any clip, reel, or desktop from a clip library for use in the current work session. You can also remove from a clip library any saved items that you no longer need.

HINT: The desktop includes all clips on all reels.

With **flame**, you can create multiple clip libraries to manage different projects. For example, you can create one clip library to save the clips that were used for a commercial spot, and another to save all the clips that were used to produce an effect such as a warp or a morph.

Each clip library that you create is stored in the current framestore partition. For each partition, **flame** also creates a clip library named *Default*.

NOTE: It is recommended to use several smaller clip libraries instead of one large one. This helps to keep your projects organized, allows for more efficient transfers of data, and streamlines clip library performance. Also, for optimum performance, ensure that the `MaxLibrarySize` token in the *init.cfg* configuration file is set properly (see the **flame** *Installation Guide* for details).

You can access clip libraries on other partitions (either local or remote) For more information, see “Using the Network Menu” on page 226. Note that to access remote machines, you must have **wire**.

Framestore Basics

References to the clips are stored in clip libraries on your computer’s hard disk under `/usr/discreet/clip/<volume_name>/<partition_name>`. The actual frames are stored on the framestore. If the clip reference you want is not present on the hard disk, you must import it from elsewhere on the file system or an outside source (such as a VTR). For more information, see Chapter 16, “Image Import and Export,” or Chapter 13, “Clip Input and Output.” When the framestore becomes full, or when an editing session is finished, you can export or archive clips to an external device. For more information, see Chapter 18, “Archiving.”

Each frame consists of:

- The image, which is composed of pixels.
- A frame ID.

Each clip consists of:

- Reference material, composed of names and numbers (such as timecodes and video and audio track numbers). The clip refers to the frame IDs.
- Audio tracks, if they have been linked to the clip.

You can load the same clip from a clip library multiple times onto the desktop. This repetition of the original frames does not take up space on the framestore because what you have actually loaded are references to the original frames. The reference does not occupy extra space on the framestore; however, if you perform an effect such as colour correction and then process new frames, these will use space on the framestore.

The Clip Library Box

The Clip Library box on the desktop shows the currently selected clip library. You can save clips to, and load clips from, the current clip library.

Clip Library box



The Clip Library box contains a list of all defined clip libraries. Use the Clip Library box to:

- Select a different current clip library.
- Create new clip libraries (see “Creating Clip Libraries” on page 218).

Opening a Clip Library

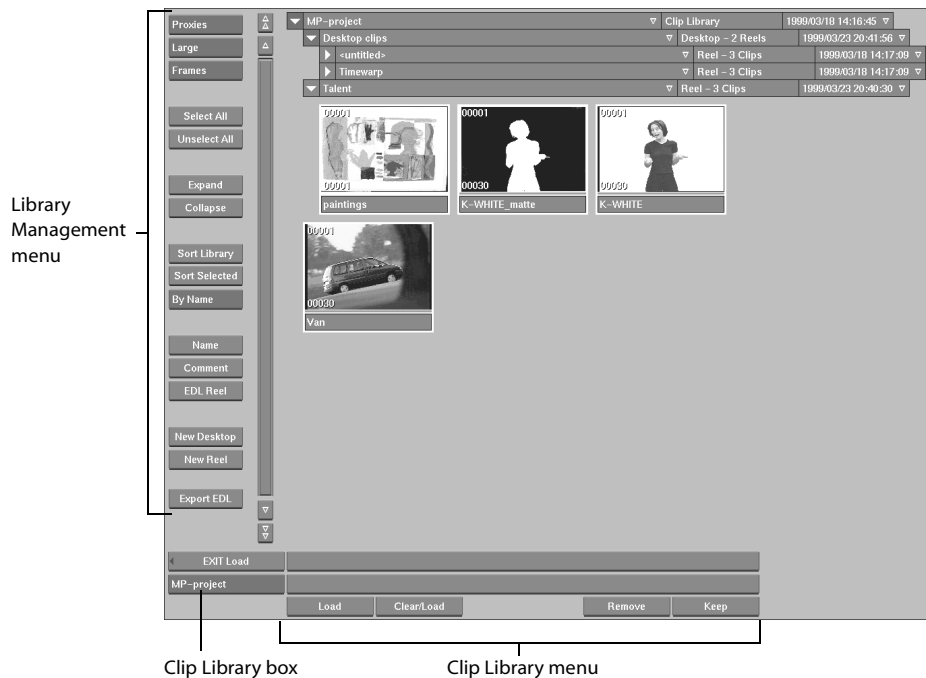
Open a clip library to select and load clips from it, or to organize your clips.

To open a clip library:

1. In the Clip Library box, select the clip library you want to open.
2. Click the Load button in the Library menu and select a destination. Alternatively, click on the Load button of the reel to which you want to load clips.

The current clip library opens.

NOTE: If the list of clip libraries is very long, and some libraries are not displayed, move the cursor just beyond the last one in the list, and the list will scroll up or down to reveal the remaining libraries.



To manage clips in a clip library, use the:

- Library Management menu to select entries and organize clips.
- Clip Library box to display a different clip library or create a new one.
- Clip Library menu to load selected clips to destination reel and remove clips from a clip library.

Information Bar

The information bar provides the following information:

- Software name and version number.
- Total number of frames in framestore and remaining frames in framestore.
- Percentage of memory remaining for increasing the size of the clip library. This memory is used for the clip library references (clip IDs) stored on the system disk. **flame** sets aside a certain amount of memory for this purpose when it loads. This amount is specified in the `MaxLibrarySize` token in the `init.cfg` configuration file.

NOTE: If the library exceeds the size set in the `MaxLibrarySize` token, the memory for the library is increased. If this occurs too many times in the same session, you will get a message stating that the library size is being exceeded, meaning that the memory is being fragmented. This is highly undesirable as **flame** could start swapping and run out of memory. In this case the `MaxLibrarySize` token must be increased.

Exiting the Clip Library

To exit a clip library without loading any clips, click the EXIT Load button.

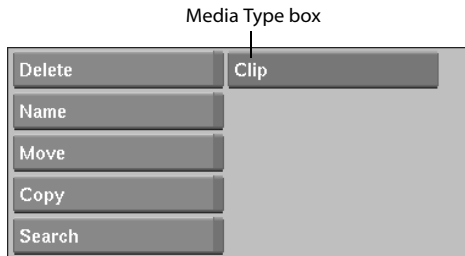
Saving Items to a Clip Library

Use the Save command in the Library menu to save a clip, reel, or desktop to the current clip library.

To save a clip, reel, or desktop:

1. From the Clip Library box on the desktop, select the clip library where you want to save the item.
2. Click Save in the Library menu.

The cursor changes to a red selection arrow and the Media Type box appears.



3. Select an option from the Media Type box.

Select:	To:
Clip	Save a clip or soft edit in the current clip library.
Reel	Save a reel in the current clip library.
Desktop	Save the desktop in the current clip library.

When you select Clip or Reel, the selection cursor remains. When you select Desktop, a Confirm button appears.

4. If you are saving a clip or reel, select the one you want to save:

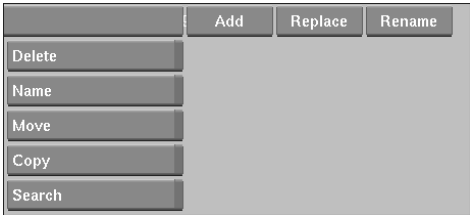
- To select a clip, click in its upper left corner.
- To select a reel, select any clip on the reel.

If you are saving a desktop, click the Confirm button. To cancel, click in a grey area of the menu.

The on-screen keyboard appears.

5. Type a name for the clip, reel, or desktop and click Enter. If the clip, reel, or desktop already has a name, the name appears in the keyboard window. You can change the name, or just click Enter to accept the existing one.

If there are already one or more entries in the clip library with the same name and of the same type that you are saving, a warning message states that an entry with the same name already exists and the Add, Replace, and Rename buttons appear.

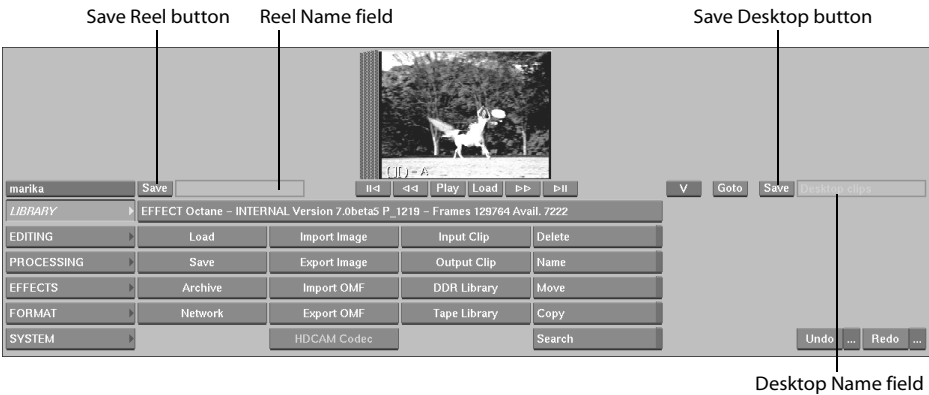


Click:	To:
Add	Add clip, reel, or desktop to the clip library. You will have two entries with the same name in the clip library.
Replace	Replace existing entry or entries with clip, reel, or desktop you are saving.
Rename	Enter a different name for clip, reel, or desktop you are saving.

To cancel the operation, click elsewhere in the menu.

Shortcuts for Saving Clips

Under each reel and in the lower right corner of the desktop there are Save buttons.



Use the Save Reel button to save all the clips in a particular reel to the current clip library, and the Save Desktop button to save all the clips on the desktop.

To save a reel or desktop:

1. Click the Save Reel or Save Desktop button.

The on-screen keyboard appears. If the reel or desktop already has a name, it is displayed in the Name Entry field; otherwise, the Name Entry field is blank.

2. Type a new name or accept the existing one and click Enter.

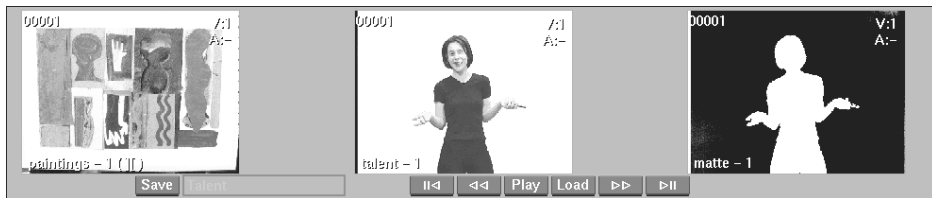
The reel or desktop is saved in the current clip library.

Clip Library Structure

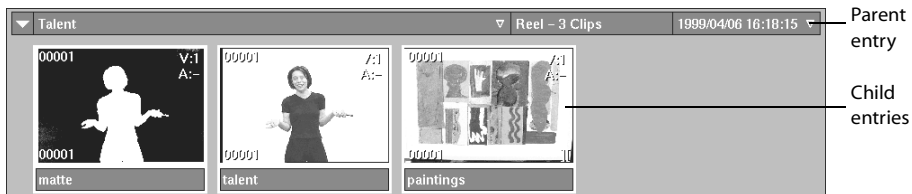
In clip libraries, the hierarchical relationship among groups of clips is represented using indentation. Entries that “belong” to other entries—child entries—are indented and are shown below their parent entries.

For example, if you save a reel, the clips within it are shown within a reel entry. The following figures show the result of saving a reel to a clip library.

The reel (named “Talent”) with three clips



The “Talent” reel saved in a clip library

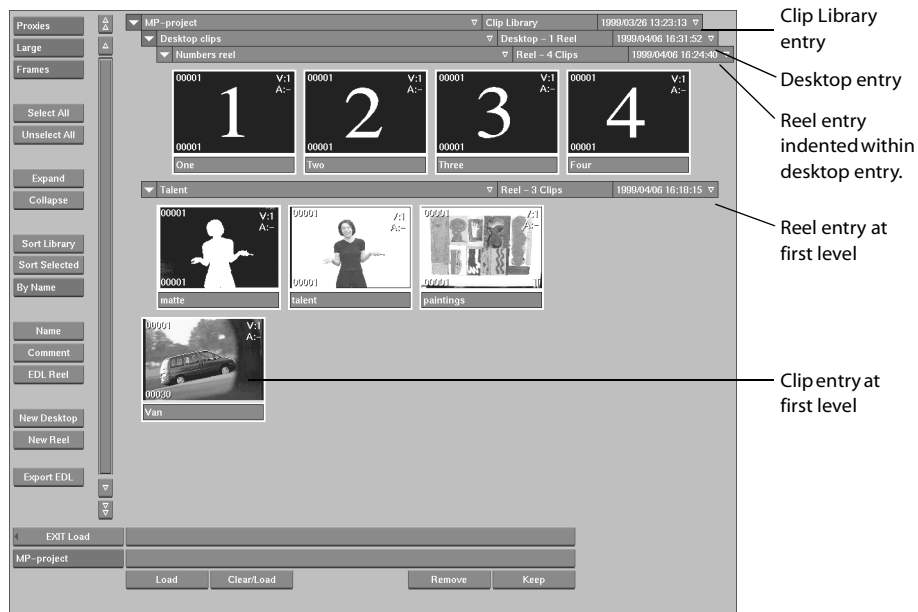


Parent and Child Entries

As shown in the previous example, an entry that contains other entries is called a parent entry, and an entry within a parent is called a child entry. A desktop, reel, and clip can all appear at the first level of indentation. This is because the indentation represents a parent-to-child relationship. If you save a reel alone, the reel appears at the first level in the clip library. Similarly, if you save an individual clip, the clip appears at the first level.

The following is a list of possible parent entries and the child entries they can contain.

Parent:	Can Contain:
Clip Library	Child entries for desktops, reels, or clips saved independently to the library.
Desktop	Child entries for each reel that contained clips when the desktop was saved.
Reel	Child entries for the clips and soft edits on the reel when it was saved.
Soft Edit	Child entries for the source clips used to create the soft edit.



Entry Colours

Each type of entry uses a different colour in the box at the left of the entry. In parent entries, this box also contains the Expand or Collapse arrow.

The colour:	Represents:
Dark Grey	Clip libraries.
Blue	Desktops.
Green	Reels and Source Areas (from Editing products).
Red	Record Areas (from Editing products).
Medium Grey	Clips and soft edits.
Dark Grey	Clips from Editing products.

Displaying Entries in Clip Libraries

Use the controls in the Library Management menu to display entries to best suit the operation you are performing.

Scrolling a Clip Library

Scroll the contents of the current clip library using the slider on the scroll bar. You can also use the incremental scrolling buttons (hold down any arrow key to scroll continuously).

Click:

To:

Scroll up a page.

Scroll up by one entry.

Scroll down by one entry.

Scroll down a page.

Changing the View Mode

You can view clip libraries in Proxies, Titles, or List View mode. Switch between modes by selecting a view in the View Mode box.

View Mode box

Proxies

Large

Timecodes

Select All

Unselect All

Titles Mode

In Titles mode, entries are presented as a series of horizontal bars. The bars consist of several arrow buttons and fields.

Collapse arrow

Name Sort arrow

Date Sort arrow

▼ Talent

Reel - 3 Clips

1999/04/06 16:18:15 ▼

Name

Description

Creation Date

Expand or Collapse Arrow (on parent entries) — Click the Expand or Collapse arrow to expand or collapse the child entries immediately below a parent entry. See “Expanding and Collapsing Entries” on page 212 for details.

The colour of the box surrounding the arrow indicates the type of entry (see “Entry Colours” on page 204 for details). The colour of the arrow itself indicates the selection of child entries (see “Selecting Entries in a Clip Library” on page 212 for details).

Name field — Contains the entry’s name. If a name is not specified when an entry is saved, this field is labelled “<untitled>”. For information on renaming entries, see “Renaming Entries” on page 219.

Name Sort Arrow (on parent entries) — Click to alphabetically sort the immediate child entries of a parent entry by name (that is, the child entries directly below the parent entry). To sort all child entries below the parent entry, **CTRL**-click this button.

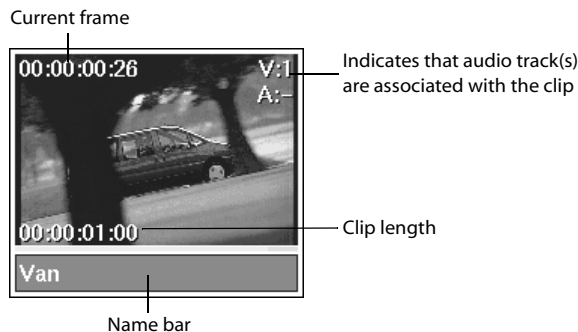
Description field — Contains a brief description of the entry. For example, a reel is described as “Reel - *n* Clips” where *n* is the number of clips in the reel.

Creation Date field — Provides the date and time the entry was created.

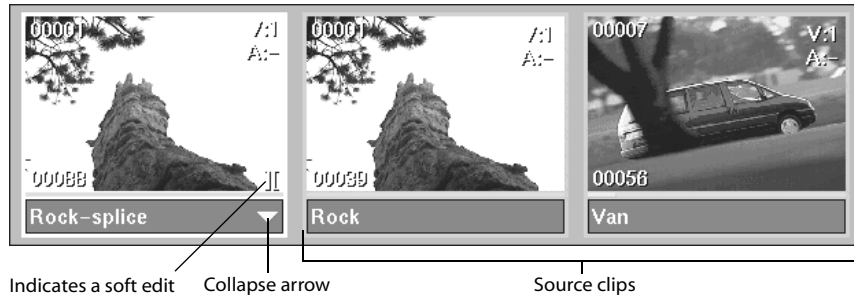
Date Sort Arrow (on parent entries) — Click to sort the immediate child entries of a parent entry by creation date (that is, the child entries directly below the parent entry). To sort all child entries below the parent entry, **CTRL**-click.

Proxies Mode

In Proxies mode, clips and soft edits are shown as proxies (small images). Information regarding the clip is shown on the proxy.



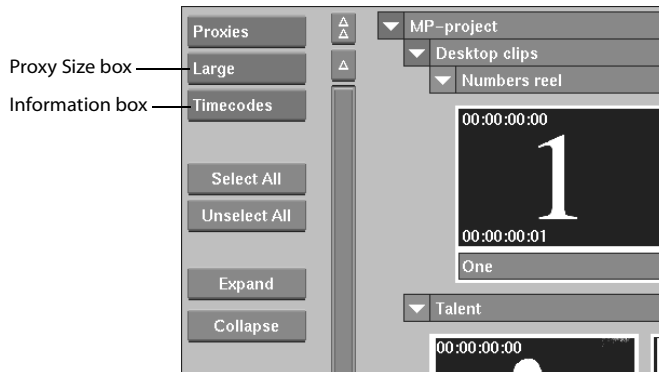
Proxies for Soft Edits — Proxies for clips with soft edits (such as uncommitted cuts, dissolves, or splices) have a “[]” label. If the clip has source clips, the proxy also has an Expand/Collapse arrow. Click the arrow to display or hide the source clips.



Proxies for source clips are identified by a dark grey border (proxies for regular clips have a light grey border).

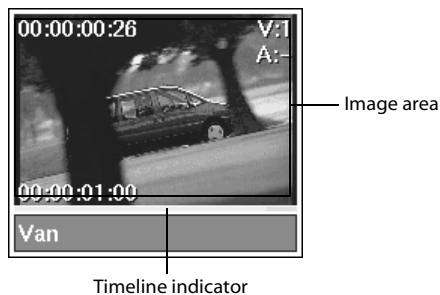
NOTE: If a portion of a soft clip resides on a remote partition, a black proxy appears with the text “Not Transferred”. You must load the clip to the local partition to view it in the library.

You can control the amount and type of information displayed on proxies using the Information box. To change the proxy information, select Frames, Timecodes, or No Display.



You can also change the size of proxies by selecting Large, Medium, or Small in the Proxy Size box.

Scrubbing a Proxy — You can view the contents of the clip by pressing on the proxy in the image window and dragging left or right. A yellow bar under the image, the timeline indicator, shows the approximate position of the frame being shown.



List View Mode

List View mode displays entries in the clip library as a list of clips. The list gives extra information concerning each clip, such as start timecode, clip length, and EDL reel.

List View

Location	Clip	In	Out	Duration	#Frames	#V	#A	#E	Created	Tape	Comment
mp-clips->	Rock	00:00:00:00	00:00:00:01	00:00:00:01	00:00:00:01	1	0	1	1999/08/02 12:38:41		
mp-clips->	Lion	00:00:00:00	00:00:00:10	00:00:00:10	00:00:00:10	1	0	1	1999/08/02 12:36:52		
mp-clips->	cat	00:00:00:00	00:00:00:01	00:00:00:01	00:00:00:01	1	0	1	1999/08/02 12:39:09		
Desktop clips->->output3		00:00:00:00	00:00:00:14	00:00:00:14	00:00:00:14	1	0		10:54		
Desktop clips->->Horse_Vipe	Horse_Vipe	00:00:00:00	00:00:00:28	00:00:00:28	00:00:00:28	1	0		2000/05/29 12:17:56		
Desktop clips->->Horse_Vipe	hellfire.aiff	00:00:00:00	00:00:27:29	00:00:27:29	00:00:00:00	0	1		10:18		
Desktop clips->->Horse_Vipe	hellfire.aiff	00:00:00:00	00:00:27:29	00:00:27:29	00:00:00:00	0	1		10:18		
Desktop clips->	RED	00:00:00:00	00:00:06:04	00:00:06:04	00:00:00:01	1	0	1	2000/05/29 10:25:02	<n/a>	
Desktop clips->	output3	00:00:06:04	00:00:06:18	00:00:00:14	00:00:00:15	1	0	3	10:54	<n/a>	
Desktop clips->	output3	00:00:00:00	00:00:00:14	00:00:00:14	00:00:00:14	1	0	1	11:10		
Desktop clips->	output3	00:00:00:00	00:00:06:04	00:00:06:04	-00:00:00:01	1	0	1	10:54	<n/a>	
Desktop clips->	Horse_Vipe	00:00:00:00	00:00:00:28	00:00:00:28	00:00:00:28	1	2	3	2000/05/29 12:17:40	<n/a>	
Desktop clips->	BLUE	00:00:00:00	00:00:06:04	00:00:06:04	00:00:00:01	1	0	1	2000/05/29 10:24:58	<n/a>	

The names of the parent entries of clips are shown in the Location column. If a parent entry has no clips in it, it does not appear in List View.

Like Titles and Proxies modes, clips only appear in List View mode if their parent entry is expanded (see “Expanding and Collapsing Entries” on page 212). This allows you to control the set of clips that are displayed (by expanding only those parent entries whose clips you wish to view). You can use the Expand and Collapse buttons in List View mode.

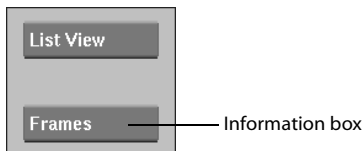
You can select and load entries from List View. However, you cannot perform drag and drop operations such as moving or copying entries in List View.

Clip Information in List View — The following information is provided in List View.

Column:	Displays:
Location	Path to the clip in the clip library.
Clip	Name of the clip.
In	Start timecode for the clip.
Out	End timecode for the clip.

Column:	Displays:
Duration	“Out” minus “In”.
# Frames	Number of frames referenced by a clip.
V#	Number of video tracks.
A#	Number of audio tracks.
E#	Number of softedits.
Created	Creation date and time of the clip.
@	Indicates that a clip has been saved to an archive tape.
Tape	Name of the associated EDL reel. <N/A> indicates that the reel cannot be renamed (see “Renaming a Clip’s Tape in List View” on page 210).
Comment	Comment for the selected clip.

Information Box — Lets you control whether the clip length is displayed in frames or timecodes. For frames, select either Frames or No Display.



Sorting Clips in List View — List View provides you extra sorting capabilities: you can sort on any field. Simply click on the label of any field to sort by that field. The sort field is highlighted and contains a sort arrow at the right end of the label.

Description field

Sort arrow

Location	Clip	In	Out	Duration	#Frames	#V	#A	#E	Created	@	Tape	Comment
imp-clips->	Rock	00:00:00:00	00:00:00:01	00:00:00:01	00:00:00:01	1	0	1	1999/08/02 12:38:41			
imp-clips->	Lion	00:00:00:00	00:00:00:10	00:00:00:10	00:00:00:10	1	0	1	1999/08/02 12:36:52			
imp-clips->	cat	00:00:00:00	00:00:00:01	00:00:00:01	00:00:00:01	1	0	1	1999/08/02 12:39:09			
Desktop clips->->output3		00:00:00:00	00:00:00:14	00:00:00:14	00:00:00:14	1	0		10:54			
Desktop clips->->Horse_Yipe	Horse_Yipe	00:00:00:00	00:00:00:28	00:00:00:28	00:00:00:28	1	0		2000/05/29 12:17:56			
Desktop clips->->Horse_Yipe	hellFire.aiff	00:00:00:00	00:00:27:29	00:00:27:29	00:00:00:00	0	1		10:18			
Desktop clips->->Horse_Yipe	hellFire.aiff	00:00:00:00	00:00:27:29	00:00:27:29	00:00:00:00	0	1		10:18			
Desktop clips->	RED	00:00:00:00	00:00:06:04	00:00:06:04	00:00:00:01	1	0	1	2000/05/29 10:25:02			<n/a>
Desktop clips->	output3	00:00:06:04	00:00:06:18	00:00:00:14	00:00:00:15	1	0	3	10:54			<n/a>
Desktop clips->	output3	00:00:00:00	00:00:00:14	00:00:00:14	00:00:00:14	1	0	1	11:10			
Desktop clips->	output3	00:00:00:00	00:00:06:04	00:00:06:04	-00:00:00:01	1	0	1	10:54			<n/a>
Desktop clips->	Horse_Yipe	00:00:00:00	00:00:00:28	00:00:00:28	00:00:00:28	1	2	3	2000/05/29 12:17:40			<n/a>
Desktop clips->	BLUE	00:00:00:00	00:00:06:04	00:00:06:04	00:00:00:01	1	0	1	2000/05/29 10:24:58			<n/a>

The sort can be either an ascending or descending. The arrow points down for an ascending alphanumeric sort and up for a descending sort. To toggle between an ascending and descending sort, click on the field label a second time.

Secondary Sort — When you perform a sort on any field other than the Clip field, you automatically get a secondary sort on the Clip field. This means that you can sort by one field—the primary sort—and get a sort of the clip names within the other field—the secondary sort. For example, if you sort by the Location field, you get a secondary sort by the clip name. Clips would then be sorted within their parent entries. Any field can act as a primary sort field, but only the Clip field can be the secondary sort field.

To sort by clip name under another field:

1. Click the Clip label for an ascending sort. Click it a second time for a descending sort.
2. Click the label of the field within which you want clip names to be sorted.

A sort arrow appears on both the fields. The last field you clicked is highlighted, indicating that this is the primary sort field.

		Primary sort on Clip field			Secondary sort on # Frames field							
Location	Clip	In	Out	Duration	#Frames	#V	#A	#E	Created	@ Tape	Comment	
Desktop clips->	output3	00:00:00:00	00:00:06:04	184	-1	1	0	1	10:54	<n/a>		
Desktop clips->->Horse_Vipe	hellFire.aiff	00:00:00:00	00:00:27:29	839	0	0	1		10:18			
Desktop clips->->Horse_Vipe	hellFire.aiff	00:00:00:00	00:00:27:29	839	0	0	1		10:18			
Desktop clips->	BLUE	00:00:00:00	00:00:06:04	184	1	1	0	1	2000/05/29 10:24:58	<n/a>		
mp-clips->	cat	00:00:00:00	00:00:00:01	1	1	1	0	1	1999/08/02 12:39:09			
Desktop clips->	RED	00:00:00:00	00:00:06:04	184	1	1	0	1	2000/05/29 10:25:02	<n/a>		
mp-clips->	Rock	00:00:00:00	00:00:00:01	1	1	1	0	1	1999/08/02 12:38:41			
colours->WHITE		00:00:00:00	00:00:00:00	0	1	1	0		14:39			
colours->WHITE		00:00:00:00	00:00:00:00	0	1	1	0		14:39			
colours->WHITE		00:00:00:00	00:00:00:00	0	1	1	0		14:39			
colours	WHITE	00:00:00:06	00:00:03:00	84	3	1	0	5	14:39	<n/a>		
mp-clips->	Lion	00:00:00:00	00:00:00:10	10	10	1	0	1	1999/08/02 12:36:52			
Desktop clips->	output3	00:00:00:00	00:00:00:14	14	14	1	0	1	11:10			
Desktop clips->->output3		00:00:00:00	00:00:00:14	14	14	1	0		10:54			
Desktop clips->	output3	00:00:06:04	00:00:06:18	14	15	1	0	3	10:54	<n/a>		
Desktop clips->->Horse_Vipe	Horse_Vipe	00:00:00:00	00:00:00:28	28	28	1	0		2000/05/29 12:17:56			
Desktop clips->	Horse_Vipe	00:00:00:00	00:00:00:28	28	28	1	2	3	2000/05/29 12:17:40	<n/a>		

NOTE: The default sort when you display List View is a primary sort on Location and a secondary sort on Clip.

Changing Column Width — List View is useful if you have clips with very long clip names because you can resize any column to display more or less information. Place the cursor between any two columns in the column name area to bring up the column resize cursor. Drag the cursor to resize the column.

Renaming a Clip's Tape in List View

You can name or rename the tape (or physical reel) of a captured clip or a pure source clip (such as a clip rendered in **flame**) in List View. This is useful when a manually captured clip has been given the wrong tape name. Also, use this method to assign the correct tape name to a 24 fps clip created from an original 30 fps with a manual hard-committed timewarp (as described in “Capturing with an EDL” on page 310).

You cannot rename the tape for any clip in the clip library (<N/A> appears in the Tape column if you cannot rename the tape). The structure of the clip must resemble that of a captured clip, so that it can be associated with the EDL. Use the following guidelines to determine whether you can assign the tape name to a particular clip.

Video clip with no audio: — The clip can be a captured clip or a pure source clip (a clip rendered within **flame**). Virtual source clips (for example, an uncommitted colour clip created with the Colour Source tool or a multitrack containing a virtual source) cannot be assigned a tape name.

Video clip with audio: — The start and end timecodes of the audio and video must be the same. Also, if the video *or* audio have been rendered/mixed down, the clip cannot be assigned a tape name. However if *both* video and audio have been rendered/mixed down, the clip can be assigned a tape name.

To name or rename the tape for one or more clips:

1. In the clip library, select List View from the View Mode box.

View Mode box
Tape name

List View	Δ	Location	Clip ▾	In	Out	Duration	#Frames	#V	#A	#E	Created	@	Tape	Comment
		Desktop clips->	BLUE	00:00:00:00	00:00:06:04	184	1	1	0	1	2000/05/29 10:24:58			
		mp-clips->	cat	00:00:00:00	00:00:00:01	1	1	1	0	1	1999/08/02 12:39:09			

2. Select the clip(s).
3. Click the Tape button and then click Confirm.
4. Enter a name of up to seven characters using the on-screen keyboard.

The name is applied to all selected clips. If there are clips in the selection that do not conform to the requirements for renaming the tape, they are not renamed. A message informs you of how many clips were renamed.

NOTE: You can also rename the tape from Proxies or Titles view, but since there is no Tape field in these views, you cannot see the result.

Adding a Comment to a Clip

This procedure lets you add a comment to a clip.

To add a comment to a clip:

1. In the clip library, select List View from the View Mode box.
2. Click the Comment button.
3. Enter a text string using the on-screen keyboard.

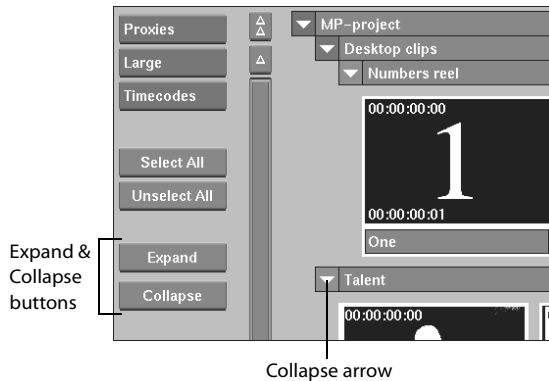
The text string appears in the Comment column for the selected clip.

Exporting an EDL

For information about exporting an EDL, see “Exporting an EDL” on page 299.

Expanding and Collapsing Entries

Click the Expand button to expand all selected parent entries except for soft edit entries. Soft edits are expanded separately using an Expand arrow that appears on the soft edit entry itself. Click the Collapse button to collapse all selected parent entries, including soft edits.



Alternatively, use the Expand and Collapse arrows located to the left of parent entries to expand or collapse the child entries immediately below the parent entry.

When the list is collapsed, the Expand arrow points to the right; when it is expanded, the Collapse arrow points downwards.

To expand or collapse all child entries below the parent entry, **CTRL**-click the arrow.

Selecting Entries in a Clip Library

The following table describes methods for selecting and deselecting entries.

To:	Do this:
Select a single entry	<p>Click anywhere on the entry except on the Expand/Collapse arrow and Sort buttons of parent entries.</p> <p>In Titles and List View mode, a selected entry is light grey. On proxies, the name bar is yellow.</p> <p>When you click on a parent entry, the parent entry and all its child entries are selected.</p> <p>When a parent entry contains some selected child entries, but not all of them, its Expand or Collapse arrow is yellow. When all of a parent entry's child entries are selected, its Expand or Collapse arrow is green.</p>

To:	Do this:
Select all child entries of a parent entry	Click the parent entry.
Deselect one or more entries	Click them a second time.
Add more entries to a selection	In Titles and Proxies mode, click the entries to add. In List View, CTRL -click the entries to add.
Select a range of entries	In Titles and Proxies modes, click the first entry in the range, then press SHIFT and click the last entry in the range. In List View, you can also drag the cursor over the range of entries.
Select all entries in the clip library	Click the Select All button in the Library Management menu. Alternatively, click the clip library entry.
Deselect all entries	Click the Unselect All button in the Library Management menu. Alternatively, click the clip library entry.

Loading Entries from a Clip Library

Use the Load button to load entries from a clip library to the desktop. If a clip contains audio, you can load it with its audio tracks, or as video only. You can load clips from clip libraries in the current partition or in other local partitions (Discreet Filesystem required). If you have **wire**, you can load clips from remote partitions. For information on accessing clip libraries from other partitions, see “Using the Network Menu” on page 226.

To load entries from a clip library:

1. On the desktop, select the clip library containing the entries that you want to load from the Clip Library box.
2. Click Load in the Library menu.
3. Select a destination.

NOTE: A shorter method is to click the Load button of the reel you want to use as the destination.

The current clip library appears.

4. Select the entries that you want to load to the desktop (see “Selecting Entries in a Clip Library” on page 212 for details).

The type of entry you select determines where the entry is loaded.

If you select: Then:

Desktop	Each saved clip is loaded onto the same desktop reel that it was on when the desktop was saved. The destination reel is ignored. Any clips that are on the desktop when you load a saved desktop are retained.
Reel	All clips associated with the reel are loaded on the destination reel.
Clip	The clip is loaded on the destination reel.

HINT: You can load entire reels or selected source clips within soft edits. You may have to expand some parent entries to find the entries you are looking for. It is also possible to load desktops containing up to eight reels, even if the current desktop is set to fewer. The clips are loaded, but cannot be accessed unless the user increases the number of reels under desktop preferences.

5. If you are loading clips from a partition that has a different resolution from the current partition, the Fit Method button appears in the menu. Use this button to select a cropping or scaling method. For information on available options, see “Fit Method” on page 328.
6. If the clip contains audio tracks, and you only want to load the video, disable the Include Audio button.
7. Click Load.

The desktop reappears with the selected entries loaded.

Clear/Load

Use the Clear/Load command in the clip library to delete the destination before loading the selected entry. For example, if you are loading a reel entry, the destination reel is deleted before the reel is loaded.

To clear and load an entry, use the Clear/Load button rather than the Load button.



WARNING: Since the Clear/Load command may delete a desktop, reel, or clip, use this command with caution.

Transferring Clips Between Partitions

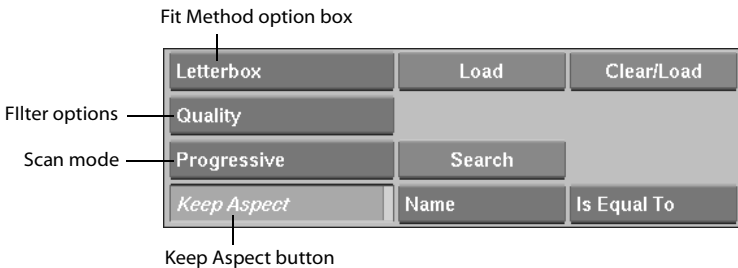
You can save or load individual clips, reels, or entire libraries to and from other partitions (local or remote) using **wire**. Make sure that the clip library you need access to is selected in the network library (see “Using the Network Menu” on page 226 for details), then select the library in the Clip Library box. Save or load clips in the same way as when using clip libraries in the current partition.

You can even load an entire clip library from another partition to the current partition. Select the remote clip library and deselect any individual clips you do not want to transfer. When you click Load, a new clip library is created on the current partition and the original clip order is

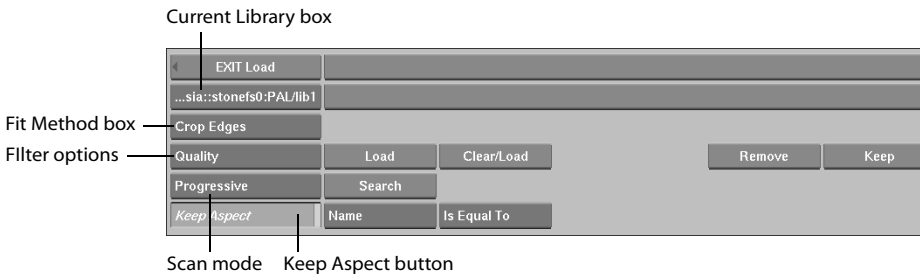
maintained. If a clip library with the same name already exists on the partition, the clips you are transferring get appended to the existing clip library. If a read-only clip library already exists, the new clip library will be copied to the partition and given the name “<library name> -1”.

Transferring Clips with Different Resolutions

You can save clips to, and load clips from, partitions that have a different resolution from the one you are working in. A number of options are available for optimum re-scaling when transferring the clips. When you select the Save option on the desktop to save a clip to a partition of a different resolution, the following additional options appear.



Similarly, when you select a clip to load from a partition that has a different resolution than the current library, the same options appear.



To transfer images between partitions with different resolutions:

1. Make sure the library to which you are saving, or from which you are loading, is the current library (appears in the Clip Library box).
2. If you are saving from the desktop, click the Save button. If you are loading from the clip library, select the clip you want to load.

3. Select the Fit Method.

Select:	To:
Centre/Crop	Transfer the image without resizing it. If the imported image is larger than the frame size, the image is centered and cropped. If the image is smaller than the frame size, the image is centered and surrounded by a black border.
Letterbox	Scale the longest edge to fit the frame size. Unused parts of the frame are filled with black.
Crop Edges	Scale the shortest edge to fit the frame size and crop the longest edge. The frame will be filled and part of the original image will be missing.
Fill	Stretch or squeeze the image to fill the new resolution. This option scales the X and Y dimensions of the image non-proportionality to make it fit in the current frame. This may change the aspect ratio if the source and destination aspect ratios are different and cause a distorted image).

4. If you selected Letterbox, Crop Edges or Fill, select the Filter Type.

Select:	To:
Coarse	Use a coarse filter for the transfer. This gives the poorest results, but provides a faster conversion.
Medium	Use a medium filter for the transfer. This gives adequate quality and average conversion times.
Quality	Use a Quality filter for the transfer. This gives very good quality images, but results in slower conversion times.
Bicubic	Use a Bicubic filter for the transfer. This results in the slowest conversion time. This option is most useful when importing an image with a lower resolution than that of the current partition. It reduces blur in resulting image.

5. If you selected Letterbox or Crop Edges, enable the Keep Aspect button. Enable this button to force **flame** to consider the aspect ratio of both the source and destination partitions during the transfer.

When Keep Aspect is enabled, images transferred between partitions with different pixel aspect ratios will not appear stretched or squeezed. For example, if your source material contains geometry, such as a circle, and it resides in an HD partition, when you transfer the image to an NTSC partition with the Keep Aspect button enabled, the circle will look correct. If the Keep Aspect button is disabled during such a transfer, the circle will look slightly stretched because the difference between the aspect ratio of the source and destination pixels is not taken into account. This button is enabled by default.

NOTE: When Keep Aspect is disabled, the functionality is the same as in versions prior to 3.6. The ability to enable or disable this feature has been included for backward-compatibility with your older projects. Typically, you should leave the Keep Aspect button enabled.

6. Select the Scan Mode of the material.

Select:	To:
Progressive	Transfer the material as Progressive.
Interlaced	Transfer the material as Interlaced.
Rev Dominance	Transfer the material as Interlaced and then change the field dominance.

NOTE: NTSC is the reverse dominance of both PAL and HDTV. When transferring between NTSC and PAL, or NTSC and HDTV, use the reverse dominance option. If you are transferring between PAL and HDTV, you do not need to reverse the dominance of the clips because field dominance for PAL and HDTV is the same.

7. If you are saving the clip, select the clip you want to save and name it. If you are loading the clip, click Load.

The clip is saved to the other partition or loaded to the local partition with the scaling options you have defined.

Aborting a wire transfer

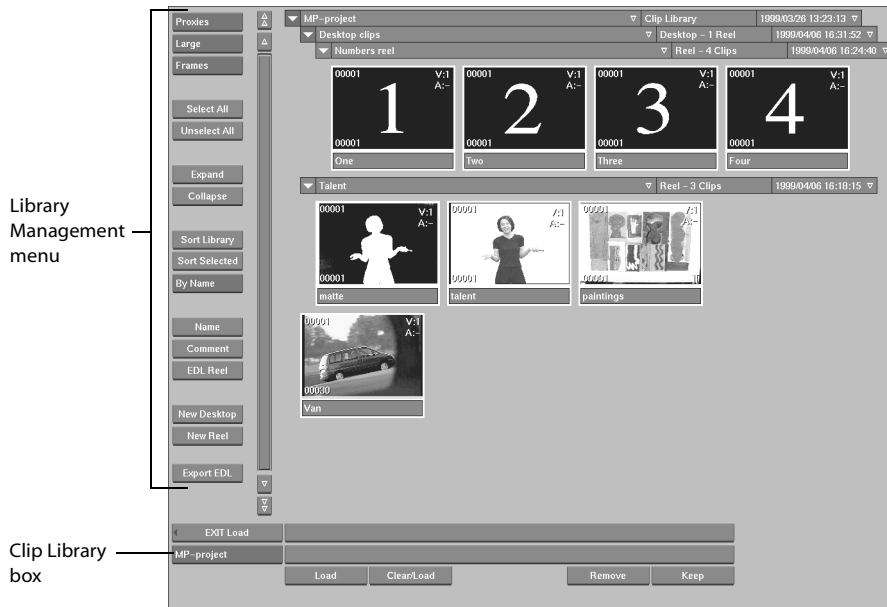
To abort a **wire** transfer of video or audio clips, click the mouse button or tap the pen on the tablet.

Organizing Clips

As clips accumulate in the clip library, you can organize them according to your needs. You can:

- Create new clip libraries and remove clip libraries that are no longer needed.
- Create new desktop or reel entries from a clip library and then move clips or other entries into the new entries.
- Copy entries.
- Remove clip and other entries that are no longer needed.
- Sort the entire clip library, or selected entries by name or by creation date.
- Name and rename entries.

To manage clips use the controls of the Library Management menu, the Clip Library box, as well as drag and drop operations.



Creating Clip Libraries

You can organize your projects by creating your own clip libraries and using them to store related desktops, reels, and clips together.

To create a clip library:

1. Select <new> in the Clip Library box.
The keyboard appears.
2. Type a name for the new clip library. Use only alphanumeric characters, single spaces, and underscores. Click Enter on the on-screen keyboard or press **ENTER**.
The new clip library is created.

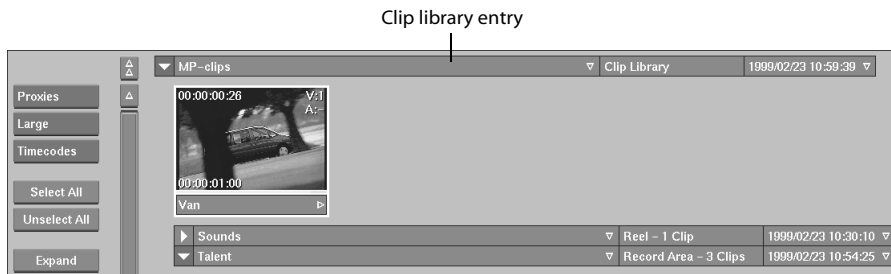
Removing Clip Libraries

You can remove a clip library using the Remove button in the clip library menu.

NOTE: You cannot remove a clip library if it is the only one in the partition. Also, you cannot remove remote clip libraries. For more information, see “Using the Network Menu” on page 226.

To remove a clip library:

1. Select the clip library to be removed from the clip library box.
2. Select the entry representing the clip library. The clip library entry is the top entry.



3. Click Remove, then click Confirm.

The clip library and all its entries are removed from the framestore.

NOTE: Images in a deleted clip library that are used by clips and reels in other clip libraries are not removed from the framestore. Only images that are not referenced elsewhere in the partition are removed. For more information on the framestore, see “Framestore Basics” on page 198.

Renaming Entries

You can rename an entry (a clip, a reel, a desktop, or the clip library itself) by selecting the entry you want to rename and clicking the Name button.

Renaming Clips in a Clip Library

If you rename a clip in a clip library and the same clip is on the desktop or elsewhere in the same clip library, the name is changed in both places. However, if you created a copy of the clip on the desktop, the name of the copy is not changed.

Renaming Clips on the Desktop

If you rename a clip on the desktop and the clip also exists in a clip library, the name of the clip in the clip library is *temporarily* changed to the new clip name. This helps you to identify clips on the desktop that are identical to clips in the clip library. Once you remove the clip from the desktop, the name of the identical clip in the clip library returns to the original name. This allows you to give different names to identical clips in different clip libraries.

Sorting Entries

You can sort selected entries in clip libraries using the sort buttons in the Library Management menu, or you can sort the clips within individual parent entries using the sort arrows.

Sort Library — Organizes entries according to the hierarchical structure of the library (see “Clip Library Structure” on page 203). Entries directly below the main clip library entry are sorted by individual clips, desktops, and then reels.

Sort Selected — Sorts only selected entries. You can sort by name or by date.

To sort using the sort arrows:

- Click the Name Sort arrow to alphabetically sort the immediate child entries of a parent entry by name (that is, the child entries directly below the parent entry). To sort all child entries below the parent entry, **CTRL**-click this button.



- Click the Date Sort arrow to sort the immediate child entries of a parent entry by creation date (that is, the child entries directly below the parent entry). To sort all child entries below the parent entry, **CTRL**-click this button.

NOTE: For information on sorting in List View, see “Sorting Clips in List View” on page 209.

Adding Reel and Desktop Entries

You can add reel or desktop entries to a clip library to use as containers as you organize clips.

To add a new reel or desktop entry to a clip library:

1. Click the New Reel or New Desktop button.
The on-screen keyboard appears.
2. Type a name for the new entry and click Enter.

Moving Entries

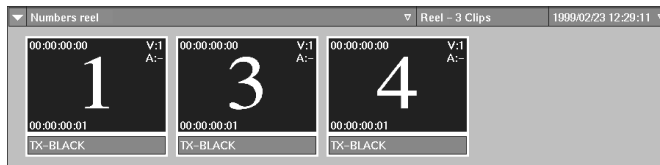
Move clip library entries using drag and drop operations in either Titles or Proxies mode. To drag and drop an entry, select the entry (by clicking it) and drag it to the new location. Release the stylus or mouse button to “drop” the entry.

You can move any entry to another location as long as the library’s hierarchal structure is maintained (see “Clip Library Structure” on page 203). For example, you can move a reel entry into a desktop entry, but not a desktop entry into a reel entry.

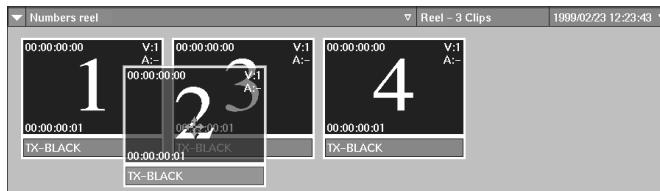
To move an entry:

1. Select the clip’s title bar (in Titles mode) or the name bar of the clip’s proxy (in Proxies mode) and drag the entry over the new position.
The cursor changes to a white or green four-headed arrow. When the cursor is green, it means the clip is in a location where you can successfully drop it. When the cursor is white, you cannot drop it at the current location. You need to move it closer to the destination.
2. Release the cursor to drop the entry. In Titles mode, the dropped clip is placed above the nearest clip. In Proxies mode, the dropped clip is placed to the left of the nearest clip.

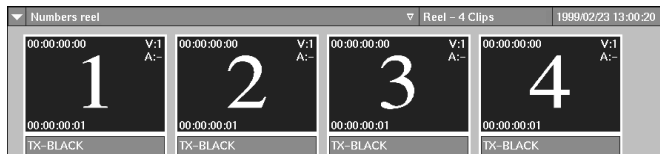
Dropping a clip in a new location



Before



Placing the clip



After: the clip is placed to the left of the nearest clip.

Moving Entries between Clip Libraries

To move an entry to another clip library, load it to the desktop and save it to the other clip library.

Copying Entries

Copy entries using drag and drop operations in Titles or Proxies mode. You can copy entries and drop the copies anywhere in the clip library.

To copy an entry:

1. Position the cursor over the entry you want to copy. When copying a clip displayed as a proxy, position the cursor over the proxy name bar.
2. Press and hold the **SPACEBAR**.
3. Drag the cursor away from the original entry.
The copy breaks away from the original and stays with the cursor. (You can release the Space Bar when you see the copy.)
4. Drag the copy to the desired location and release the mouse button. (If you have problems dropping the copy, see “Moving Entries” on page 220.)

Removing Entries from a Clip Library

Use the Remove and Keep buttons to remove selected desktops, reels, or clip entries from the current clip library:

- Use the Remove button to remove selected entries.
- Use the Keep button to keep selected entries and remove unselected entries.

Alternatively, you can use drag and drop operations to remove an entry.

NOTE: When you remove an entry from a clip library, copies of the entry that are used in other clips and reels are not removed from the framestore. Only entries that are not referenced elsewhere on the framestore are removed. For more information on the framestore, see “Framestore Basics” on page 198.

To remove selected entries:

1. From the Clip Library box, select the clip library from which you want to remove entries.
2. Select the entries that you want to remove. You can select parent entries, child entries, and combinations of parent and child entries. When an entry is selected, it is highlighted. For more information, see “Selecting Entries in a Clip Library” on page 212.
3. Click Remove, then click Confirm.

The selected entries are removed from the clip library.



WARNING: This procedure permanently removes the selected entries from the clip library.

NOTE: You cannot remove entries from a remote clip library. For more information on remote clip libraries, see “Using the Network Menu” on page 226.

To remove an entry using drag and drop operations:

1. Select an entry with the cursor and drag it to the bottom of the screen. When the green recycling cursor appears, drop the entry by releasing the stylus or mouse button.
2. Click Confirm.

NOTE: You can remove only one entry at a time using drag and drop operations.

To remove unselected entries:

1. Select all the entries that you want to keep.
2. Click Keep and then click Confirm.

Selected desktops, reels, and clip entries are kept and unselected entries are removed from the clip library.

Clip Compatibility Between Products

Effects and Editing products can share material in the same clip library. The Effects products consist of **inferno**®, **flame**®, **flint**®, and **effect**, while the Editing products consist of **fire**®, **smoke**® and Discreet Edit Utility.

This section describes some of the compatibility issues involved with sharing material between Effects and Editing products.

Running an Effects and Editing Product on the Same System

This section provides general guidelines for sharing material between Effects products and Editing products, and sharing material between older and newer versions.

When sharing clip libraries from different products:

- The following products share the same library format: **flame** 5.5 and 6.0, **inferno** 2.5 and 3.0, **effect** 5.5, **fire** 2.5, **smoke** 2.5, and Discreet Edit Utility 2.5. When using these versions of the software, you cannot share material from partitions created with older versions.
- The following products share the same library format: **flame** 6.1, **inferno** 3.1, **effect** 6.0, **flint** 6.1 (SE), **flint** 6.0 (Indigo2), **fire** 3.0, and **smoke** 3.0 and Discreet Edit Utility 3.0.

When using several products that have the same library format, you can share clip libraries transparently: you can load from and save to any clip library from any product.

When using several products with different library formats:

- You can access an older format library from an application using the latest library format, but you can only load clips from the library—you cannot save clips to the older library.
- Do not access a newer format library from an application that uses the older format. This can cause the newer format library to be converted to the older format. Once this happens, you can no longer save clips to the library from the newer format application.

See “Working with Clip Libraries from Previous Versions” on page 225 for information on transferring older clips to a clip library with a newer format.

Other points to keep in mind:

- Backward compatibility is available for archiving with all versions of Editing products greater than 2.0 and with Effects products versions greater than 2.0/4.0.
- Archives can be shared between products. An archive that is created in an Effects product is readable in an Editing product, and vice versa.
- Effects and Editing clips can be stored in the same clip library. You can identify the product where a clip originated by the shade of grey in the box at the left of the clip entry in Titles mode. In Editing products this box is dark grey, and in Effects products it is a lighter grey.
- Each time you load a clip from an Editing product into an Effects product, translation occurs and a new clip is generated. If you load same clip again, a new clip is generated with a new clip ID.

- The EditDesk and Desktop files are product-specific. Clips on the Effects product's Desktop will not appear on the Editing product's EditDesk, and vice versa.
- Both products can share the same framestore partition. However, the bit depth must be eight bits per component.
- Using **wire**, you can load clips from, and save clips to, other partitions, but you cannot delete clips on other partitions. This is also true when working within one product.
- SGI audio, which may be used in some Effects products, is completely compatible with discreet audio, which is used in Editing products. However, to be compatible with **flame**, the **wire** network must be configured properly for audio in the Editing product
- You can toggle between SGI audio and discreet audio in an Effects product. All clips on the desktop and clip libraries are compatible with the two types of audio.

Pointing to Clip Libraries

If you have older versions of Effects or Editing products, you may have clip libraries in a location other than `/usr/discreet/clip` and have a Clip keyword in your `init.cfg` configuration file that points to the location of these libraries. The clip library location for all products is now standardized at `/usr/discreet/clip` and the Clip keyword is no longer used.

To access the older clip libraries, copy or move all the directories into the default location: `/usr/discreet/clip`.

Loading Editing Clips into Effects Products

Keep the following points in mind when loading Editing clips into an Effects product:

- The primary video track, which is the track set to output channel V1, is the only one imported when you load an Editing clip with multiple-video tracks into an Effects product. Be sure to verify your primary video track before you transfer the clip, since the other video tracks will be ignored in translation.
- Wipes, strobes, soft commits and custom transitions are hard-committed. Dissolves and timewarps are kept soft.
- Only tracks A1 and A2 are kept when you load an Editing clip with multiple-video tracks into an Effects product. All other audio tracks are discarded. If the audio tracks contain soft commits, they are hard-committed during import.
- If you are transferring an audio-only clip, a black source is automatically added to the clip for the duration of the audio. Audio-only clips appear as black proxies in the clip library.
- If the audio is longer than the video in the Editing clip, black sources are automatically created to fill the video gaps for the duration of the audio.
- In and Out marks are removed from Editing clips when imported into Effects products.

Loading Effects Clips into Editing Products

Effects clips are 100% compatible with Editing products; no translation or rendering occurs. Keep in mind that in Effects products, audio that extends beyond the video elements is not considered part of the clip. While in Editing products, all audio is considered part of the clip, regardless of the duration of the video element.

Working with Clip Libraries from Previous Versions

In **flame** 7.0, you cannot save clips to clip libraries created in previous versions because the library format has changed. If you upgraded to version 7.0 in the middle of a project and need to work with older clip libraries, use one of the following procedures:

- Archive the material from the older version, and then restore it into a clip library created in **flame** 7.0.
- Use **wire** to load the material into a new partition created in **flame** 7.0. The steps to do this are given next.

To load clips from a previous version using wire:

1. In **flame** 7.0, create a new partition.
2. Create new clip libraries in the new partition.
3. Use **wire** to access the clip libraries in the old partition.
4. Load all clips from the old clip libraries and save them to the new clip libraries.
5. Exit **flame**.
6. In a UNIX shell, delete the old library files located in */usr/discreet/clip/stonefs/old partition name*. For each old library you are removing, delete the following files:
 - *library_name.library*
 - *library_name.library-backup*
 - *library_name.library_prev_backup*
 - *library_name.ui*
7. Re-launch **flame** 7.0.
8. Delete all frames in the Lost and Found library.

Using the Network Menu

Use Network menu to specify clip libraries residing on other partitions that you want to access. You can access clip libraries on any local partition, and if you have **wire**, you can access partitions on remote systems.

NOTE: Remote systems must appear in the `sw_host_map` file in order to access them. For more information, refer to the *flame Installation Guide*.

The Network Library

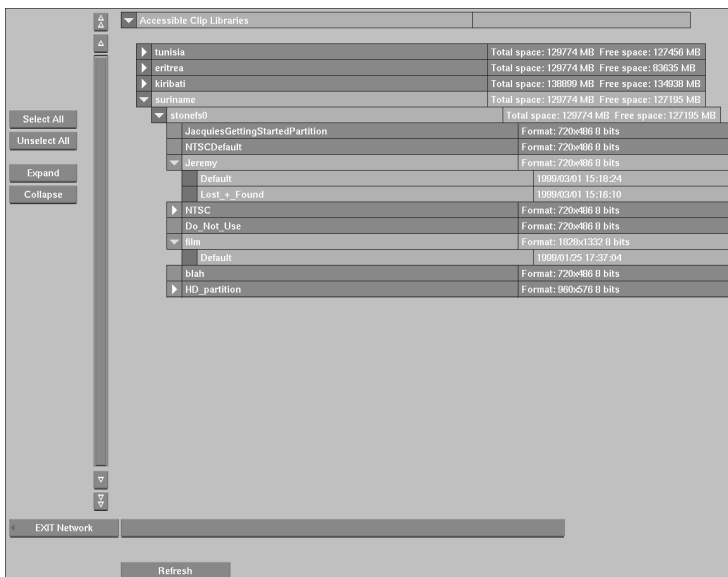
The network library shows all accessible clip libraries on the local and on remote systems. Use this library to specify the clip libraries to which you want to have access (that is, the clip libraries you want to appear in the Clip Library box).

To access the network library, click the Network button in the Library menu.



Network button

The network library appears.



To view the entire contents of the network library:

1. Click Refresh
2. Select and expand all entries by clicking Select All and then the Expand button.
3. Click Unselect All.

Remote systems only appear in the network library if they are defined in the *sw_host_map* file. For more information, refer to the *flame Installation Guide*.

Network Library Structure

The network library has a similar structure to a clip library. Like a clip library, the hierarchical relationship among groups of available remote systems, partitions, and clip libraries is represented using indentation. The child entries are indented and are shown below their parent entries.

Entries

The network library can contain the following entries.

Entry:	Contains:
Host (local or remote)	A child entry for each framestore on the local or remote system
Framestore	A child entry for each partition on the framestore.
Partition	A child entry for each clip library in the partition.
Clip Library	No child entries.

The colour of the box at the left of the entry indicates the type of entry.

The colour:	Represents:
Blue	Host (local or remote).
Red	Framestore volumes.
Green	Partitions.
Dark grey	Clip libraries.

The Network Library Menu

Use the network library menu to select entries.

Select All and Unselect All — Click the Select All button to select all the entries in the network library. Click the Unselect All button to unselect all entries.

Expand and Collapse — Click the Expand button to expand selected entries. Click the Collapse button to collapse selected entries.

HINT: You can also click the Expand and Collapse arrows at the left of a parent entry to expand and collapse its child entries.

Refresh — Use the Refresh button to update the list of local partitions and accessible remote systems.

Accessing Clip Libraries on Other Partitions

To make a clip library on another partition accessible, select it in the Network Library.

To access a clip library on a another partition:

1. Open the Network menu.
2. Click Refresh.
flame searches for all accessible partitions as specified in the *sw_host_map* file.
3. If needed, expand entries to see the available partitions or clip libraries. To view all entries, click Select All, then Expand. Next, click Unselect All and make individual selections.
All accessible remote systems appear.
4. Select the clip libraries you want to access. The procedure for selecting entries is the same as that used in clip libraries. For more information, see “Selecting Entries in a Clip Library” on page 212.

HINT: Select a host entry to access all libraries on the remote system; select a partition entry to access all libraries on that partition.

5. Exit the Network menu.
All the clip libraries that you selected in the network library are listed in the Clip Library box.

Hot Keys

Use the following hot keys in clip libraries.

Press:	To:
HOME	Go to the top of the library.
END	Go to the bottom of the library.
PAGE UP	Go up one page.
PAGE DOWN	Go down one page.
↑	Scroll up by one line.
↓	Scroll down by one line.
N	Name or rename the selected entry.
T	Change view to Titles mode.

Press:	To:
P	Change view to Proxies mode.
S	Select all entries.
U	Unselect all entries.
E	Expand selected entries.
C	Collapse selected entries.
O	Sort library.
SPACEBAR	Copy an entry.
R	Remove (delete) selected entries.
L	Load selected entries to the desktop.
A	Remove any clips from the selected reel and then load selected entries to the reel.
F	Refresh the library cache.
X	Exit the clip library.

[illegible]

13

Clip Input and Output

*Nabbing your clips...
...and letting them go*

*To work with video and audio clips in **flame**, you must bring them in from an outside source. Once you add your edits and effects, you need a way to get them back into a usable format. This chapter guides you through these procedures.*

Summary

In this chapter, you learn about:

- “Inputting Clips from a VTR” on page 232
- “Outputting Clips to a VTR” on page 245
- “Inputting Clips from a VCR, Camera, or Camcorder” on page 250
- “Outputting Clips to a VCR” on page 251
- “DDR Input and Output” on page 252
- “Clip I/O with the Sony HDCAM Codec” on page 256
- “Data Tape Input and Output” on page 263
- “Troubleshooting” on page 265

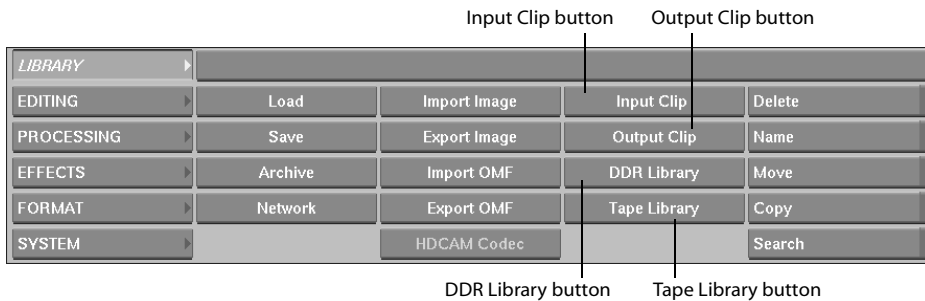
About Clip Input and Output

Several methods can be used to transfer clips to and from various devices. These devices include:

- VTR
- DDR
- VCR, camera or camcorder
- DAT and DST tape drives

Use the Input Clip and Output Clip menus to perform real-time input and output of clips at 720 x 486 resolution to and from a VTR, VCR, camera or camcorder.

Use the DDR Library menu for transfer to and from a DDR. The Tape Library allows transfer to and from DAT and DST drives. These menus are all accessed from the Library menu.



Inputting Clips from a VTR

Use the Input Clip menu to input source material from a VTR or a DDR in VTR emulation mode. You can input video clips with or without audio.

NOTE: Only the ProntoVision HD DDR is currently supported for **flame**.

The VTR you are using must be defined in the VTR KEYWORD section of the *init.cfg* configuration file. See the **flame** *Installation Guide* for details.

To input source material, you can:

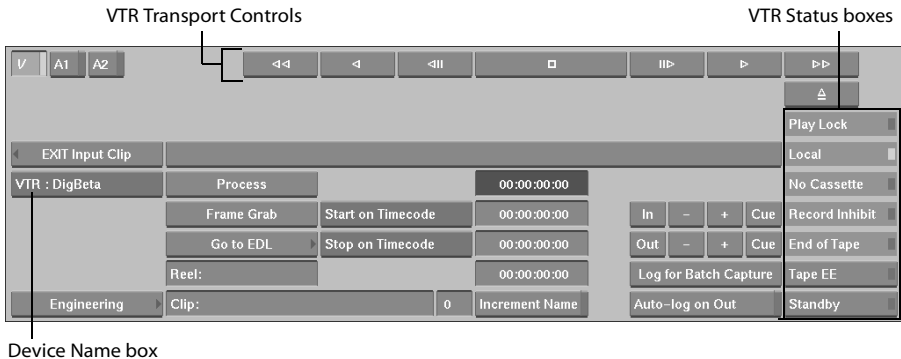
- Input clips individually using the Process button.
- Use clip logging options to generate an EDL for multiple clips and then capture them from the EDL module.
- Input individual frames using Frame Grab.

NOTE: You can also input source material using an existing EDL, by accessing the EDL module with the EDL button in the Editing menu. For more information, see Chapter 14, “EDLs.”

To access the Input Clip menu:

1. Open the Library menu.
2. Click the Input Clip button.
3. Select a destination.

The Input Clip menu appears.



Device Name box

4. Select a VTR from the Device Name box.

The video on the selected device appears in the image window.

NOTE: The device you are using must be enabled in the VTR KEYWORD section of the *init.cfg* configuration file; otherwise, it will not appear in the Device Name box. See the **flame** *Installation Guide*.

5. Make sure that the VTR is in remote mode; otherwise, **flame** cannot transfer clips. You can only switch the VTR from local to remote mode on the VTR panel itself. If the Local box in the VTR Status boxes is enabled, the VTR is not in remote mode. For more information, see “VTR Status Boxes” on page 236.
6. Use the VTR Transport controls to play forwards, play backwards, jog, rewind, and fast forward. For more information, see “Playing the VTR” on page 237.

Capturing Clips Individually

You can capture a single clip from a VTR.

Summary of steps to capture a single clip from a VTR:

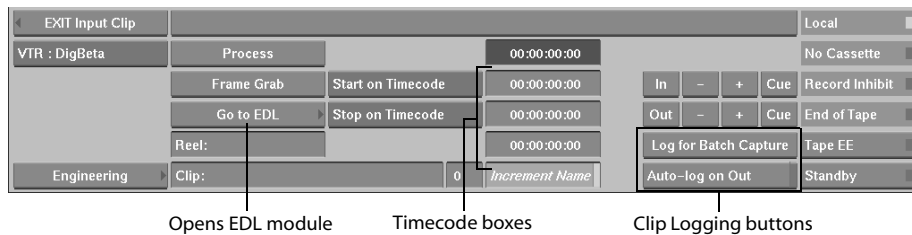
1. From the Input Clip menu, select the correct VTR from the Device Name box. See “To access the Input Clip menu:” on page 232.
2. Enable the channels you want to capture. See “Selecting Channels for Input” on page 240.
3. Click the Engineering button and set various options in the Engineering menu. See “Setting Engineering Menu Options” on page 240.
4. Exit the Engineering menu.
5. Set the in- and out-timecodes. See “Setting the In- and Out-Timecodes for Input” on page 238.

- Click Process to begin transferring the segment specified by the in- and out-timecodes from the VTR.
- After the segment is processed, you can input another clip. When you have finished inputting clips, click the EXIT Input Clip button to return to the desktop reels.

NOTE: When audio channels are enabled, **flame** loads the audio channels and the video channel simultaneously. If a frame is dropped during capture, **flame** continues capturing the audio. Once this is complete, **flame** cues the tape at the drop point and resumes capturing the video alone.

Capturing Multiple Clips

You can capture more than one event from a VTR. When you log a clip for batch capture, it is automatically added to an EDL. You can then go directly to the EDL module and capture all the clips you logged. You can log clips dynamically using the Auto-log on Out button, or you can log clips individually using the Log for Batch Capture button.



To auto-log clips:

- From the Input Clip menu, select the correct VTR from the Device Name box. See “To access the Input Clip menu:” on page 232.
- Enable the Auto-log on Out button.
- Enable the channels you want to capture. See “Selecting Channels for Input” on page 240.
- Enter the in- and out-timecodes of the clip. See “Setting the In- and Out-Timecodes for Input” on page 238.
The clip coordinates are automatically logged in an EDL when you click the Out button.
- Repeat steps 3 to 4 for each clip to capture.
- To capture the clips, click the Go to EDL button.
The EDL module appears, with an EDL containing all the logged clip coordinates.
- Capture the clips. See “Auto-Capturing an EDL” on page 288.

To log clips individually for batch capture:

1. From the Input Clip menu, select the correct VTR from the Device Name box. See “To access the Input Clip menu:” on page 232.
2. Disable the Auto-log on Out button.
3. Enable the channels you want to capture. See “Selecting Channels for Input” on page 240.
4. Enter the in- and out-timecodes of the clip. See “Setting the In- and Out-Timecodes for Input” on page 238.
5. Click the Log for Batch Capture button.
The clip coordinates are logged in an EDL.
6. Repeat steps 3 to 5 for each clip to capture.
7. To capture the clips, click the Go to EDL button.
The EDL module appears, with an EDL containing all the logged clip coordinates.
8. Capture the clips. See “Auto-Capturing an EDL” on page 287.

Capturing Single Frames

You capture a single frame from a tape on the VTR. A clip composed of the frame displayed in the image window is obtained.

To capture a single frame:

1. Open the Engineering menu and set the following parameters:
 - Input Connection
 - Input Sync
 - Colourspace
 - Precision

For information on these options, see “Setting Engineering Menu Options” on page 240.

2. Use the VTR Transport controls to go to the frame you want to capture—it should be displayed in the image window. See “Playing the VTR” on page 237 for details.
3. Exit the Engineering menu.
4. Click the Frame Grab button.

When you return to the desktop reel, it will contain a clip composed of the selected frame.

If the image acquired using Frame Grab contains bad anti-aliasing, the VTR is not capable of displaying both fields of the image simultaneously when in Pause mode. Therefore, the two fields are not successfully transferred. Some VTRs have the option of repeating the lines from F1 or F2 or displaying both fields of the image when in Pause/Shuttle mode. If the VTR can be set to display both fields, verify that it is set correctly. Otherwise, use Start on Timecode and

Stop After Frames (set to one frame). See “Setting the In- and Out-Timecodes for Input” on page 238 for details.

Using the Input Clip Menu Options

This section shows you how to:

- Verify the status of the VTR
- Play and jog the VTR
- Name the reel and clip
- Set in- and out-timecodes
- Select channels for input

NOTE: In and Out 2:3 Sequence Frame icons appear to the right of the timecode fields when 2:3 Removal is enabled in the Engineering menu. See “Clip Input and Output for 24p Projects” on page 306 for details.

VTR Status Boxes

The status boxes at the right side of the Input Clip menu show the current status of the VTR.



The boxes are enabled (blue light lit) under certain conditions. Depending on the model of the VTR, you may be able to change the Tape EE and Standby boxes.

Status box:	Indicates:
Play Lock	All the motors in the VTR are locked at play speed when playing or recording. Play Lock should light up about 1 second after starting to play or record. When recording, Play Lock must be lit before the preroll has finished, to ensure that the correct speed has been reached.
Local	Whether the VTR is in local or remote control mode. Enabled when the VTR is in local control. When in local control, the VTR cannot be controlled from within flame . You can only switch the VTR between local and remote control on the VTR itself.

Status box:	Indicates:
No Cassette	Whether the VTR contains a cassette. Enabled when the VTR does not contain a cassette.
Record Inhibit	The Record Inhibit status. Enabled when record is inhibited by either a control on the VTR or the record tabs on the VTR cassette.
End of Tape	The VTR is at the end of the tape.
Tape EE	The VTR is bypassed. In this case, the output of the VTR comes directly from its input.
Standby	The Standby mode. When Standby is on, the play heads on the VTR are engaged. When Standby is off, the play heads are disengaged. Standby is lit when you process a clip.

Playing the VTR

The current frame on the cassette in the VTR appears in the image window and the Current Timecode box displays the timecode of the current frame. Use the VTR Transport controls beneath the image window to play the VTR.

Click:	To:	Hot Key:
◀◀	Rewind the tape.	HOME
◀	Play the tape backward.	-
◀	Move backward one frame. Hold down to slowly jog backward.	← ARROW KEY (↓ FOR -10 FRAMES)
□	Stop the tape.	SPACE BAR
▶	Move forward one frame. Hold down to slowly jog forward.	→ ARROW KEY (↑ FOR +10 FRAMES)
▶	Play the tape forward.	ENTER
▶▶	Fast-forward the tape.	END
△	Eject the tape.	none

You can also shuttle the VTR by dragging the cursor in the image window. Drag the cursor to the right to shuttle forward and to the left to shuttle backward. The farther you move the cursor to the right or left of the screen, the faster the speed.

Naming the Reel and Clip

Use the following fields to name the reel and clip.

Frame Grab	Start on Timecode	00:00:00:00	In	-	+	Cue
Go to EDL	Stop on Timecode	00:00:00:00	Out	-	+	Cue
Reel:		00:00:00:00	Log for Batch Capture			
Clip:	0	Increment Name	Auto-log on Out			

Clip field Reel field Increment field Increment Name button

Select:

To:

- Reel field Enter a name for the reel on which you want to place the clip after it is captured. The name can be up to seven characters long.
- Clip field Enter a name for the clip to be transferred from the VTR.
- Increment field The value in this field is incremented each time you grab a frame or transfer a clip from the VTR.
- Increment Name button Add the value in the Increment field to the name of a captured clip.

Setting the In- and Out-Timecodes for Input

There are four timecode fields in the Input Clip menu:

	Current Timecode field	In-Timecode field	
	00:00:00:00		
Start on Timecode	00:00:00:00	In	+ Cue
Stop on Timecode	00:00:00:00	Out	+ Cue
	00:00:00:00	Log for Batch Capture	
	Duration Timecode field	Out-Timecode field	

Cue In button Cue Out button

- Current Timecode field: The timecode of the current frame of the tape. This is updated as the tape is played.
- In-Timecode field: The timecode of the first frame in the segment to be transferred.
- Out-Timecode field: The timecode of the frame after the last frame in the segment to be transferred.
- Duration Timecode field: The duration of the segment to be transferred.

To set the in- and out-timecodes for input:

1. Select the way you want the transfer to start and stop using the Start and Stop Transfer Method boxes.

Start Transfer Method box

	00:00:00:00	
Start on Timecode	00:00:00:00	In - + Cue
Stop on Timecode	00:00:00:00	Out - + Cue
	00:00:00:00	Log for Batch Capture

Stop Transfer Method box

Select:	To:
Start on Timecode	Start transferring the clip when the in-timecode is reached.
Start on Pen	Start transferring the clip when the pen is pressed.
Select:	To:
Stop on Timecode	Stop transferring the clip when the out-timecode is reached.
Stop on Pen	Stop transferring the clip when the pen is pressed.
Stop After Frames	Stop transferring the clip after a specified number of frames. When this option is selected, an additional box is shown where you enter the number of frames. The Out and Duration timecodes are updated to reflect the number of frames in this additional box.

- If you selected Start on Timecode and/or Stop on Timecode, complete steps 2-5 to set the in- and out-timecodes.
 - If you selected Start on Pen, Stop on Pen, or Stop on Frames, you do not need to set the in- and out-timecodes.
2. Play the tape to the first frame of the segment to be transferred. For more information, see “Playing the VTR” on page 237.
The timecode for the current frame appears in the Current Timecode box.
 3. Click the In button beside the In-Timecode box.
The In-Timecode field is set to the current timecode.
 4. Play the tape to one frame after the last frame of the segment to be transferred.
The timecode for the current frame appears in the Current Timecode field.
 5. Click the Out button beside the Out-Timecode field.

The Out-timecode field is set to the current timecode and the duration is updated as follows: $\text{Duration} = \text{Out-timecode} - \text{In-timecode}$. If you change the duration, the Out-timecode is automatically updated.

You can adjust the In- and Out-timecodes by:

- Using the + and - buttons to advance or back up the timecode by one frame
- Clicking on the timecode field and entering it manually
- Dragging in the timecode field

To view the In- or Out-timecode frames in the image window, click the Cue In or Cue Out buttons.

Selecting Channels for Input

You can input any combination of the channels in a shot—video alone, video and audio, or audio alone.

Use the Channel Selection boxes to indicate which channels you want to input.

Channel Selection boxes



Click a box to select or deselect it.

Setting Engineering Menu Options

Use the Engineering menu to select options for transferring clips between **flame** and the VTR.

Most of the default settings for the Engineering options are specific to the VTR selected in the Device Name box, as specified for each VTR in the VTR KEYWORD section of the *init.cfg* configuration file. Making changes in the Engineering menu overrides the *init.cfg* settings for the current **flame** session only: the next time you start **flame**, the *init.cfg* settings take effect again. You can change the default settings by modifying the *init.cfg* file. For more information, see the **flame** *Installation Guide*.

For information on Engineering menu options for 24p mastering — the 2:3 Removal and 2:3 Insertion buttons — see “24p Mastering Engineering Menu Options” on page 307.

To display this menu, click the Engineering button in the Input Clip menu, Output Clip menu, or Archive menu.

Timecode Source box		Preroll and Postroll boxes		
EXIT Engineering				
VTR : DigBeta	Timecode VITC & LTC	Serial 1	10 bit precision	Audio Preferences
00:00:00:00	PreRoll 150	House	Output Digital Filter	
F1 Dominance	PostRoll 30	Normal RGB	Stop VTR On EXIT	
	Video Input Delay -1	Serial 1	Cueup VTR	Audio Input Delay 0.00
	Video Output Delay -4	House	Error Retry 10	Audio Output Delay 0.0

Field Dominance VTR Play and Record Delay boxes

Field Dominance — If needed, change the field dominance to match that of the material on the tape. You can also set the field dominance in the System Preferences menu.

NOTE: If the HIRES KEYWORD in the project configuration file is set to 50Hz, 60Hz or 72Hz, the field dominance on the broadcast monitor will be random during playback, resulting in jitter. To solve this problem, set the HIRES KEYWORD to 30Hzf for NTSC or 25Hzf for PAL.

Timecode Source — Specify the type of timecode to obtain from the VTR. **flame** supports the following types of timecode:

Select:	To:
VITC	Use Vertical Interval Timecode.
LTC	Use Longitudinal Timecode.
VITC & LTC	Allow the VTR to choose the more appropriate timecode. This is the recommended option for inputting and outputting clips from and to a VTR.

Preroll and Postroll — Use the Preroll field to enter the number of frames to which you want the tape to be prerolled. The preroll ensures that the tape is running at the proper speed before transferring clips to and from the VTR.

Use the Postroll field to enter the number of frames you want the tape to play after the last frame is reached. This ensures that the tape is running at the proper speed at the end of the clip.

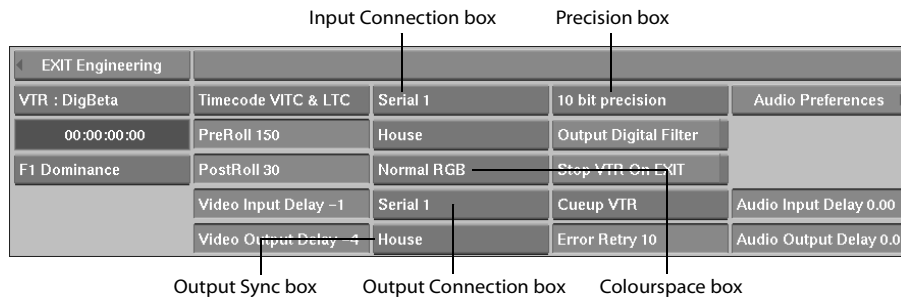
NOTE: If the preroll or postroll is too low, frames could be dropped from the transfer. If the preroll is too low, the system and VTR may not be ready in time for the in-timecode. Allow sufficient time to obtain the Play Lock.

Input Delay and Output Delay — Use Input Delay to specify the number of frames the clip should be shifted ahead as it is transferred from the VTR to **flame**. A negative number of frames shifts the clip back. This option compensates for frame delays that may exist while transferring clips from the input device to **flame**.

Use Output Delay to specify the number of frames the clip should be shifted ahead as it is transferred from **flame** to the VTR. A negative number of frames shifts the clip back. This option compensates for frame delays that may exist when transferring clips from **flame** to the output device.

NOTE: You can test and set the play delay automatically by turning debug mode ON (**SHIFT-D-INS**) and clicking the Auto Test button. This writes to the tape for 1 second. You must exit debug mode once this procedure is complete (**SHIFT-D-INS**). The Auto Test button is only visible in debug mode.

The default Play and Record delays are set according to the type of video hardware and VTR used.



Input and Output Connection — Specify the input and output connection you are using. Select from the following options.

Select:	To:
Serial 1 or Serial 2	Use the Serial 1 or 2 ports for input from or output to any 4:2:2 (Y'CrCb) capable device, such as a VTR.
Serial 1	Use the Serial 1 port for input from or output to any 4:2:2 (Y'CrCb) capable device, such as a VTR.
Serial 2 (O2s with Analog AV1 video card only)	Use the Serial 2 port for input from or output to any 4:2:2 (Y'CrCb) capable device, such as a VTR.
MGV2 (O2s with Digital AV2 card only)	Use the MGV2 port for input from or output to any 4:2:2 (Y'CrCb) capable device, such as a VTR.
Serial Dual (444)	Use both Serial 1 and Serial 2 for input from or output to any 4:4:4 (RGB) capable device, such as a Telecine.
Serial 1 and 2	Output to both Serial 1 and 2 the same images simultaneously to any 4:2:2 (Y'CrCb) capable devices, such as a VTR and a broadcast monitor. (Output only)
Composite (O2s with Analog AV1 video card only)	Use the Composite port for input from or output to any composite-capable device, such as a VCR.

Select:	To:
SVHS (O2s with Analog AV1 video card only)	Use the SVHS port for input from or output to any Svideo-capable device, such as a VCR or camcorder.



Input Sync — When you input clips, a reference signal is needed to synchronize the SGI and the VTR. **flame** uses the signal associated with the incoming images as the reference source (regardless of the port being used). For best results, set the Input Sync option to House.

Output Sync — When outputting clips to a VTR, you need to use a reference signal to synchronize the SGI and the VTR. The signal may originate from several different sources. Specify the source you are using in this box according to the following table.

Option	Description
House	Use House when you are using a reference signal generator to synchronize the signal. This is generally used when there are several devices. The signal generator is connected to the Genlock port on the video board.
Digital	Uses the video input (601 signal) as the reference source. The signal originates from the device connected to the video input (generally the VTR). WARNING: Do not use Digital when outputting directly to a VTR that is also referenced to its own input since in this case, none of the devices generate a stable reference.
Standalone	The reference signal is generated by the SGI. This is appropriate for standalone systems where a signal generator is not available.

Colourspace — Use this box to control colour space conversion for both input and output.

Use the following table to help you select a conversion type.

Select:	To:
Normal RGB	Convert clips being input from Y'CrCb to RGB colour space, and convert clips being output from RGB to Y'CrCb colour space. All colours falling within the Super White range (235-255) in Y'CrCb are set to 255 in RGB. Similarly, all colours in the Super Black range (0-16) are set to 0 in RGB. Use this option for normal input and output of clips from and to a VTR.
RP175 RGB	Convert clips being input from Y'CrCb to RGB colour space, and convert clips being output from RGB to Y'CrCb colour space. No range expansion of 16-235 in Y'CrCb to 0-255 in RGB is made, resulting in a grey overcast. Use this option to input matte information stored in the Super Black area. Once you input the clip, you can use the histogram in the Keyer to restore the matte (see "Adjusting the Luminance of the Key" on page 641).

Select: To:

Conversion OFF No conversion or luminance expansion is made. Use if you are using a Telecine to input a 4:4:4 RGB signal directly to the Serial Dual Video Input of the Octane. (In this case, choose Serial Dual for the Input Connection.)

Precision box — When you transfer clips, the images go through $Y'CrCb$ -RGB and RGB- $Y'CrCb$ conversion. To minimize image degradation from multiple colour conversions, make sure you set the correct option in the Precision box. If you have a 10-bit VTR such as a digital betacam or Panasonic D5, select 10-bit precision. If you have an 8-bit VTR such as the Sony DVR 2100, select 8-bit precision.

Output Digital Filter — This box gives you the option of applying a 4:4:4 to 4:2:2 decimation filter to the material you are outputting. This filter is designed to minimize signal degradation in the RGB- $Y'CrCb$ conversion that is performed during output.

This filter provides more effective results on clips containing a lot of contrasting colour, as it reduces “ringing”.

Stop VTR on Exit Option — When this option is enabled, the VTR is stopped when you exit to desktop reels or change VTR through Device Name box. When this option is disabled, the VTR continues the last transport operation (for example, playing) when you exit to the desktop reels.

Stop VTR on Exit option			Output Digital Filter	
EXIT Engineering				
VTR : DigBeta	Timecode VTC & LTC	Serial 1	10 bit precision	Audio Preferences
00:00:00:00	PreRoll 150	House	Output Digital Filter	
F1 Dominance	PostRoll 30	Normal RGB	Stop VTR On EXIT	
	Video Input Delay -1	Serial 1	Cueup VTR	Audio Input Delay 0.00
	Video Output Delay -4	House	Error Retry 10	Audio Output Delay 0.0

Cueup box
Error Retry box
Audio Delay fields

Cueup — Use this box to specify how the VTR cues up to each segment. Cueup Fast Forward is usually faster than Cueup VTR. If you experience problems when using Cueup Fast Forward, try using Cueup VTR.

NOTE: If you are using a DDR in VTR emulation mode, always use Cueup VTR (using this will be much faster for cueing).

Error Retry — Instead of aborting an input or output when an error such as lost playback speed occurs, **flame** retries the edit. Use the Error Retry field to specify the number of times **flame** will retry the input or output before aborting. See “I keep getting Lost Playback Speed errors. How do I fix it?” on page 267.

Audio Preferences — This button opens the Audio Preferences menu. When inputting or outputting clips, verify the following audio preferences:

- For clip input: audio sync lock, audio source, sample rate, and meters.
- For clip output: audio sync lock, sample rate, and meters.

For complete details on setting these preferences, see “Audio Preferences” on page 125.

Audio Input Delay and Audio Output Delay — Use Audio Input Delay to specify the number of frames the audio should be shifted ahead as it is transferred from the VTR to **flame**. A negative number of frames shifts the audio back. This option compensates for hardware delays that may occur while transferring clips from the input device to **flame**.

Use Audio Output Delay to specify the number of frames the audio should be shifted ahead as it is transferred from **flame** to the VTR. A negative number of frames shifts the clip back. This option compensates for hardware delays that may occur when transferring clips from **flame** to the output device.

The default Audio Play and Record delays are set according to the type of hardware used. Values of 0.0 should yield synchronized audio and video.

Outputting Clips to a VTR

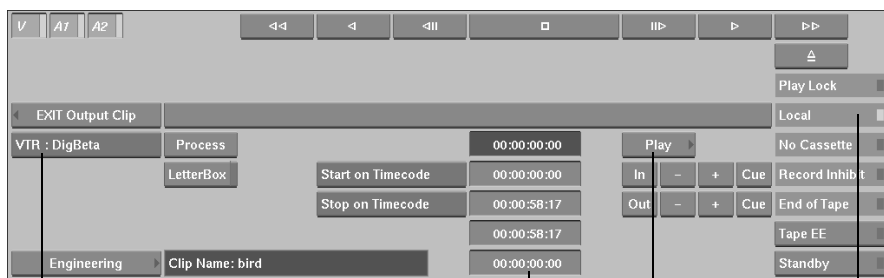
Use the Output Clip menu to output clips, with or without audio, to a VTR. Clips are output one at a time.

The VTR you are using must be defined in the VTR KEYWORD section of the *init.cfg* configuration file. See the *flame Installation Guide* for details.

Summary of steps to output a clip to the VTR:

1. Click the Output Clip button in the Library menu.
2. Select the clip.

The Output Clip menu appears.



Device Name box

Start Offset

Play button

Local box

3. Select a VTR from the Device Name box.

The clip on the selected device appears in the image window.

NOTE: If the correct device is not in the list, it is not enabled in the *init.cfg* configuration file. See the *flame Installation Guide*.

4. If you want to preview the clip you are outputting, click the Play button to access the Play module.
5. If necessary, set an offset from the beginning of the clip using the Start Offset timecode.
6. Enable the tracks you want to output. See “Selecting Tracks for Output” on page 249.
7. Click the Engineering button to set up other parameters for the transfer, such as timecode source, preroll and postroll. See “Setting Engineering Menu Options” on page 240.
8. Set the in- and out-timecode on the tape. (The in- and out-timecodes default to the clip timecodes.) See “Setting the In- and Out-Timecodes for Output” on page 247.
9. Make sure that the VTR is in remote mode; otherwise, **flame** cannot control the VTR. You can only switch the VTR from local to remote mode on the VTR panel itself. If the Local box is enabled, the VTR is not in remote mode. See “VTR Status Boxes” on page 236.
10. Click Process to begin transferring the source clip to the VTR.

NOTE: When audio tracks are enabled, **flame** transfers the audio and video simultaneously. If a frame drop occurs, the audio track transfer is completed first. Afterwards, the tape is cued a second time to transfer the video from the drop point.

11. After the transfer is complete, verify that it was successful by playing the transferred clip: cue to the in-timecode and click the Play button in the VTR Transport controls (▶).
12. When you are done, click EXIT Output Clip to return to the desktop reels.



Using the Output Clip Menu Options

This section shows you how to:

- Use a letterbox format
- Set the in- and out-timecodes
- Select tracks for output

NOTE: The In and Out 2:3 Sequence Frame icons appear to the right of the timecode fields when 2:3 Insertion is enabled in the Engineering menu. See “Clip Input and Output for 24p Projects” on page 306 for details,

Outputting Clips with Letterbox Formatting

The Letterbox option is used to change the aspect ratio of video clips. This is achieved by masking the top and bottom of the image, not by applying an anamorphic effect to the image.

Enable the Letterbox button to turn letterbox formatting on or off. When Letterbox is enabled, the letterbox formatting controls are displayed, allowing you to choose which format to use, and how it is displayed.

To set a letterbox:

1. In the Output Clip menu, enable the Letterbox button.

The letterbox formatting controls appear.

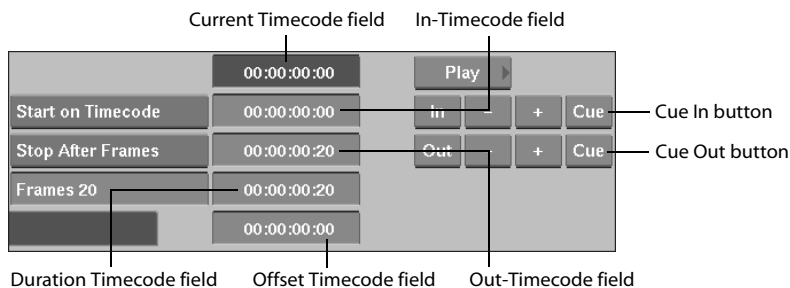
2. Change the letterbox format options.

Use:	To:
Aspect	Enter a custom aspect ratio.
Offset	Offset the position of the letterbox masks by entering positive or negative values in the Offset field.

NOTE: By default, the top and bottom letterbox masks are identical, and the image is centred in the viewer.

Setting the In- and Out-Timecodes for Output

There are five timecode fields in the Output Clip menu:



- **Current Timecode field:** The timecode of the current frame of the tape. This is updated as the tape is played.
- **In-Timecode field:** The timecode for the first frame in the segment to be transferred.
- **Out-Timecode field:** The timecode of the frame after the last frame in the segment to be transferred.
- **Duration Timecode field:** The duration of the segment to be transferred.
- **Offset Timecode field:** The offset from the beginning of the clip.

To set the in- and out-timecodes for output:

1. Play the tape to the frame where you want to start recording the clip.
The timecode for the current frame appears in the Current Timecode field.
2. Click the In button beside the In-Timecode field.
The In-Timecode field is set to the current timecode. You can adjust the in-timecode using the + and - buttons (beside the In button) to advance or back up the timecode by one frame. Use the Cue In button to view the in-timecode frame in the image window.
3. Select a stop option.

Select:	To:
Stop after Frames	Stop transferring the clip after a specified number of frames. When this option is selected, an additional field is shown where you enter the number of frames. The Out and Duration timecodes are updated to reflect the number of frames in this additional field.
Stop on Timecode	Stop transferring the clip when the Out-Timecode is reached.

By default, the out-timecode is set to the end of the selected clip. If you want to stop recording at a frame before the end of the clip, either use the Stop After Frames option, or complete steps 4-5 to set the out-timecode.

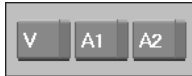
4. Play the tape to one frame after the last frame of the segment to be transferred.
The timecode for the current frame appears in the Current Timecode field.
5. Click the Out button beside the Out-Timecode box.
The Out-Timecode field is set to the current timecode and the duration is updated as follows: $\text{Duration} = \text{Out-timecode} - \text{In-timecode}$. If you change the duration, the out-timecode is automatically updated.

If needed, adjust the out-timecode using the + and - buttons (beside the Out button) to advance or back up the timecode by one frame. Use the Cue Out button to view the out-timecode frame in the image window.
6. If necessary, set an offset from the beginning of the clip in the Offset Timecode field. The Duration and Out-Timecode field are automatically updated.

Selecting Tracks for Output

You can output any combination of the tracks on a clip—video alone, video and audio, or audio alone.

Track Selection boxes



Use the Track Selection boxes to select which tracks you want to output. By default the tracks used on the clip are selected. Click a box to select or deselect it.

Outputting Clips to a Non-Blackened Tape

You can assemble output clips automatically without having to black or encode the VTR tape first, by holding down **CTRL** when you click the Process button.

When you use this option, **flame** puts the VTR into Assemble mode for the duration of the output and automatically writes the timecode from your output clip onto the tape. You can also use this option to black an entire tape by disabling V and all of the audio channels prior to output.

NOTE: With this feature, you can create a tape with discontinuous timecode if you do not do multiple assemble outputs sequentially. To avoid this, black the entire tape before outputting your clip. However, this option will allow you to quickly output to a tape if you do not have time to black it first.

To assemble a clip on output:

1. Make sure the tape has some black at the start of the tape. There must be slightly more black on the tape than the amount of preroll specified in the Engineering menu.

NOTE: If you do not have this amount of black on the tape, you will not be able to control the VTR or use preroll in the Output Clip menu.

2. On the VTR, set the TC generator switch to Internal and Regen.
3. Select the clip for output and access the Output Clip menu.
4. Add at least 2 minutes to the Out timecode.

NOTE: This will add black to the tape at the end of the clip you are outputting. If you do not add this additional black to the clip, you may encounter problems at the broadcast facility because there will be no timecode information directly after your material.

5. Hold **CTRL** and click Process.
6. The clip is output in Assemble mode.

7. Assemble mode will record all tracks (video and audio). If you want to black the tape without outputting either video and audio to the tape, disable the Video and Audio buttons. This will output black video or silence to the tape.

Inputting Clips from a VCR, Camera, or Camcorder

You can capture material without VTR control using the Input Clip menu. For example, you can input clips from a VCR, camera, or camcorder using an analog-to-digital converter between the device and the serial video input.

You can also use this procedure to capture live digital video.

For the device type, enable the Live Video line in the VTR KEYWORD section of the *init.cfg* configuration file (refer to the **flame** *Installation Guide* for details).

NOTE: For best results, use the default settings in the Engineering menu. However, you may need to experiment with such options as preroll and postroll to determine the optimal values for your setup.

To input clips from a VCR, camera, or camcorder:

1. Connect the device to the video input of the workstation via a converter.
2. On a VCR, cue the tape to a point that is well before the point that you want to start capturing (at least 5 seconds). For a camera or camcorder, set it up for live video output.
3. Make sure the Live Video line is enabled in the VTR KEYWORD section of the *init.cfg* file.
4. Open the Input Clip menu. See “To access the Input Clip menu:” on page 232.
5. Select Live Video in the Device Name box.
6. Select the appropriate Input Connection in the Engineering menu. See “Input and Output Connection” on page 242 for details.
7. In the Start Transfer Method box, select Start on Pen. This allows you to start the input by pressing the pen.

Start Transfer Method box

Start on Pen	00:00:00:00	In	-	+	Cue
Stop on Pen	00:00:00:00	Out	-	+	Cue
	00:00:00:00	Log for Batch Capture			

Stop Transfer Method box

8. In the Stop Transfer Method box, select Stop on Pen. This allows you to stop the input by pressing the pen.
9. Click Process.

10. When you see a prompt to press the pen, **flame** is ready to start capturing:
 - For VCRs, press the Play button on the VCR.
 - For cameras or camcorders, make sure you are ready to start and are receiving a stable video signal.
11. Press the pen on the tablet to start capturing.
12. Once you capture all the material you want, press the pen again to stop capture.

Outputting Clips to a VCR

You can output material without VTR control using the Output Clip menu. For example, you can output clips to a VCR using a digital-to-analog converter between the workstation and the device.

You can also use this procedure to output live digital video.

For the device type, enable the Live Video line in the VTR KEYWORD section of the *init.cfg* configuration file (refer to the **flame** *Installation Guide* for details).

When using Live Video in the Output Clip menu, you control the start and stop of output by pressing the pen on the tablet (the Start Transfer and Stop Transfer Method fields are ignored by **flame**). This is described in the following procedure.

NOTE: For best results, use the default settings in the Engineering menu.

To output clips to a VCR:

1. Connect the VCR to the video output of the workstation via a converter.
2. If you are outputting audio, connect the audio output on the workstation with the audio input on the VCR.
3. Make sure the Live Video line is enabled in the *init.cfg* file.
4. Cue the tape to a point before the point where you want to start recording.
5. Click the Output Clip button in the Library menu.
6. On the desktop reels, select the clip you want to output.
The Output Clip menu appears.
7. Select Live Video from the Device Name box.
8. Select the channels to output.
9. Select the appropriate Output Connection in the Engineering menu. See “Input and Output Connection” on page 242 for details.
10. Click Process.
11. When prompted to press the pen, press the Record button on the VCR.

12. When the VCR has reached the point at which you wish to record the clip, press the pen.

13. Once the clip is output, press the Stop button on the VCR.

NOTE: To abort the process, press the pen again.

NOTE: Only the ProntoVision HD DDR is currently supported for **flame**.

DDR Input and Output

With **flame**, you can input and output material to and from a DDR (digital disk recorder) using an Ethernet or SCSI connection. You can have more than one DDR connected to the system, but you can only use one device at a time. In other words, there can be only one load or save operation in progress at a time.

Use Ethernet transfer when the DDR is connected to a remote machine on the network; use SCSI transfer when the DDR is connected to the local machine. Transfers are much faster over SCSI.

You can transfer clips as a background process, continuing to work in **flame** during the transfer.

NOTE: The only HD (High Definition) DDR that is currently supported is the DVS Prontovision HD DDR.

Inputting a Clip from a DDR

Use the DDR Library menu to load clips from a DDR to the desktop.

To input a clip using SCSI or Ethernet transfer:

1. Click the DDR Library button in the Library menu.

The DDR Library menu appears.

2. Select SCSI Transfers or Ethernet Transfers from the Connection Type box.

Use Ethernet transfer when the DDR is connected to a remote machine on the network. Use SCSI transfer when the DDR is connected to the local machine. Transfers are much faster over SCSI.

3. Select the DDR you are using from the Device Name box.

If the DDR you are using does not appear in the list, see “Setting the Configuration File for DDR Input and Output” on page 255.

4. Click Load.
5. The Frames field appears.



6. In the Start Frame field, enter the number of the first frame to load from the DDR. This number is the offset from 0 of that frame on the DDR.
7. Enter the number of frames to be transferred in the Frames field.
8. Select the destination.

The clip is loaded from the DDR and appears on the destination reel.

Loading Clips in the Background

You can use the Load Background option to load clips from a DDR as the background process.

NOTE: The software controlling the DDR must support background processing. If the selected DDR does not support this option, the Background Load button does not appear in the menu.

To load a clip in the background:

1. Click the DDR Library button in the Library menu.
The DDR Library menu appears.
2. Select Ethernet Transfers from the Connection Type box.
3. Select the DDR you are using from the Device Name box.
If the DDR you are using does not appear in the list, see “Setting the Configuration File for DDR Input and Output” on page 255.
4. Click the Background Load button.
The Frames field appears.
5. In the Start Frame field, enter the number of the first frame to load from the DDR. This number is the offset from 0 of that frame on the device.
6. Enter the number of frames to be transferred in the Frames field.
7. Select the destination.
An empty clip appears on the destination reel.
8. Click Exit to return to the Library menu.

The frames allocated for the empty clip are filled as the background load progresses. You can use other modules while the clip is being loaded from the device.

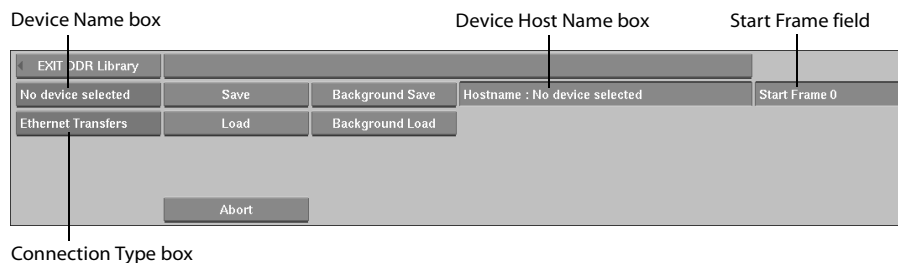
Outputting a Clip to a DDR

Use the DDR Library menu to save clips from the desktop to a DDR through an SCSI or Ethernet interface.

To output a clip:

1. Click the DDR Library button in the Library menu.

The DDR Library menu appears.



2. Select SCSI Transfers or Ethernet Transfers from the Connection Type box.
3. Select a DDR from the Device Name box.
If the DDR you are using does not appear in the list, see “Setting the Configuration File for DDR Input and Output” on page 255.
4. Click Save.
5. In the Start Frame field, enter the starting frame number where the clip is to be written on the DDR. This number is the offset from 0 of that frame on the DDR.
6. Select the clip to save from the desktop reels.
The clip is output to the selected DDR.

Saving Clips in the Background

You can use the Save Background option to save clips to a DDR as a background process.

NOTE: The software controlling the DDR must support background processing. If the selected DDR does not support this option, the Background Save button does not appear in the menu.

To save a clip in the background:

1. In the Library menu, click the DDR Library button.
The DDR Library menu appears.

2. Select Ethernet Transfers from the Connection Type box.
3. Select the DDR to use from the Device Name box.
If the DDR you are using does not appear in the list, see “Setting the Configuration File for DDR Input and Output” on page 255.
4. Click the Background Save button.
5. In the Start Frame field, enter the starting frame number where the clip is to be written on the DDR. This number is the offset from 0 of that frame on the DDR.
6. Select the clip to save from the desktop reels.
7. Click Exit to return to the Library menu.
You can use other modules while the clip is being saved to the DDR.

Cancelling a Load or Save Operation

You can cancel the load or save operation any time by pressing the pen against the tablet. If you are loading or saving in the background, you can cancel the operation by clicking the Abort button in the DDR Library menu.

Setting the Configuration File for DDR Input and Output

You may need to modify your *init.cfg* configuration file if your DDR does not appear as an option in the DDR Library menu. There are two keywords that must be set, one for the DDR itself, and one for the connection type (Ethernet or SCSI).

NOTE: You can define both an Ethernet and an SCSI connection for a DDR.

To define a DDR in the *init.cfg* configuration file:

1. Exit **flame** if you are currently using it.
2. Open a UNIX shell.
3. Use the jot text editor to open the *init.cfg* configuration file.
4. Scroll through the configuration file until you find the DDR keyword.
5. Remove the number sign (#) from the definition for your DDR.

NOTE: Only the ProntoVision HD DDR is currently supported for **flame**.

To define the connection type:

1. Open the *init.cfg* configuration file.
2. Scroll through the configuration file until you find the Ethernet and SCSI keywords. You should see something similar to the following:

```
# ETHERNET KEYWORD
# -----
```

```
# The Ethernet keyword defines the devices used for clip
transfers using the Ethernet Library.

#

# Syntax: Ethernet <type>, <host name>, <duration>, <start
location>

#

#Ethernet          Accom, accom1,          815, 0
```

NOTE: Host name is the name of the DDR on the network.

3. Under the Ethernet or SCSI keyword, remove the number sign (#) in front of the line that corresponds to the type of DDR you are using.

To:	Enable a DDR under the:
Use a DDR through the network	Ethernet keyword.
Use a DDR connected directly to the machine you are using	SCSI keyword.

Note that SCSI transfer is faster than Ethernet transfer, but the DDR must be within 6 m (18 ft.) for SCSI transfer due to cable length limitations. Note also that you can define the same DDR under both keywords.

4. From the File menu, choose Save, and then choose Exit.
5. Restart **flame**.

Clip I/O with the Sony HDCAM Codec

Using the Sony HDCAM™ codec, which has been integrated into **flame**, you can input and output 1920 x 1080 HD images with the 601 video I/O card of your system when connected to a Sony HDW-500 HDCAM VTR.

You capture the compressed image data from the VTR in real time, and the codec (compression/decompression software) then decompresses the images after capture into full-resolution HD images. Prior to output, you use the codec to compress the HD images back into the HDCAM compressed format. The mechanism used to transfer compressed image data over a normal 601 video link is referred to as SDTI (Serial Data Transport Interface).

The compressed data is stored in an encoded format which is placed in both the frame and its proxy image. When outputting the clips, the data in the proxy is used. For this reason, proxies must be defined in the framestore setup when using the codec.

NOTE: The HDCAM codec option is purchased with an optional licence. “HDCAM” is a trademark of Sony Corporation.

24p Mastering with the Codec

Captured and decoded HDCAM images have a frame rate of 59.94 Hz interlaced. The 2:3 Insertion and Removal options for clip input and output, described in Chapter 15, “24p Mastering,” are not available when using the codec. However, you can still perform 24p mastering with clips that have been input with the codec, by removing pulldown after capture and adding it back before recording to tape. To do this, use the Film Compress and Film Expand options as described in “Compressing Clips from 30 to 24 fps” on page 618, and “Expanding Clips from 24 to 30 fps” on page 621.

Hardware Configuration for the HDCAM Codec

The following configuration is required to use the codec:

- An OCTANE video I/O card
- A Sony HDW-500 HDCAM VTR

NOTE: In this release, SDTI transfers with the HDCAM VTR and HDCAM software encoding/decoding is only supported with the HDW-500 VTR operating at 1920 x 1080 resolution and at 59.94 Hz interlaced frame rate. The HDW-F500 VTR operating at 24 Hz progressive is not yet supported.

- The HKDV-506 SDTI Data Dubbing Option board for the HDW-500 VTR. This is an optional board; you can check for its presence by entering the Maintenance menu on the front panel of the VTR and selecting the “OPTION INFO” function.

Note that when doing video I/O using the codec, you need a framestore which can sustain a bandwidth of 40Mb/second. You do not need a framestore which can sustain the full uncompressed HD bandwidth.

Clip Input Using the HDCAM Codec

Use the Input Clip menu when capturing with the codec. The procedure for inputting clips using the codec is similar to that described in “Inputting Clips from a VTR” on page 232. Refer to that section for details on setting particular options in the Input Clip and Engineering menus.

Decoding and encoding performance depends on the number and speed of the CPUs in the system. Encoding usually takes about twice as long as decoding. You have the option of decoding the clip automatically right after capture, or doing it manually from the Library menu at a later time.

To input a clip using the codec:

1. Make sure the HDW-500 HDCAM VTR is configured for 59.94 Hz and not 60.00 Hz operation. A green indicator on the VTR front panel indicates 59.94 Hz or 60 Hz. The SDTI interface is activated only when the VTR is running at 59.94 Hz.

To change the rate, follow these steps:

- Select the Maintenance menu by pressing on the recessed button on the front panel.
 - Select MAINTÉ EXEC by pressing on the SFT and F8 buttons.
 - Select OTHERS CHECK by pressing on the F9 button.
 - Select SYSTEM MENU by pressing on the F9 button.
 - Select HD FRQ by pressing on the F2 button. Repeat until you have selected 59.94 Hz.
 - Exit and confirm by pressing twice on the F9 button.
2. Make sure the HDCAM VTR is enabled in the VTR token of the *init.cfg* configuration file (for details, refer to the **flame** *Installation Guide*).
 3. Connect the SDTI output (Dub Out) of the Sony HDW 500 HDCAM VTR to the OctaneVideo In 1 input.
 4. Create a project in an 8-bit 1920 x 1080 partition. Set the Frame Depth to 24 to create an 8-bit partition. Select Proxies Stored, and set the Proxy Size to 0.476 or larger, and the Proxy Depth to 24. The optimal proxy size is 0.476 — more disk bandwidth is needed for larger sizes, which could potentially cause dropped frames on input and output.

You should use the *1920 x 1080_SDTI.cfg* template when creating a project that will use the HDCAM codec. The Video token in the *init.cfg* configuration file should be set to OctaneVideo.

5. From the Library menu, open the Input Clip menu.

The image you see in the image window is an encoded (compressed) signal, which appears as coloured noise. To view the actual image while capturing, connect the HD broadcast monitor directly to the HD SDI output of the VTR.

NOTE: If the monitor does not have a direct HD SDI input, an HD Digital to Analog converter is required.

6. In the Device Name box, select the HDCAM VTR.
7. Open the Engineering menu.
8. Enable the HDCAM I/O button.

The HDCAM Decode button appears.

NOTE: If you are unable to select the HDCAM I/O option, it means that the partition you are in does not have 1920 x 1080 8-bit resolution, that proxies are not defined, or that the proxy size is below 0.476 (see step 2, above).

9. If you want to decode the clip automatically following capture, enable the HDCAM Decode button.
10. Make sure that the Field Dominance is set to 1.

11. Set any other options in the Engineering and Input Clip menus as described in “Inputting Clips from a VTR” on page 232.

NOTE: HDCAM I/O requires 10 bits to be transferred, so 10-bit precision is automatically used (the setting in the Precision box is ignored).

12. Process the clip. If you opted to decode as a post process, decoding takes place right after capture.
13. Exit to the Library menu.

The captured clip appears on the desktop.

- The proxy of a decoded clip appears as a regular HD clip.
- The proxy of an encoded clip appears as coloured noise. To decode the clip manually, see “Manual Decoding and Encoding” on page 260.

Capturing with an EDL

You can also use an EDL when capturing clips with the codec. As with clip I/O without an EDL, you have the option of decoding the clips as they are being input, or doing it afterwards on the desktop. With the second method you spend less time using the VTR, since decoding is done after capture.

- To decode after capture, capture the material as you normally would with the EDL, then multiple-select the clips for decoding on the desktop.
- To decode during capture, load the EDL as usual and access the Auto-Capture menu. In the Engineering and Auto-Capture menus, select settings for automatic decoding as described in “Clip Input Using the HDCAM Codec” on page 257. Decoding is applied to each event immediately after its capture.

NOTE: EDL 2:3 removal of pulldown is not available when using the codec.

Manual Decoding and Encoding

To work with clips captured with the codec, they must first be decoded. To decode several clips, place them together on a reel.

To manually decode one or more clips:

1. In the Library menu, click the HDCAM Codec button.

The Codec Operation and Selection Range boxes appear at the right of the menu.



2. Select Clip to decode a single clip, or Reel to decode all the clips on a reel.
3. Select HDCAM Decode from the Codec Operation box.
4. Select the encoded clip (or any encoded clip on a reel containing the set of encoded clips you want to decode), then select a destination reel.

Decoding begins and when the process is finished, any decoded clips appear on the destination reel. New clips have the prefix HDCAMDEC- to identify them as decoded clips.

When you are ready to output a clip to the Sony HDCAM VTR using the codec, encode the clip first, as described below. To encode several clips, place them together on a reel.

To encode one or more clips prior to output:

1. In the Library menu, click the HDCAM Codec button.

The Codec Operation and Selection Range boxes appear at the right of the menu.

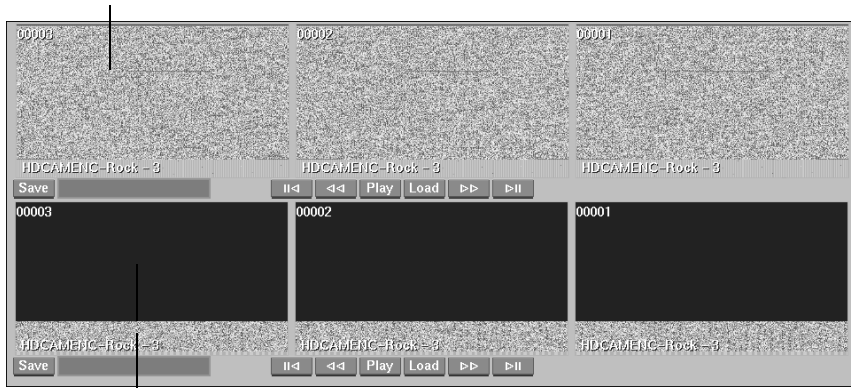
2. Select Clip to encode a single clip, or Reel to encode all the clips on a reel.
3. Select HDCAM Encode from the Codec Operation box.
4. Select the clip you want to encode (or any clip on a reel containing the set of clips you want to decode), then select a destination reel.

When the process is complete, any encoded clips appear in the destination reel. The new clips have the prefix HDCAMENC- to identify them as encoded clips.

Regenerating Proxies

When outputting encoded clips, the proxies must be generated correctly. The correct generation of proxies is lost when transferring encoded clips over the **wire** network, and also when retrieving clips from an archive.

Encoded clip with the correct codec proxy



Encoded clip that has lost the correct codec proxy

If this occurs, regenerate the proxies using the HDCAM Proxy Regen button in the Library menu before outputting the clips. To do so, select HDCAM Proxy Regen from the Codec Operation box then select the clip to be regenerated. The data from each frame is copied back to the proxy, and the clip is ready to output.

Clip Output Using the HDCAM Codec

Use the Output Clip menu to output with the codec. The procedure for outputting clips using the codec is basically the same as that described in “Outputting Clips to a VTR” on page 245. For complete details on setting particular options in the Output Clip menu, refer to that section.

The current revision of the HDCAM firmware does not allow audio input via the AES or Analog audio inputs when the video input is set to “SDTI”. Thus it is not possible to output audio at the same time as HDCAM; compressed video data is output via SDTI. To output a clip with audio, first output its video using the following procedure. Then switch the “VIDEO IN” setting (option 701) on the VTR back to “SDI” and do an Audio Only output (for more details, see step 3 on page 294.)

NOTE: Make sure the HDCAM videotape you are outputting to has been blacked with a 1080 active line video signal and not a 1035 active line video signal. You cannot output HDCAM data to a tape formatted for a 1035 line signal. If you are using the internal signal generator of the HDCAM VTR to black your videotape, set the T0 VID SG LINE parameter to 1080 in the VTR setup menu.

To output a clip using the codec:

1. Make sure the HDW-500 HDCAM VTR is configured for 59.94 Hz and not 60.00 Hz operation. A green indicator on the VTR front panel indicates 59.94 Hz or 60 Hz. The SDTI interface is only activated when the VTR is running at 59.94 Hz.

To change the rate, follow these steps:

- Select the Maintenance menu by pressing on the recessed button on the front panel.
 - Select MAINTENANCE EXEC by pressing on the SFT and F8 buttons.
 - Select OTHERS CHECK by pressing on the F9 button.
 - Select SYSTEM MENU by pressing on the F9 button.
 - Select HD FRQ by pressing on the F2 button. Repeat until you have selected 59.94Hz.
 - Exit and confirm by pressing twice on the F9 button.
2. Connect the SDTI input (Dub In) of the Sony HDW 500 HDCAM VTR to the OCTANE video board output.
 3. On the HDW-500 VTR, go to the VTR SETUP menu and change option 701, “VIDEO IN”, from “SDI” to “SDTI”. This specifies the input from which the VTR will accept video data. Remember to change this setting back to “SDI” if you want to use the HD-SDI inputs on the VTR.
 4. On the desktop, select the clip to output. This clip must be encoded as described in “To encode one or more clips prior to output:” on page 260. Also, verify that the proxy is generated correctly. See “Regenerating Proxies” on page 261 for details.
 5. Open the Output Clip menu.
 6. In the Device Name box, select the HDCAM VTR. If it is not available, exit **flame** and enable the device in the *init.cfg* configuration file (for details, refer to the *flame Installation Guide*).
 7. Open the Engineering menu.
 8. Enable the HDCAM I/O button.
 9. Make sure that the Field Dominance is set to 1.
 10. Set any other options in the Engineering and Output Clip menus as described in “Outputting Clips to a VTR” on page 245.

NOTE: HDCAM I/O requires 10 bits to be transferred, so 10-bit precision is automatically used.

11. Process the clip.

Data Tape Input and Output

Using the Tape Library menu, you can store and load **flame** material to and from Exabyte or DAT tapes. This is useful for archiving material. The advantages of using the Tape Library menu are:

- The tape device runs in the background, so you can work in **flame** as you archive material.
- You can access up to 10 tape devices concurrently; each archive process runs as a background process. Note that each additional background process causes the system to run at a slower rate.

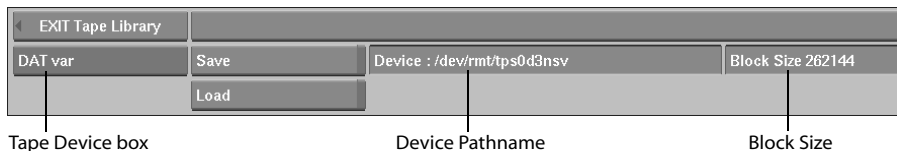
The main disadvantage of this archive method is that you can only record one clip onto any one tape. Any uncommitted editing information (cuts, splices, dissolves, timewarps) associated with the clip will be lost. If you want to record several clips onto a tape, you must splice them together before starting. Should you need to restore the archive, you will have to cut the restored clip into separate clips again.

All tape devices being used must be declared in the TAPE KEYWORD section of the *init.cfg* configuration file (this is described in the **flame** *Installation Guide*).

To save a clip to tape:

1. Insert a tape in the tape drive.
2. Click the Tape Library button in the Library menu.

The Tape Library menu appears.



3. Select the tape device you are using from the Tape Device box. The names of all the tape devices that are declared in the *init.cfg* configuration file are listed.
4. If needed, modify the Device Pathname or Block Size. The default settings are those specified for the selected tape device in the *init.cfg* configuration file.
5. Click Save.

The File Format box appears. Currently, only the flame format is supported.



6. Optionally, enable the 4.0 compat button to create a tape which can be read by **flame** 4.0.x.
7. Select the clip to save.

The clip is saved on the tape.

To load a clip from tape:

1. Insert the tape in the tape drive.
2. Click the Tape Library button in the Library menu.

The Tape Library menu appears.

EXIT Tape Library			
DAT var	Save	Device : /dev/rmt/tps0d3nsv	Block Size 262144
	Load		

Tape Device box Device Pathname Block Size

3. Select the tape device you are using from the Tape Device box. The names of all the tape devices that are declared in the *init.cfg* configuration file are listed.
4. If needed, modify the Device Pathname or Block Size. The default settings are those specified for the selected tape device in the *init.cfg* configuration file.
5. Click Load.

The Tape Load options appear.

Centre/Crop	— Fit Method box
Start Frame 1	
Frames All	

6. Select a fit method from the Fit Method box. For a full description of the available options, see “Fit Method” on page 328.
7. In the Start Frame field, enter the number of the first frame to load from the tape. The first frame on the tape is referred to as frame 1.
8. In the Frames field, enter the number of frames to load.
9. Select a destination.

The material on the tape is loaded as one continuous clip onto the destination reel. If you want to keep this clip for future use, save it in a clip library.

Cancelling a Tape Load or Save Operation

During load or save operation, the Load and Save buttons for the selected tape device are disabled and an Abort option appears. You can cancel the load or save operation by clicking Abort.

Using Another Tape Device

You can use another tape device while the current device is busy. Select the device from the Device Name box and click Load or Save as required.

Troubleshooting

Use the following troubleshooting tips if you encounter problems with clip input or output.

Problem	flame Is Dropping Frames in Clip Input
Possible Cause	You may need to configure the kernel or tune the system.
Solution 1	You can configure the kernel to reserve a CPU for flame video input and output.

WARNING: Configuring the kernel should only be attempted by a system administrator or someone who is familiar with the UNIX environment.

To configure the kernel:

1. Log in as *root* and open a UNIX shell.

You must have root privileges in order to configure the kernel.

2. Go to the */var/sysgen/system* directory by typing:

```
cd /var/sysgen/system
```

3. Edit the *irix.sm* text file by typing:

```
jot irix.sm
```

4. Add one of the following lines after line 615:

- If you have four CPUs or more, type **NOINTR: 1 2 3**
- If you have two CPUs, type **NOINTR: 1**

5. Save the *irix.sm* text file. Exit *jot* by choosing Exit from the File menu.

6. At the prompt, type:

```
autoconfig -f
```

The kernel is configured to better optimize video input and output.

7. Reboot the system by typing:

```
sync; reboot
```

You may now log in to your **flame** account and start **flame**.

Problem **flame** Is Dropping Frames in Clip Input

Solution 2 **To tune the system:**

1. Log in as root.

2. Start the *systune* utility by typing:

```
systune -i
```

The *systune* prompt appears.

3. Increase the number of buffers by typing:

```
nbuf = 6000
```

4. Type **y** when the following message appears:

```
Do you really want to change nbuf to 6000 (0x1770)?
(y/n)
```

5. Exit the *systune* utility by typing:

```
quit
```

6. Reboot your computer by typing:

```
reboot
```

NOTE: You must reboot your computer for the change to take effect.

To check in the project configuration file:

Make sure the Hires (high resolution) line in the project configuration file looks like this for NTSC and PAL projects respectively:

```
Hires 30 Hzf
```

```
Hires 25 Hzf
```

To turn off daemons:

Make sure the following daemons are turned off by typing:

```
chkconfig desktop off
```

```
chkconfig directoryserver off
```

```
chkconfig fontserver off
```

```
chkconfig mediad off
```

```
chkconfig objectserver off
```

```
chkconfig soundscheme off
```

Problem	I keep getting Lost Playback Speed errors. How do I fix it?
Solution	<p>To avoid lost playback speed errors, you should use the following settings:</p> <ul style="list-style-type: none"> • If you are using a D1 or Digital Betacam, set the Output Sync to “House” to guarantee a reliable output sync source. Using “Digital1” selects the sync information embedded in the digital input stream and gives unpredictable results if no input signal is present.
Problem	I keep getting retries during clip input. What’s up?
Possible Cause	flame should be the only program running on your workstation. Running other programs, such as <i>Xclock</i> , <i>Mailbox</i> , <i>jot</i> or Netscape® products, at the same time as flame may cause retries.
Solution	<p>You can keep other programs from running by setting the DL_TO_STOP environment variable. Use this variable to specify a list of programs to be stopped whenever you capture, record, or play clips.</p> <p>To use the DL_TO_STOP environment variable:</p> <ol style="list-style-type: none"> 1. Log in to your flame account. 2. In the UNIX shell type: <pre>setenv DL_TO_STOP "<program_name>"</pre> <p>where <code><program_name></code> is the name of the program that you want stopped. For example, if you want to stop the <i>jot</i> and <i>Xclock</i> programs from running while using flame, type:</p> <pre>setenv DL_TO_STOP "jot" "Xclock"</pre> <p>NOTE: If you use the DL_TO_STOP environment variable, there may be a slight delay when entering and exiting tools that use video input and output.</p> 3. Start flame. <p>During your flame session, whenever you enter a tool or menu that uses video input and output, the programs that you specified stop. They restart when you return to the EditDesk.</p>

Problem	I'm still getting retries during clip input Now what?
Possible Cause	You might need to change your sync signal.
Solution	<p>If you are experiencing retries, you can switch the input sync between the house sync and the input connection sync. For example, you could switch to digital1 when using parallel1 input. Output sync is always set to house sync.</p> <p>You should also make sure that you are using a good house sync, for this will better synchronize your VTR and flame software and reduce the likelihood of retries.</p>

*Use the EDL module in **flame** to import and export EDLs in any of the standard formats and edit them.*

The EDL module in **flame** uses Edl Pro™ Edit Decision List conversion software, provided by Alba Editorial, Inc. Alba and Edl Pro are trademarks of Alba Editorial, Inc. that are registered in the United States and are pending in other countries.

Summary

In this chapter, you learn about

- “The EDL Menu” on page 270
- “Loading an EDL” on page 272
- “Editing an EDL” on page 274
- “Preferences For EDLs” on page 285
- “Auto-Capturing an EDL” on page 287
- “Assembling an EDL” on page 292
- “Searching EDLs” on page 294
- “Generating an EDL” on page 296
- “Exporting an EDL” on page 299
- “Importing EDLs through Your Network” on page 302

About EDLs

Use the EDL module to import and edit an Edit Decision List (EDL), capture clips from a VTR, and automatically assemble the clips within **flame**. You can also create and save new EDLs as a file or export them to tape.

The following EDL standards are supported by **flame**:

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

- CMX 340
- SONY 910
- SONY 900

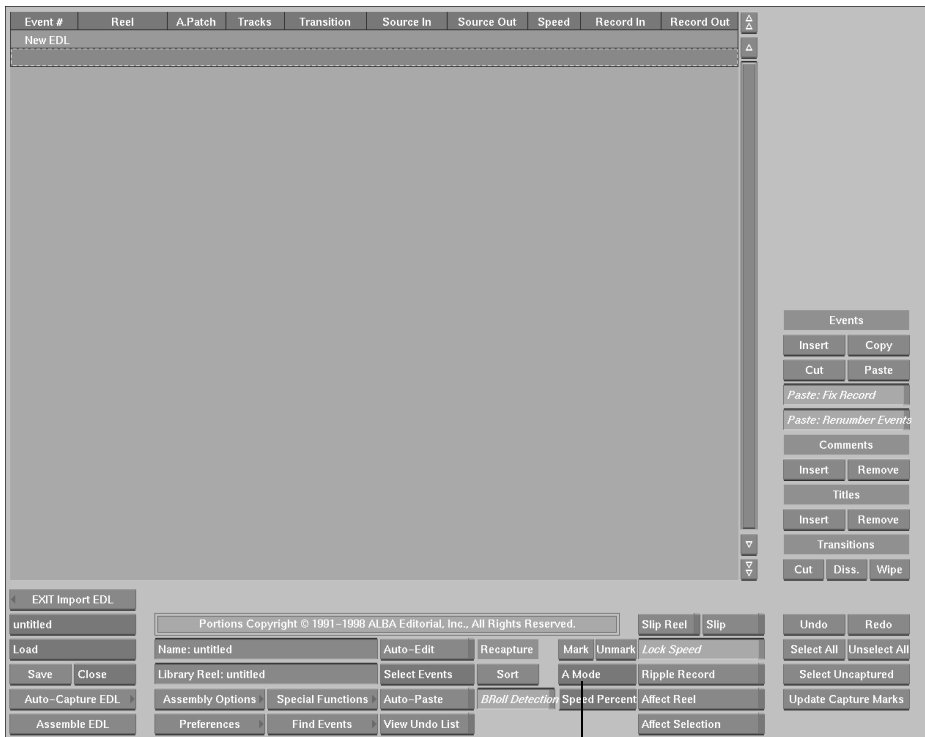
These standards are supported for the following basic functions only: cuts, splices, dissolves, SMPTE wipes, speed variations (including freeze frames), and direction changes (Forward/Reverse). Avid name and audio patching comments are also supported.

NOTE: It is highly recommended to use EDLs in the CMX 3600 format.

The EDL module supports EDLs that use framerates of 24, 50, or 60 fps.

The EDL Menu

Use the EDL menu to load, edit, auto-capture, and assemble clips and to save your EDLs. To open the EDL menu, click the EDL button in the Editing menu.



Sort Mode box

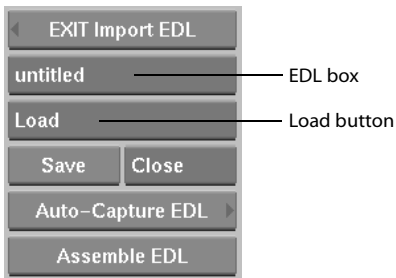
In the EDL menu, the work area of the screen is divided into nine columns.

Where: **Is:**

Event # The Event Number, followed by a capture indicator.

Where:	Is:
Reel	The name of the tape containing the source clip.
A. Patch	The Audio Patch information.
Tracks	The track for the Edit (shown as a combination of: V, 1, 2, 3, 4).
Transition	The type of transition between the clips. C is used for cuts, D<length> is used for dissolves, and W<wipe code><length> is used for SMPTE wipes.
Source In	The starting timecode of the segment in the source clip.
Source Out	The ending timecode of the segment in the source clip. (CTRL-CLICK the heading to view Source duration.)
Speed	The speed at which the source clip is timewarped in the Edit. The value is preceded by a negative sign if the effect is a reverse. The column is blank if the speed value is 100% (no timewarp).
Record In	The starting timecode of the segment in the result clip.
Record Out	The ending timecode of the segment in the result clip. (CTRL-CLICK the heading to view Record duration.)

In the EDL menu, the EDL Status box shows the format of the currently loaded EDL. The EDL box shows the name of the current EDL file. You can load up to 25 EDLs at a time. All loaded EDLs appear in the EDL box.



The EDL file may contain more than one page of Entries. To see all the Events in the EDL, use the scroll bar to scroll up or down the list.

Events, Edits and Entries

An EDL contains Events, Edits, and Entries. An Event is a complete element within the EDL such as a shot, dissolve, or wipe. An Edit is an individual component which makes up an Event, such as an outgoing shot and an incoming shot. An Entry is an individual value for any variable in the Edit, such as dissolve length, Source In, or Speed.

Edits

0007	010	V	C	01:01:12:09	01:01:13:27	00:01:05:24	00:01:07:12
0008	010	V	C	01:01:23:27	01:01:24:23	00:01:07:12	00:01:07:28
0008	010	V	D 10	01:01:34:09	01:01:35:08	00:01:07:28	00:01:08:27

Event

Entries

Every Event has a different Event Number. Some Events, such as dissolves and wipes, consist of two Edits. The two Edits in a dissolve (outgoing and incoming shots) have the same Event Number.

Loading an EDL

Use the file browser in the EDL library to select an EDL. You can load as many EDLs as you want simultaneously.

To load an EDL:

1. Click the Load button.
The file browser and the EDL library menu appear.
2. Enter the extension of the EDL you want to load in the File Extension box. See “EDL File Extensions” on page 273.
3. Enable or disable the Fix TW Match Frames option. See “Match Frame Errors” on page 273.
4. Enable or disable the Same Library Reel option. See “Same Library Reel” on page 273.
5. Using the file browser, select the EDL you want.

The EDL loads and the EDL menu reappears.

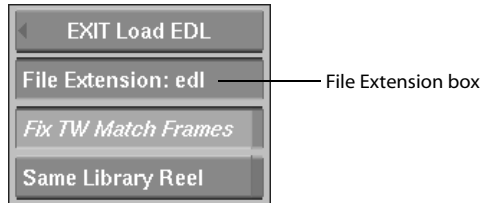
NOTE: For the best results, use clean EDLs (EDLs in which there are no overlaps in the Record timecodes). If the Record timecodes contain gaps, the resulting multitrack will contain gaps in the positions specified in the EDL.

If you have already opened EDLs, they appear in the history list. Press and hold the Load button to display the last four opened EDLs in the history list.

EDL File Extensions

The File Extension box in the EDL Library menu defines which files are listed in the EDL library. The EDL library appears when you click the Load button. Only EDLs that have the extension specified in the File Extension box are listed in the EDL library.

To specify a file extension, click the File Extension box and use the keyboard to enter the extension.



Loading an EDL from a PC

When you load EDLs from IBM-PC compatible computers, you must ensure the correct extension appears in the File Extension box. Since DOS only allows file names to use uppercase letters, EDL files have the file extension “.EDL”. When you want to import one of these files, you must do one of the following:

- To change the extension for the current session, enter the extension “EDL” in the File Extension box.
- To change the extension for all sessions, change the default extension on the EDL line in the configuration file from “edl” to “EDL.”
- Enter no extension in the File Extension box to view all files in the directory.

Same Library Reel

Use the Same Library Reel option to load a series of EDLs and assemble them on the same reel in the clip library. If you already have an EDL loaded, and then load another EDL with the Same Library Reel option enabled, both EDLs will be assembled on the same reel in the clip library. See “Assembling an EDL” on page 292.

Match Frame Errors

When you import an EDL that contains timewarps, a match frame error may occur. This can cause an unwanted cut at the point where the timewarp begins in your EDL.

To fix match frame errors, **flame** has a Fix Match Frame feature that automatically detects a match frame problem and fixes it in your EDL. This feature is enabled by default. To disable the Fix Match Frame feature, go to the Import EDL menu, then to the Load menu. When the Fix TW Match Frame button is light grey, the feature is enabled. Click the button to disable the feature.

Sorting an EDL

You can change the way **flame** sorts an EDL before or after it is loaded.

To sort an EDL:

1. Select the appropriate sort order from the Sort Mode box.

Select:

To:

- | | |
|---------------|--|
| A Mode | Sort the EDL according to the Record In timecode. If you want to view the EDL in the order of the final assembly, sort the EDL in A Mode. |
| B Mode | Sort the EDL according to the Reel and Record In timecode. |
| C Mode | Sort the EDL according to the reel number and Source In timecode. If you want to view the EDL in the order the clips are captured, sort the EDL in C Mode. EDLs are always captured in C Mode, regardless of the sort mode you have selected. See "Auto-Capturing an EDL" on page 287. |
| S Mode | Sort the EDL by the Source In timecode regardless of reel number. This sort mode is useful for multi-camera real-time EDLs. |
| by Event# | Sort the EDL by Event Number. |
| by Tracks | Sort the EDL by tracks. In this sort mode, video tracks are placed at the bottom of the list, and the highest audio track is placed at the top of the list. |
| by Uncaptured | Sort the EDL by placing all uncaptured Events at the top of the list. |

2. Click the Sort button.

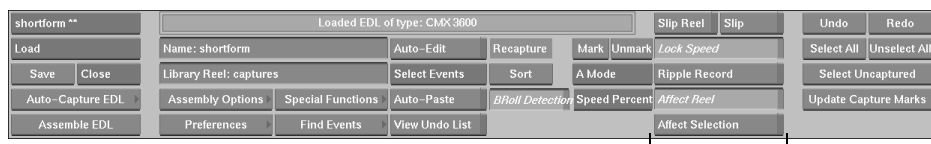
The EDL is sorted in the order you selected.

Editing an EDL

Once you load an EDL, you can edit any value except the Event Number. For example, you can change the reel name for single or multiple Events, change the audio patching information, change the transition rate and speed value of dissolves, change a cut to a dissolve or wipe, and modify the source and record timecodes of Events.

You can open multiple EDLs and copy and paste Events between them. You can also use Auto Edit mode to quickly make changes to the Entries you specify.

NOTE: EDLs that have references to a generated output tape (EDLs created with the **flame** Export EDL feature using the Output All or Output Generated options) are opened as Read Only and cannot be edited. To edit a read only EDL, save it to the file system with a different name.



Editing Control options

Options For Editing EDLs

When you modify an Entry in an EDL, there are six options that control how other Entries are affected. Make sure these options are either enabled or disabled, depending on how you want to edit the EDL.

Enable:	To:
Slip Reel	Slip all sources on the same source reel.
Slip	Slip a single source.
Lock Speed	Edit the timecode without changing the speed value for the Event.
Affect Reel	Affect every relevant Entry on the same reel when you modify a single Entry.
Affect Selection	Affect only the selected Events.
Ripple Record	Ripple record timecodes when you modify a timecode Entry.

Before you modify an Entry, make sure you have selected the preferences you want; otherwise, you may affect Entries you do not want edited.

Editing a Reel Name

You can change a single Event's reel name, or the reel name for all Events on the same reel. This is useful if the Events you want to capture are on a different reel than listed in the EDL.

To edit a reel name:

1. To change the reel name for all Events on the same reel, enable the Affect Reel option. To change only the selected reel name, disable the Affect Reel option.
2. Click the Reel Entry in the Event you want to edit and enter the new name.
 - If the Affect Reel option is enabled, the reel name for all Events on the same reel automatically changes.
 - If the Affect Reel option is disabled, only the reel name for the selected Event changes.
 - If the Affect Selection option is enabled, only the reel names for selected Events change.

Editing Audio Patching

If you want to capture audio to a different track than specified in the EDL, you can change the audio patching for a single Event, selected Events, or all Events on the same reel. For example, if an Edit specifies the audio tracks as 1 and 2, but you want to capture the audio from tracks 3 and 4, use the Audio Patching menu to reroute the audio during auto-capture.



To change audio patching:

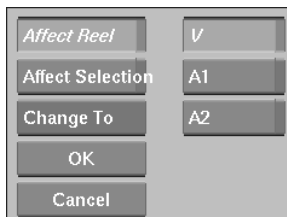
1. Select the Event(s) you want to modify.
2. Click the entry in the A. Patch column.
The Audio Patching menu appears.
3. Enable or disable the Affect Reel and Affect Selection boxes. See “Options For Editing EDLs” on page 275.
4. Reroute the audio in the Audio Patch boxes.
5. Click OK.

NOTE: To exit the Audio Patching menu without changing the patching information, click Cancel.

Editing Tracks

You can edit the tracks of an Event in the EDL. This is useful if you want to modify which tracks are captured when you auto-capture or assemble the EDL. See “Auto-Capturing an EDL” on page 287 and “Assembling an EDL” on page 292 respectively.

For example, if the Track Entry for an Event is V1 and you want to capture only the video track (V), change the Track Entry to V. When you click Auto-Capture on the EDL menu, only the video track is captured for that Event.



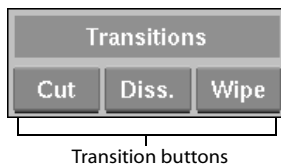
To edit the tracks for an Event:

1. Select the Track Entries in the Event you want to edit.
The Track menu appears.
2. Enable the tracks you want to edit.
3. Enable or disable the Affect Reel and Affect Selection boxes. See “Options For Editing EDLs” on page 275.
4. In the Edit Mode box, select the mode you want to use.

Select:	To:
Change To	Replace tracks in the Event with tracks you specified in the Track menu.
Add	Add the tracks you specified in the Track menu to the tracks in the Event.
Filter	Keep only the tracks you specified in the Track menu. For example, if an Event is “V12” and you specify “V1”, audio track 2 is filtered out, and the resulting Event has “V1”.
Flip	Reverse the status of tracks you specified in the Track menu. For example, if an Event is “V23” and you specify “V34”, the resulting Event has “24”.

Editing Transitions

You can change any transition to a cut, dissolve, or SMPTE wipe. If the effect of an Event is a dissolve, you can edit its duration (Transition Rate). If the effect of an Event is a wipe, you can select the type of wipe in the SMPTE Wipe library.

**To change the type of transition:**

1. Select the transition in the EDL.
2. Click the type of transition you want:
 - Click Cut to make the transition a cut.
 - Click Diss to make the transition a dissolve.
 - Click Wipe to make the transition a wipe.

You can also change a cut into a dissolve by clicking on the Transition Entry of the cut. You can change a dissolve to a cut if you set the Transition Rate to 0, and if the Allow Zero-Length Transitions option is disabled. See “Preferences For EDLs” on page 285.

By default, when you turn a transition into a wipe, it is created as SMPTE type 001. Change the wipe type in the Transition Entry, or use the Wipe Chooser to select a different type of wipe. See “Editing Wipes” on page 278.

Editing Dissolves

Dissolves are represented in the EDL by two consecutive Edits with the same Event Number. The first Edit represents the outgoing shot of the dissolve and the second Edit represents the incoming shot of the dissolve. The dissolve is listed in the Transition column of the second Edit as *D n* (where *n* is the dissolve rate).

The Transition Rate appears in the Transition Entry of the second Edit in the dissolve. The Start Location for the dissolve is also indicated in the Transition Entry.

If the EDL was exported from **flame**, the dissolve can be Centred, From Cut, or Up To Cut. The start location for the dissolve appears beside the Transition Rate.

The following illustration is a typical dissolve in an EDL.

Outgoing shot							
0020	002	V	C	09:07:36:15	09:07:36:26	10:00:16:23	10:00:16:23
0020	002	V	D 11	09:05:10:13	09:05:10:24	10:00:16:23	10:00:17:04
Incoming shot		Transition Rate					

If you change the Transition Rate of a dissolve, the Source Out of the incoming shot automatically changes by the same duration.

To edit a Transition Rate:

1. Enable or disable the Lock Speed option.
2. Click on the Transition Rate you want to edit and enter the new value.

NOTE: The value of the Transition Rate cannot be greater than the Record duration of the incoming shot.

Editing Wipes

Wipes are represented in the EDL by two consecutive Edits with the same Event Number. The first Edit represents the outgoing shot of the wipe and the second Edit represents the incoming shot of the wipe. The wipe is listed in the Transition column of the second Edit as *W n m* (where *n* is the wipe code and *m* is the wipe duration).

The SMPTE wipe number appears in the Transition Entry of the second Edit in the wipe. The Start Location for the wipe is also indicated in the Transition Entry. A wipe can be Centred,

From Cut, or Up To Cut. The start location for the wipe appears beside the Transition Rate. The following illustration is a typical wipe in an EDL.

Outgoing shot		Transition Rate					
0023	002	V	C	09:08:29:09	09:08:29:15	10:00:18:05	10:00:18:05
0023	001	V	W001	6	01:10:21:25	01:10:22:15	10:00:18:05
Incoming shot		SMPTE Wipe Number					

If you add 500 to the SMPTE wipe number, the wipe is inverted during assembly. For example:



If you change the Transition Rate of a wipe, the Source Out of the incoming shot automatically changes by the same duration.

To edit a Transition Rate:

- 1. Enable or disable the Lock Speed option.
- 2. Click on the Transition Rate you want to edit and enter the new value.

NOTE: The value of the Transition Rate cannot be greater than the Record duration of the incoming shot.

Editing Timecodes

If you want to capture different material than what appears in the EDL, you can slip or trim the Edits to specify the correct material for your final assembly.

You can change any source or record timecode in the EDL. Use the options in the EDL menu to specify how changes you make to a single timecode affect other timecodes in the EDL.

To edit a timecode, click the source or record timecode Entry you want to modify and enter the new value. If the timecode is not already selected, you must click it twice to get the calculator.

You can also click the Entry and drag right to increase the value. Drag left to decrease the value. You can use hot keys to change the values in varying increments.

HINT:

- SHIFT**-drag = Jump by seconds
- CTRL**-drag = Jump by minutes
- ALT**-drag = Jump by hours

NOTE: Other Entries in the same Edit that are affected by your modifications become highlighted as you edit an Entry.

Slipping Events

If you want to change the material in an Event without affecting its duration, you can slip it using either the Slip or Slip Reel options.

Slip — Slip a source or record clip in a single Event. When this option is enabled, you change the in and out points of the selected clip without changing its duration.

For example, if you increase the Record In of an Event by five seconds, the Record Out of the same Event automatically increases by five seconds.

To slip a single Event:

1. Enable the Slip option.
2. Click the source or record timecode Entry you want to edit and enter the new value.

Slip Reel — Slip all source clips on a specified reel or all record clips in the EDL list. When this option is enabled, you change the in and out points of all source or record clips without changing their duration.

For example, if you increase the Record In of an Edit by five seconds, the Record In and Record Out values of all Edits in the EDL automatically increase by five seconds.

Modifying the timecode of a source Entry affects only other source Entries on the same reel. For example, if you increase the Source In of an Edit on reel 004 by five seconds, the Source In and Source Out values of all Edits on reel 004 automatically increase by five seconds.

To slip a reel:

1. Enable the Slip Reel option.
2. Click the source or record timecode Entry you want to edit and enter the new value:
 - If you edit a Source timecode, all Source timecodes on the same reel automatically slip by the same duration.
 - If you edit a record timecode, all record timecodes automatically slip by the same duration.

NOTE: By default, the Slip Reel option is left enabled after editing an Event. If you do not want the Slip Reel option to disable automatically, disable the Automatically Disable Slip Reel After Use option in the Preferences menu. See “Preferences For EDLs” on page 285.

Trimming Events

You can trim Events in the EDL. If you enable the Lock Speed option and modify a source or record Entry, the Event is trimmed. When you edit either the source or record timecode with the Lock Speed option enabled, **flame** automatically updates the corresponding source or record clip to match the value of the edit.

For example, if you increase the Source Out value of a non-timewarped Event by five seconds, the Record Out of the same Event automatically increases by five seconds. If you increase the Source In value of a non-timewarped Event by five seconds, the Record Out of the same Event automatically decreases by five seconds.

To trim an Event:

- 1. Enable the Lock Speed option.
- 2. Click the source or record timecode Entry in the Event you want to edit and enter the new value.

Editing and Creating Timewarps

You can edit an existing timewarp, or create a timewarp in your EDL. When you edit a timecode Entry of an Event with the Lock Speed option disabled, no other Entries are affected. This results in a speed change because the source duration differs from the record duration.

0018	003	V	C	14:06:14:13	14:06:14:17	50.0%	10:00:16:07	10:00:16:15
------	-----	---	---	-------------	-------------	-------	-------------	-------------

Speed value

For example, if you increase the Source In Entry of an Event by five seconds, no other Entries are affected. The record clip of the same Event is five seconds shorter than the source clip, and **flame** timewarps the result clip when assembled. See “Assembling an EDL” on page 292.

Timewarps are represented in the Speed column of the EDL. If the Speed Entry is blank, the Event is not timewarped. If there is a value in the Speed Entry, the Event is timewarped.

To edit or create a timewarp:

- 1. Disable the Lock Speed option.
- 2. Click the source or record timecode Entry in the Event you want to edit and enter the new value.

The value of the timewarp appears in the Speed Entry.

NOTE: If the Event is already a timewarp, editing any timecode of that Event changes its Speed value.

You can view the Speed value in either percentage or frames per second. Select the view mode in the EDL Preferences menu. See “Preferences For EDLs” on page 285.

Ripple Record

Use the Ripple Record option to ripple all Record In and Record Out Entries following an edited Record Out Entry. This is useful if you want to edit the Record Out of an Event without overwriting or creating a gap between the following shots.

When the Ripple Record option is enabled, any edit you make to an Event that changes the Record Out timecode causes all following Record In and Record Out values in the EDL list to automatically change by the same value of the edit.

For example, if you increase the Record Out of an Event by five seconds, all following Record In and Record Out timecodes automatically increase by five seconds.

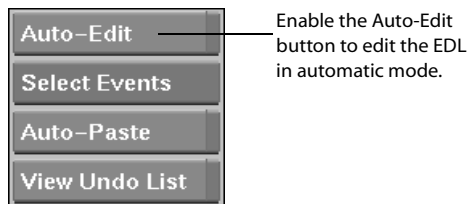
To ripple record Entries:

1. Enable the Ripple Record option.
2. Click the source or record timecode Entry in the Event you want to edit and enter the new value.

If your edit affects the Record Out Entry of the Event, all following record clips automatically ripple by the new value.

Auto Edit Mode

Use Auto Edit mode to make quick modifications to the EDL. You enable the columns in the EDL you want to edit and enable the Auto Edit button. When you modify an Entry in Auto Edit mode, the cursor automatically jumps to the next column you specified. To enable a column in the EDL, click the column's title at the very top of the EDL menu.



To use Auto-Edit mode:

1. Enable the Auto-Edit option.
2. Enable the columns you want to auto-edit. For example, to edit only Record timecodes, enable the Record In and Record Out columns.

Click a column's title to enable the column for Auto-Edit mode.

Event #	Reel	A.Patch	Tracks	Transition	Source In	Source Out	Speed	Record In	Record Out	△
TITLE: TEX30ASSEM										
0001	010		V	C	01:00:05:00	01:00:05:28		00:01:00:00	00:01:00:28	
0002	010		V	C	01:00:15:28	01:00:17:03		00:01:00:28	00:01:02:03	

3. Select the first Entry you want to edit.

4. Press Enter.

5. Edit the Entry.

6. Press Enter.

The cursor automatically moves to the next column. If no further Entries exist, a new Event is added automatically.

7. Repeat steps 4 to 6 for every Entry you want to edit.

Editing Events

Use the editing options to cut, copy and paste Events. You can also add or remove comments from Events.



Cutting, Copying, and Pasting Events

You can cut or copy single or multiple Events and paste them to a new location in any open EDL.

To cut, copy, and paste an Event:

1. Select the Event(s) you want to cut or copy. See “Selecting and Unselecting a Range of Events” on page 289.
2. Cut or copy the Event(s):
 - To cut the Event, click the Cut Events button.
 - To copy the Event, click the Copy Events button.
3. Select the paste options. See “Paste Options” on page 284.
4. Select the location where you want to paste the Event(s).
5. Click the Paste Events button.

Auto-Paste

Use the Auto-Paste option to quickly cut Events from multiple EDLs and automatically paste them to a single EDL.

To auto-paste:

1. Open the EDL to which you want to paste.
2. Enable the Auto-Paste option.
3. Cut or copy the Event(s).
4. Click Paste Events.

The Events are automatically pasted to the first EDL.

Paste Options

There are three options which affect how Events are pasted.

Enable:	To:
Paste: Fix Record	Change the Record In of the first pasted Event to start at Record Out of previous Event. All pasted Events are then rippled by the same amount.
Paste: Renumber Events	Automatically renumber all Events according to the Next Event value in the Special Functions menu. If this option is disabled, the pasted Events retain their original Event Numbers.
Ripple Record	Automatically ripple all Record Entries following the pasted Events.

There is also an option in the Special Functions menu for renumbering Events when they are created. To specify the Next Event value, open the Special Functions menu and enter the value in the Next Event field. This value is used when you paste an Event with the Paste: Renumber Events option enabled, or click the Insert Event button.

Adding and Removing Comments

You can add or remove comments from Events.

To add a comment to an Event:

1. Select the Event.
2. Click Add Comment.
3. Enter the comment.

NOTE: You can add multiple comments to an Event.

To edit a comment, click it and enter the new comment. To remove a comment, select the comment and click the Remove Comment button.

Adding and Removing Titles

You can add or remove a title from the EDL.

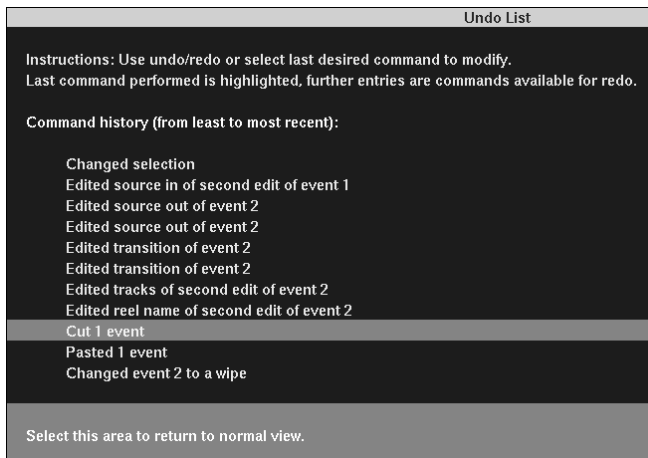
To add a title:

1. Click the Insert Title button.
2. Enter the new name.

To remove a title, click the Remove Title button.

Undo List

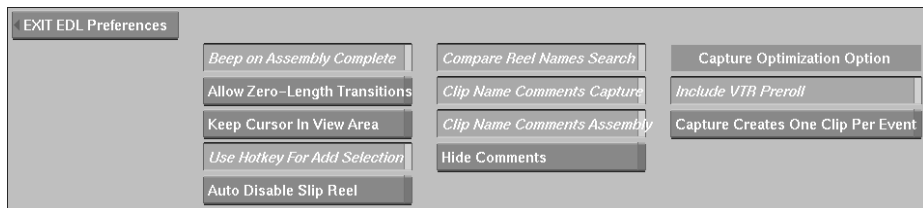
Use the Undo List to view all modifications (up to the number of undo levels you specified in the Preferences menu) you made in the current session and undo a single, or series of commands. Click the modification in the Undo List, and all operations are undone to that point.



Select any command in the Undo List. All commands after the highlighted command are undone. All commands prior to the highlighted command are redone.

Preferences For EDLs

Use the options in the Preferences menu to specify your EDL preferences.



Beep on Assembly Complete — Enable this option to hear an audible tone when the EDL has been assembled.

Allow Zero-Length Transitions — Enable this option to allow zero-length wipes and dissolves. If this option is disabled and you change a transition duration to zero, the transition automatically becomes a cut.

Keep cursor in view area — Enable this option to keep the cursor from leaving the window when you scroll an EDL. When this option is enabled and you scroll an EDL, the cursor will stop at the first or last Event on the EDL page.

Use hot key for add selection — Use this option to change the functionality of selecting Events. When this option is disabled, you add to your selection range by clicking the Events. When this option is enabled, you must hold the **CTRL** key while clicking Events to add them to your selection range.

Automatically disable Slip Reel after use — By default, the Slip Reel option remains enabled after you have slipped sources. Enable this option to automatically disable the Slip Reel option after you have slipped sources.

Compare reel names search — Disable this option to ignore reel names during assembly.

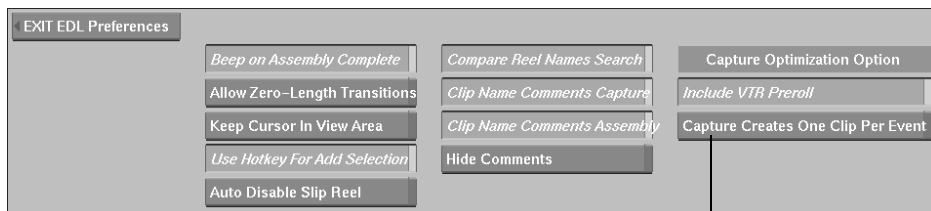
CLIP NAME comments capture — If your Events contain clip names in their comments (such as from an Avid), enable this option to use those names during capture. If this option is disabled, the Event Number is used as the captured shot's name.

CLIP NAME comments assembly — If your Events contain clip names in their comments (such as from an Avid), enable this option to use those names during assembly. If this option is disabled, the EDL title is used as the assembled clip's name.

Hide Comments — Enable this option to hide all Event comments.

Library Optimization

When you capture the EDL, you can choose to create a clip in the clip library for every Event in the EDL, or create a single clip in the library for each group of Events captured during same pass.



Enable this option to create a clip in the clip library for every Event in the EDL after auto-capture. When disabled, one clip is created for each group of Events captured during the same pass.

NOTE: You will be able to assemble the montage regardless of which option you chose to store EDL Events in the clip library.

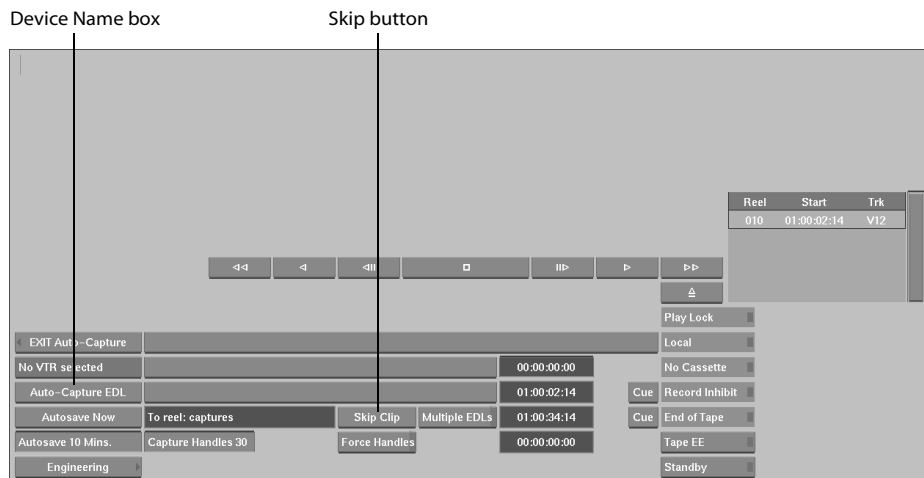
To reduce the size of clip libraries, you should disable this feature. However, if you know you will need to reference back to individual shots, they may be easier to find if you enable the option and generate a clip for every Event.

Auto-Capturing an EDL

Use the Auto-Capture command to automatically load clips specified in an EDL. You can also use the multicapture option to capture material from multiple EDLs. Click the Auto-Capture EDL button in the EDL menu to display the Auto-Capture menu.

Before a capture, **flame** searches for matching clips in the clip library reel specified in the Library Reel field. Only required clips are captured. If the Force Handles option is enabled, clips that do not contain at least as many handles as specified in the Capture Handles box are recaptured. See “Capturing Extra Frames” on page 290.

During auto-capture, the current position, and the in and out points are shown in the Timecode boxes.



Preparing to Auto-Capture an EDL

Before you auto-capture an EDL, you should make sure all of the preferences are set up correctly.

To prepare for auto-capture:

1. Load the EDL(s).
2. If you want to detect BRolls, enable the BRoll option. See “BRoll Detection” on page 289.
3. If you do not want to capture the entire EDL, select the Events you want captured. See “Selecting and Unselecting a Range of Events” on page 289.

4. Click the Auto-Capture EDL button in the EDL menu.
5. Select a VTR from the Device Name box.

NOTE: Make sure that the VTR is in remote mode; otherwise, **flame** cannot transfer clips. You can only switch the VTR from local to remote mode on the VTR panel itself. If the Local box in the menu is enabled, then the VTR is in local mode. See “Displaying the VTR Status” on page 291.

The frame at the current position of the selected device is displayed in the Viewer. You can use the controls below the Viewer to play all of the clips on the selected device. For information on playing clips, see “Playing the VTR” on page 237.

6. Set the value in the Capture Handles box. See “Capturing Extra Frames” on page 290.
7. Specify the VTR options in the Engineering menu.
8. Enable the tracks you want to capture.
9. If you want to capture all open EDLs which share the same library reel, enable the Multiple EDLs button.

Auto-Capturing an EDL

Once you set the auto-capture options, you can capture the EDL.

To auto-capture an EDL:

1. Click Capture in the EDL menu.
2. Click Auto-Capture EDL.

If it is not already loaded, **flame** prompts you to insert the appropriate cassette into the VTR.

3. Select the appropriate option for the capture.

Select:

Capture This Reel

Skip This Reel

Abort Capture

To:

Capture clips on the currently loaded cassette.

Skip the current cassette and proceed to the next reel.

Abort the auto-capture.

You can also select a different reel to capture in the Capture Order list.

Reel	Start	Trk
010	01:00:02:14	V12
011	02:00:02:14	V12

Click here to capture this reel first

4. Repeat the previous step for every cassette you are using for the auto-capture.

NOTE: The clips are always captured in C Mode, regardless of the display sort mode you selected (to minimize VTR transport, the clips are sorted according to their positions on the tape).

When the auto-capture is complete, **flame** saves the clips in the clip library on the reel specified in the Library Reel field. By default, all captured clips are named by their Event Numbers or clip name comments. Click Exit Auto-Capture.

5. Click Exit EDL.

The EDL appears in the clip library on a reel with the name of the EDL. If necessary, expand the reel to view the captured clips.

During capture, the names of the Events that use the material being captured are shown in the lower VTR Status box.

One clip is created for each Event that uses the captured material. Each clip is named according to the Event Number and given its own in and out marks set to the location used by the EDL at the time of capture. Tracks that are not used by the Event are not assigned an output channel (denoted by "--").

NOTE: For assembly, the clip with the name of the Event (Event Number) being assembled is considered first as a source for the edit. Another clip may be used as a source if it fits the criteria (source in/out, reel name).

After capture, all clips in the clip library reel are sorted according to the following criteria:

- The most recent clips with the same name as the EDL (usually assemblies).
- All other clips sorted by name. In case of identical clip names, clips are sorted by date and time.

BRoll Detection

If the EDL contains BRolls, you can choose to have **flame** detect them automatically. To enable the automatic BRoll detection option, click the BRoll detection button in the EDL menu. When the BRoll option is disabled, BRolls are indicated with a "B" following the reel number in the Reel Entry. When the BRoll detection option is enabled, BRolls appear as the original reel and you are not prompted for the reel containing the BRoll when you capture the clips.

Selecting and Unselecting Events

You can capture specific Events from an EDL. To select or unselect an Event, click on its Event Number Entry. The Event is highlighted when selected. Click the Select All button to select all Events in the EDL. Click the Unselect All button to unselect all Events.

Selecting and Unselecting a Range of Events

You can select a range of Events by holding **SHIFT** when you select an Event. To select a range, hold the **SHIFT** key and click the last Event in the range you want to select. You can also click and drag to select a range of Events.

To add to a range of Events, hold the **CTRL** key when you select an Event.

NOTE: Enable the Use Hotkey For Add Selection option in the Preferences menu to reverse the function of the Ctrl key when selecting a range of Events.

Selecting Uncaptured Events

Captured Events are indicated in the Event Number column by a + sign. To update the list, click the Update Capture Marks button.

Click the Select Uncaptured button to select all uncaptured Events in the EDL. When you click the Auto-Capture EDL button in the Auto-Capture menu, only the selected Events are captured.

HINT: To recapture an Event that is in the clip library, add a recapture mark or delete it from the clip library and recapture the EDL.

NOTE: If a clip for a selected Event already exists in the clip library, it is not recaptured.

Capturing Extra Frames

You can capture extra frames before and after the specified material. Enter the number of extra frames that you want to capture in the Capture Handles box.

NOTE: If specifying more handles causes material to overlap, or if less than 10 frames separate Events, the material is captured as a single clip.

Forcing More Handles to be Captured

You can force **flame** to recapture clips that do not have at least as many handles as defined in the Capture Handles box. By default, **flame** does not recapture clips if the specified material already exists on the reel as defined in the Source In and Source Out Entries of the EDL.

To ensure the clip contains at least the number of handles you want, enable the Force Handles button in the Auto-Capture menu. The next time you capture the EDL, **flame** recaptures clips that do not have at least as many handles as defined in the Capture Handles box.

Skipping Clips

During auto-capture, you can skip a clip. Skipping a clip aborts the current capture.

To skip a clip:

1. Click the Auto-Capture EDL button to start capturing the EDL.
2. When the clip you want to skip is being captured, click the Skip Clip button.
flame aborts the capture and marks the Event as “skipped.” The next time you click the Auto-Capture EDL button, **flame** starts the capture at the first Event following the skipped clip.
3. Repeat the previous step for every clip you want to skip.

NOTE: **flame** remembers skipped Events until you exit the Auto-Capture menu. To reset Auto Capture, click the Exit Auto-Capture button and re-enter the Auto-Capture menu. When you click the Auto-Capture EDL button, **flame** starts the capture at the first uncaptured clip.

Displaying the VTR Status

The VTR Status boxes show the current status of the VTR. The boxes are highlighted under certain conditions. Depending on the model of the VTR, you may be able to change the Tape EE and Standby boxes.

Control:	Description:
Local	Highlighted when the VTR is in local control—the VTR cannot be controlled from within flame . You can only switch the VTR between local and remote control on the VTR itself.
No Cassette	Highlighted (yellow) when the VTR does not contain a cassette. When VTR does contain a cassette, clicking this button ejects the cassette.
Record Inhibit	Highlighted when record is inhibited by either a control on the VTR or the record tabs on the VTR tape itself.
End of Tape	Highlighted when the VTR reaches the end of the tape.
Tape EE	When Tape EE is enabled, the VTR is bypassed by outputting VTR's input.
Standby	When Standby is on (highlighted green), the play heads on the VTR are disengaged. When Standby is off (highlighted yellow), play heads are engaged. Use Standby to lessen wear on the VTR's play heads.

Setting Up the Auto-Capture Session

Use the Engineering menu to set up various options for auto-capturing an EDL. To open the Engineering menu, click the Engineering button in the Auto-Capture menu.

NOTE: The default setting for the options in the Engineering menu are those that you chose when setting up your *.cfg* file. Making changes in the Engineering menu overrides your *.cfg* file settings for the current **flame** session only. The next time you start **flame**, the *.cfg* file settings take effect. For more information on setting up your *.cfg* file, see the *flame Installation Guide*.

For more information on using the Engineering menu, see “Setting Engineering Menu Options” on page 240.

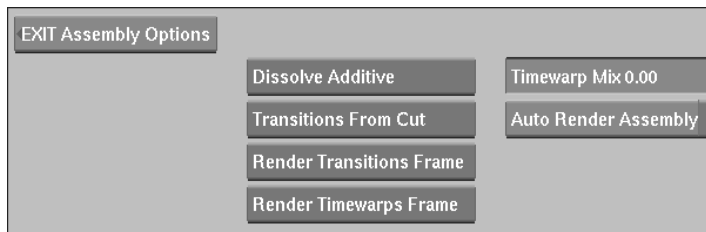
Assembling an EDL

Before assembling the EDL, make sure that you have auto-captured the clips that you want to use from the EDL. If you are assembling clips that have not been captured, **flame** replaces them with black frame segments.

Once you assemble the EDL, you can use the Player to view the final result.

Assembly Options

Before you assemble the EDL, specify the assembly options.



To assemble the EDL:

1. Specify the assembly options.

When the auto-assembly is complete, **flame** saves the assembled clip in the clip library on a reel with the name of the EDL. The assembled clip is always the first clip on the reel. By default, **flame** names the assembled clip with the EDL name. For example, an assembled EDL with the name “discreet,” has the following name when viewed in the clip library:
discreet

2. Click Exit EDL.

NOTE: If the EDL is created with the **flame** Export EDL feature, the focus alignment for each dissolve is specified in the EDL itself, overriding the default setting. The alignment is shown in the Transition column:

Symbol:	Indicates:
(F)	Transition’s focus is From Cut.
(C)	Transition’s focus is Centred.
(U)	Transition’s focus is Up to Cut.

Dissolve Curves

Specify the type of interpolation to use for all dissolves.

Select:	To:
Dissolve Curves as in Prefs	Use interpolation mode set in the main Preferences menu.
Dissolve Curves Linear	Create dissolves with linear interpolation curves.
Dissolve Curves S-Shaped	Create dissolves with S-Shaped interpolation curves.

Rendering Dissolves and Timewarps

You can specify rendering options in the Assembly Options menu for rendering dissolves and timewarps during EDL assembly.

Select:	To:
Dissolve Additive	Render dissolves as additive.
Diss. Non Additive	Render dissolves as non additive.
Diss. Inv. Non Additive	Render dissolves as inverse non additive.

Auto Rendering Assemblies

Enable the Auto Render Assembly button to render transitions and timewarps when you assemble the final clip.

Rendering Options

When assembling EDLs, you can change the rendering mode for transitions and timewarps.

Transition Render Options

Choose:	To:
Render Field	Override the Preferences settings and render transitions in Field mode.
Render Frames	Override the Preferences settings and render transitions in Frames mode.
Render Trans. as in Prefs.	Use the Preferences settings.

Timewarp Render Options

Choose:	To:
Render TW Frames	Override the Preferences menu setting and render timewarps in Frames mode.
Render TW Field/No Interp.	Override the Preferences menu setting and render timewarps in Field mode with no interpolation.

Choose:

Render TW Field/Half Interp.

Render TW Field/Full Interp.

Render TW as in Prefs.

To:

Override the Preferences menu setting and render timewarps in Field mode with half interpolation.

Override the Preferences menu setting and render timewarps in Field mode with full interpolation.

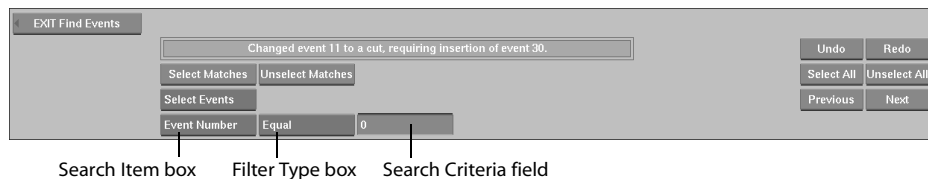
Use the Preferences settings.

Timewarp Mix

Specify the mix value for assembled timewarps in the Timewarp Mix field. You can enter a value from 0.00 to 1000.00.

Searching EDLs

Use the Find Events menu to search entries for text or timecode within an EDL.



Searching an EDL

When you search the EDL, you need to specify the item you want to search, the filter type you want to use, and the criteria for which you want to search.

To search an EDL:

1. Click Find Events.
2. Specify the item you want to search.

Select:

Event Number

Reel

Transition Length

Src In

Src Out

Rec In

Rec Out

Speed

To:

Search for Event Numbers.

Search for Reel Names.

Search for a specific Transition duration.

Search for a Source In timecode.

Search for a Source Out timecode.

Search for a Record In timecode.

Search for a Record Out timecode.

Search for a specific timewarp speed.

Select:	To:
Comment	Search for a comment or text within a comment.

3. Specify the filter type. The filter type will be different depending on the item you are searching.

For numerical searches:

Select:	To:
Is Equal to	Find all Event Numbers that match your search criteria.
Is Not Equal to	Find all Event Numbers that do not match your search criteria.
Is Less Than	Find all Event Numbers that are less than your search criteria.
Is Greater Than	Find all Event Numbers that are greater than your search criteria.

For alphabetical searches:

Select:	To:
Matches	Search for all reels that match your search criteria.
Does Not Match	Search for all reels that do not match your search criteria.
Contains	Search for all reels that contain your search criteria.
Does Not Contain	Search for all reels that do not contain your search criteria.
Less Than	Search for all reels lexicographically less than your search criteria.
Greater Than	Search for all reels greater than your search criteria.

4. Enter the search criteria in the Search Criteria field.
5. Select the option you want in the Select Type box.

Select:	To:
Select Events	Highlight all Events that match the search criteria.
Select Edits	Highlight all Edits that match the search criteria.

NOTE: For an explanation of Events and Edits, see “Events, Edits and Entries” on page 272.

6. To match results by case, enable Case Sensitive.
7. Find the item(s):
 - Click Next Match to move cursor to the next item that matches the search criteria.
 - Click Previous Match to move cursor to the previous item that matches the search criteria.
 - Click Select Matches to highlight all items that match the search criteria.
 - Click Unselect Matches to unhighlight all items that match the search criteria.

Generating an EDL

Use the EDL module to generate an EDL for a clip you select in the clip library. Once you generate an EDL, you can save it using one of the following methods:

- Save the EDL file to the EDL library.
- Export the EDL as a clip in a clip library. (See “Exporting an EDL” on page 299.)
- Export the EDL to tape. (See “Exporting an EDL” on page 299.)

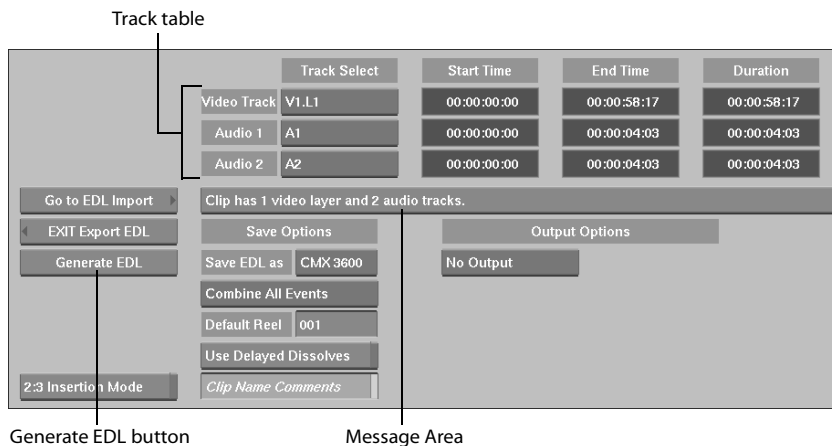
To generate an EDL:

1. Select the clip in the clip library.
2. Click the Export EDL button in the Clip Library menu.
The Export EDL menu appears.
3. Select the tracks to include in the generated EDL. See “Selecting Tracks for EDL Generation” on page 297.
4. Set the Save options. See “Setting Save Options” on page 297.
5. Set the Output Options. See “Setting Output Options” on page 299.
6. Click the Generate EDL button.

The EDL is generated.

The Export EDL menu

Use the Export EDL menu to save and output a generated EDL to a VTR or the clip library. To open the Export EDL menu, select a clip and click the Export EDL button in Clip Library menu.



In the Export EDL menu, the Track table shows the video track and audio tracks that the EDL will be based on. The table is divided into five columns:

Where:	Is:
Track Type	The track type. The first row is for the video track and the remaining rows are for audio tracks.
Track Select	The selected track's name and number. Use this display to select tracks that the EDL is based on. See "Selecting Tracks for EDL Generation" on page 297.
Start Time	The track's start timecode.
End Time	The track's end timecode.
Duration	The track's duration.

Selecting Tracks for EDL Generation

A **flame** clip may contain an unlimited number of video and audio tracks. EDL Export, however, interprets only one video track and two or four audio tracks, depending on the EDL format you want to generate. When a format only uses two audio tracks, the third and fourth audio tracks are disabled.

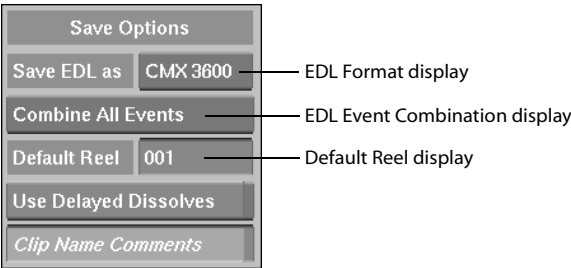
Select the video and audio tracks using the displays in the Select Track column. When None is selected, no EDL information is generated for the track. For example, you can generate an EDL based on the first audio track (A1).

Saving a Generated EDL

Before generating the EDL, make sure you have set the appropriate options in the Save Option displays.

Setting Save Options

Before generating an EDL, you can change the EDL format, the way matching Events are combined, and the default reel number:



EDL Event Combination

Use the EDL Event Combination display to select how Events that have the same source timecodes, record timecodes, and reel ID are combined when the EDL is generated.

Select:

Combine All Events
Combine Audio Events
Never Combine Events

To:

Use single Entry for all video and audio Events.
Use one Entry for audio Events and a separate Entry for video Events.
Use a separate Entry for each video and audio Event.

Default Reel

Use the Default Reel display to enter the reel ID for clips created in **flame**. Source clips are assigned reel IDs when loaded using the Input Clip or Import EDL menu.

For example, an Edit that uses a clip created with the Colour Corrector does not have a reel ID. When the EDL is generated, the clip is given the reel ID in the Default Reel display.

To generate and save an EDL:

1. Select the clip in the clip library.
2. Click the Export EDL button in the Clip Library menu.
The Export EDL menu appears.
3. Select the tracks to include in the generated EDL. See “Selecting Tracks for EDL Generation” on page 297.
4. Select the format for the EDL.

• CMX 340	• SONY 910
• CMX 3600	• SONY 5000
• CMX OMNI	• SONY 9000
• GVG 4	• SONY 9000 Plus (v2.21 or higher)
• GVG 4 Plus (GVG v4.1 or higher)	• SONY 9100
• SONY 900	
5. Select the EDL Event combination mode. See “EDL Event Combination” on page 298.
6. Specify the default reel for clips created within **flame**. See “Default Reel” on page 298.
7. Select No Output in the Output Type display.
8. Assign a reel ID for the output material. See “Output Reel” on page 300.
9. Specify the Start Timecode for the output material. See “Start Timecode” on page 301.
10. Click the Generate EDL button.
11. Click the Save Generated EDL button.

12. In the File Extension display, enter the file extension of the EDL you want to save.

NOTE: When you are saving EDLs to be used on an IBM-PC compatible computer, you must ensure the correct extension appears in the File Extension display. Since DOS only allows file names to use uppercase letters, EDL files must have the file extension ".EDL".

13. Type the new name for the EDL and press Enter, or click on an existing EDL in the EDL library.

If you select an existing EDL, you are prompted to confirm that you want to overwrite the existing EDL.

14. If necessary, click Confirm.

Exporting an EDL

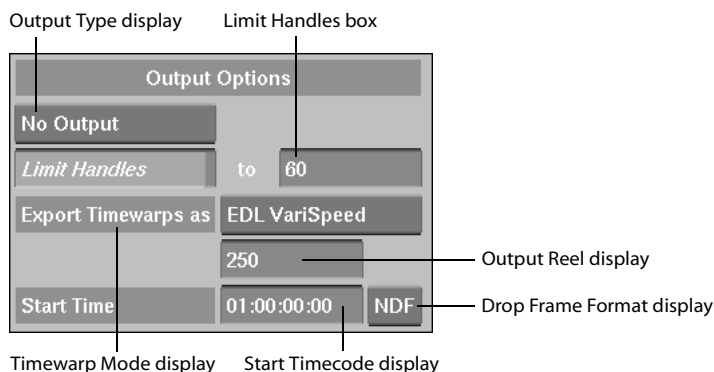
After you generate an EDL, you can use the Export EDL tool to export the EDL as a clip in the clip library or to tape, which can be used as a method of archiving your work.

Note that the Edit's original start timecode and end timecode are added to the Comment field along with the reel ID. This is useful if you want to trace the material reference in the generated EDL back to its location on the original source tape.

Before generating the EDL, make sure you have set the appropriate options in the Output Option displays.

Setting Output Options

Before generating an EDL, select the material that is output, the length of handles (if any), the method of exporting timewarps, the output reel ID, and the starting timecode.



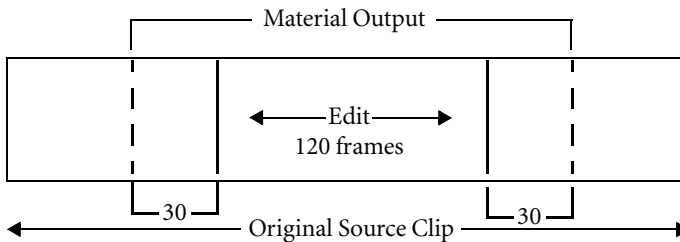
Output Types

Use the Output Type display to select the material to output to a VTR.

Select:	To:
Output All Material	Output all source clips and clips created within flame .
Output Generated Only	Outputs only the clips created within flame .
No Output	No clips are output. Only the EDL is generated.

Limit Handles

When Limit Handles is disabled, the entire source clip used by an Edit is output. When Limit Handles is enabled (highlighted yellow), the Edit is output with a specified maximum number of frames before and after the Edit. For example, the following figure shows what is output for a 120-frame Edit with Limit Handles set to 30 frames.



Exporting Timewarps

Use the Timewarp Mode display to select how timewarps are output:

Select:	To:
EDL Varispeed	Create a timewarp command in the EDL that plays the clip at the timewarp speed.
Rendered Source	Render the timewarp before outputting the clip to the VTR. This creates a new rendered clip and replaces the timewarp.



WARNING: There is no standard EDL format that recognizes timewarps created using **flame**'s Timewarp Editor. If you create a timewarp using a custom curve, you must use the Rendered Source option; otherwise, the timewarp curve is lost.

Output Reel

Use the Output Reel display to assign a reel ID to the output material. The Output reel ID is used in the EDL to reference the destination of the material.

Note that the Edit's original reel ID is added to the Comment field. This is useful if you want to trace an element back to its original source.

Start Timecode

Use the Start Timecode display to specify the start timecode for the output material. If the start timecode is 01:00:00:00, then the clip with all the material begins at 01:00:00:00. The start timecode is used in the EDL to reference the location of material.

Drop Frame Format

Use the Drop Frame Mode display to specify the drop frame mode for the output material. Select DF (drop frame) or NDF (non-drop frame) format (NTSC only).

To generate and export an EDL:

1. Select the clip in the clip library.
2. Click the Export EDL button in the Clip Library menu.
The Export EDL menu appears.
3. Select the tracks to include in the generated EDL. See “Selecting Tracks for EDL Generation” on page 297.
4. Use the Output Type display to select the material to output.
5. Specify the handles (if any) to include with the material. See “Limit Handles” on page 300.
6. Specify whether or not you want to render timewarps in the Timewarp Mode display. See “Exporting Timewarps” on page 300.
7. Assign a reel ID for the output material. See “Output Reel” on page 300.
8. Specify the Start Timecode for the output material. See “Start Timecode” on page 301.
9. Specify the Drop Frame format. See “Drop Frame Format” on page 301.
10. Click the Generate EDL button.
11. Save the EDL.
12. Export the EDL.

Click:

Output EDL Material

Save Output to Library

To:

Export the EDL material to the VTR.

Export the EDL material as a clip in the clip library.

13. To verify the generated EDL, click Go to EDL Import.

Importing EDLs through Your Network

Import the EDLs you want to use through your network and save them in the following directory:

```
/usr/discreet/flame_<version#>/edl
```

Where <version#> is the version of the **flame** release. Once you import the EDLs into the correct directory, you can load them into **flame**.

For complete information on importing EDLs, see Appendix A, “Networking a Macintosh or PC to Your SGI.”

Hot Keys

Use the following hot keys in the EDL module.

Editing Hot Keys

Press:	To:
CTR-CLICK EVENT	Add Event to selection.
SHIFT-CLICK EVENT	Add range to selection.
↑	Move cursor up.
↓	Move cursor down.
SHIFT + ↑	Move cursor up and select the Event.
SHIFT + ↓	Move cursor down and select the Event.
→	Move cursor right.
←	Move cursor left.
CTRL + ↑	Page up.
CTRL + ↓	Page down.
ALT + ↑	Go to top of list.
ALT + ↓	Go to bottom of list.
CTRL-CLICK SRCOUT	TOGGLE SRCDUR.
CTRL-CLICK SRCDUR	TOGGLE SRCOUT.
CTRL-CLICK RECOUT	TOGGLE RECDUR.
CTRL-CLICK RECDUR	TOGGLE RECOUT.
ENTER	Select or Edit an Entry.
NUMERIC PAD ENTER	Select or Edit an Entry.
CTRL-ENTER	Select or add to selection.

Press:	To:
SHIFT+D	Delete.
ALT+CTRL+F	Cycle font size.
-	Cycle current EDL.
ALT + -	Close current EDL.
ALT+CTRL + -	Close all EDLs.
ALT+CTRL+SHIFT+S	Cycle sort mode.

EDL Player Hot Keys

Press:	To:
F7	Set record as focus.
F8	Set source as focus.
MARK IN	Set mark in.
MARK OUT	Set mark out.
SPACE	Stop playing.
SHIFT + \	Make Record.
CTRL + →	Advance to Next Frame.
CTRL + ←	Advance to Previous Frame.

EDL Menu Hot Keys.

Press:	To:
ALT+R	Affect Reel.
ALT+S	Affect Selection.
CTRL+B	Ripple Record.
CTRL+N	Lock Speed.
9	Change EDL name.
[Cut Events.
O	Copy Events.
P	Paste Events.
U	Insert Events.
ALT+E	Select Events/Edits.

Press:	To:
ALT+M	Update Capture Marks.
ALT+CTRL+U	Select Uncaptured.
ALT+U	Unselect All.
ALT+A	Select All.
CTRL+9	Change Library Reel.
DEL	Cut.
DISSOLVE	Dissolve.
WIPE	Wipe.
SHIFT+F9	Auto-Capture.
.	Slip.
SHIFT+ .	Slip Reel.
ALT+CTRL+S	Sort.
ALT+C	Add Comment.
ALT+CTRL+C	Remove Comment.
ALT+F	Find Events.
ALT+T	Insert Title
ALT+CTRL+T	Remove Title.
V	Play.
S	Go to Out.
A	Go to In.
'	Exit Import EDL.
\	Assemble EDL.
0	Load.
SHIFT+0	Save.
BACKSPACE	Undo.
SHIFT+BACKSPACE	Redo.

*This chapter explains how to work with 24p material in **flame**.*

Summary

In this chapter, you learn about:

- “Creating a 24p Project” on page 306
- “Clip Input and Output for 24p Projects” on page 306
- “Using EDLs with 24p Projects” on page 310
- “Playing 24 fps Clips” on page 317
- “PAL Mastering” on page 319

NOTE: Before performing the procedures in this chapter, you should be familiar with the basic image capture and output procedures described in Chapter 13, “Clip Input and Output.”

About 24p Mastering

flame includes tools for performing 24p mastering on film-based material in 60i or HD resolution. Tools are provided for:

- Real-time 2:3 removal of pulldown on input and insertion on output.
- EDL capture with 2:3 removal of pulldown.

Advantages of Working in 24p

flame provides several tools to make it easier to work in 24p using film-based 30 fps material. When you use **flame** to assemble a project in 24p, you can master other formats from the project, such as NTSC or HD. Working with clips at 24 fps offers several other advantages:

- You save approximately 20% in rendering time and disk space.
- You work with the actual frames all the time (no more need to perform a temporary 2:3 removal of pulldown when retouching, tracking, or working in Action — a time-consuming process).

The following table summarizes a typical workflow for creating 24p masters from film-based material in **flame**:

To:	Refer to:
Create a 24p project	"Creating a 24p Project" on page 306.
Capture a single clip with 2:3 removal	"Clip Input and Output for 24p Projects" on page 306.
Capture film-based material using a 30 fps EDL and apply 2:3 removal	"Capturing with an EDL" on page 310.
Play back 24p clips	"Playing 24 fps Clips" on page 317.
Output 24 fps material with 2:3 insertion	"Clip Input and Output for 24p Projects" on page 306.

NOTE: Features in this chapter are only described in the context of 24p mastering. For complete information on each feature, refer to the appropriate chapter in this user's guide.

Creating a 24p Project

When you want to perform 24p mastering in **flame**, create a project using the following 24p template:

Project Template	Description
<i>24p_ntsc.cfg</i>	For NTSC standard.

The template sets the framerate of the project to 29.97 fps. You must work in this framerate when creating a 24p master, since the framerate in **flame** must be the same as that used by the VTR for clip input and output. The template also sets the default timecode format used by the project to 24 fps. This allows you to work at 24 fps, and output the material at 24 or 30 fps.

NOTE: The only difference between the *24p_ntsc.cfg* template and the regular NTSC template is the default Timecode setting.

Clip Input and Output for 24p Projects

Using the Input Clip menu, you can capture 30 fps film-based material and convert it to 24 fps material in real time as you capture. When the project is complete, you can output the 24 fps material at 30 fps with 2:3 insertion in real time from the Output Clip menu.

NOTE: You cannot perform 2:3 removal and insertion when using the Sony HDCAM codec option. For a workaround procedure, see "Clip I/O with the Sony HDCAM Codec" on page 256.

24p Engineering menu options to specify 2:3 removal or insertion, as well as the location of an A-frame. To correctly remove or add the pulldown, **flame** needs to know where on the tape an A-frame is located. Usually the first frame of the clip on the tape is an A-frame.

24p Mastering Engineering Menu Options

Several options in the Engineering menu must be set for input and output of clips for use in 24p mastering projects.

2:3 Removal button — Enable this button to remove 2:3 pulldown that was applied to film-based video material prior to capture. This converts clips from 30 fps to 24 fps in real time during input.

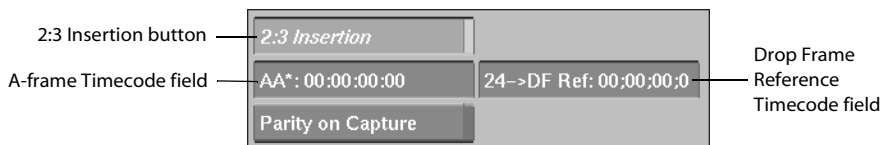
2:3 Removal button



The 2:3 Removal button appears in the Engineering menu when it is accessed from the Input Clip menu.

NOTE: When using 2:3 Removal, you must set the Field Dominance (in the Engineering menu) to F1.

2:3 Insertion button — Enable this button to insert 2:3 pulldown when outputting a 24p project to 30 fps video. This converts clips from 24 fps to 30 fps in real time during output.



The 2:3 Insertion button appears in the Engineering menu when it is accessed from the Output Clip menu.

NOTE: When using 2:3 insertion, you must set the Field Dominance (in the Engineering menu) to F1.

A-Frame Timecode field — This field appears when the 2:3 Removal button is enabled for clip input or the 2:3 Insertion button is enabled for output.

Use this field to indicate where on the tape an A-frame is located. The default is set to 00:00:00:00. If the first A-frame on the tape does not occur at timecode 00:00:00:00, specify the timecode of any A-frame on the tape. **flame** uses this as the reference point when determining the correct sequence for 2:3 removal or insertion.

HINT: If you do not know the timecode for an A-frame, simply capture a few seconds of material without removing the pulldown and examine several frames in the Player. The second frame after two jitter frames in a row (that is, frames where fields do not match) is an A-frame.

Drop Frame Reference Timecode field — Use this field when performing 2:3 removal on input of material with drop frame timecode, and when performing 2:3 insertion on output to drop frame tape.

Inputting Drop Frame Material

When you are performing 2:3 removal on input of drop frame material, the Drop Frame Reference Timecode field allows you to set a reference timecode for **flame** to use when calculating the conversion between 30 fps drop-frame timecode and 24 fps timecode. The timecode you set in this field controls the start point of the timecode conversion. To get the best conformity between the drop frame timecode and the 24 fps timecode, set the reference to the reel start timecode.

When material with drop frame timecode is converted to 24 fps timecode, the resulting material is missing a small number of frames.

When the reference is 00:00:00:00, the 24 fps timecode does not correspond well with its drop frame equivalent, as shown in the following table.

Reference of 00:00:00:00

Drop Frame Timecode	Resulting 24 fps Timecode
00:10:00:00	00:09:59+06
10:00:00:00	09:59:24+00

When the reference timecode is the same as the reel start timecode, the correspondence is much closer. This is shown in the following example, where the reel start timecode is 10:00:00:00.

Reference of 10:00:00:00

Drop Frame Timecode	Resulting 24 fps Timecode
10:00:00:00	10:00:00+00
10:00:59:29	10:00:59+23
10:01:00:02	10:01:00+00
10:01:59:29	10:01:59+21
10:02:00:02	10:01:59+22

Outputting to Drop Frame Tape

If you output a 24 fps clip to a drop frame tape, and you want to be sure to output to the same spot on the tape, use the same reference timecode that was used when you input the clip. The Drop Frame Reference Timecode field is used to calculate the correct in point on the drop frame tape. Therefore, you will notice that when you set the reference timecode in this field, the in point timecode field is updated accordingly.

To input a clip with 2:3 Removal:

1. From the Library menu, open the Input Clip menu.
2. Open the Engineering menu and enable the 2:3 Removal button.
3. Set an A-frame timecode.
4. If the material is drop frame, set the Drop Frame Reference Timecode.
5. Set the Field Dominance to F1.
6. Return to the Input Clip menu and set the in- and out-timecodes.

	Out-Timecode	In-Timecode	In and Out 2:3 Sequence frames			
		00:00:00:00				
Start on Timecode		00:00:00:00	A	in	-	+ Cue
Stop on Timecode		00:00:00:00	A	Out	-	+ Cue
		00:00:00:00				Log for Batch Capture
	0	Increment Name				Auto-log on Out

Capture can be done starting and ending on any frame. The In and Out 2:3 Sequence frames indicate whether the in or out point originates as a jitter frame. The 2:3 Sequence icons appear when 2:3 Removal mode is enabled in the Engineering menu. The in and out points are established as follows:

- If the in point falls on a B/C frame, C is the first frame of the 24 fps clip.
- If the in point falls on a C/D frame, D is the first frame of the clip.
- If the out point falls on a B/C frame, B is the last frame of the clip.
- If the out point falls on a C/D frame, C is the last frame of the clip.

7. Process the clip.

To output a clip with 2:3 Insertion:

1. Select the 24p clip and open the Output Clip menu.
2. Open the Engineering menu and enable the 2:3 Insertion button.
3. Set the timecode of an A-frame on the tape.

4. If the tape is drop-frame, set the Drop Frame Reference Timecode.
5. Return to the Output Clip menu and set the Start and End timecodes.



The In and Out 2:3 Sequence frames indicate which type of inserted frame falls at the specified in- and out-timecode. The 2:3 Sequence icons appear when 2:3 Insertion mode is enabled in the Engineering menu. When outputting clips:

- The in point can occur at any 2:3 sequence frame except D/D.
- The out point can occur at any 2:3 sequence frame except B/C.

This ensures the integrity of the output material on the tape.

6. Process the clip.

Using EDLs with 24p Projects

In addition to capturing individual 30 fps clips with 2:3 removal using the Input Clip menu, you can use the EDL module to capture an EDL with 2:3 removal.

Capturing with an EDL

The EDL module has several features to make it easy to capture film-based material for use on a 24p mastering project. The film-based material you are capturing must have already been converted to 30 fps prior to capture. You can import a 30 fps EDL and capture the associated material while removing the pulldown in real time to get 24 fps clips. Use the 2:3 Removal Mode option in the EDL module to achieve this.

When capturing, you need to specify the type of pulldown frame of a reference frame. This is determined by the starting frame that was used when the material was originally converted to 30 fps. Usually, the reference frame is an AA-frame or a BB-frame, since these respectively correspond to a 2:3 (starting on an A-frame) or a 3:2 pulldown (starting on a B-frame). By default, **flame** assumes an AA-frame, but you can specify any frame type within the 30 fps pulldown sequence.

To use 2:3 Removal Mode, you should use EDLs that were generated from an edit session that was done at 24 fps and then translated to a 30 fps EDL with 2:3 pulldown added. For more information, see “Using 2:3 Removal Mode” on page 311.

On each tape, the 2:3 sequence must be consistent throughout the entire tape. If the tape does not have a consistent 2:3 sequence, see “Working with Tapes with Inconsistent Pulldown” on page 317.

Using 30 fps Original Material — If you have some original 30 fps material that you want to use in the 24p project, you can capture it from the EDL module along with the film-based material. Using the Edit Configuration window (described in the next section), you identify which tapes contain video-based material. Instead of applying a pulldown removal to this material, **flame** applies a 125% timewarp to it so that it may be played at 24 fps. You will end up with both the original clip and the timewarped clip. You also have the option of timewarping and hard-committing the clip manually. In this case, you must save the timewarped clip to the same library reel as the captured events from your EDL.

Using 2:3 Removal Mode

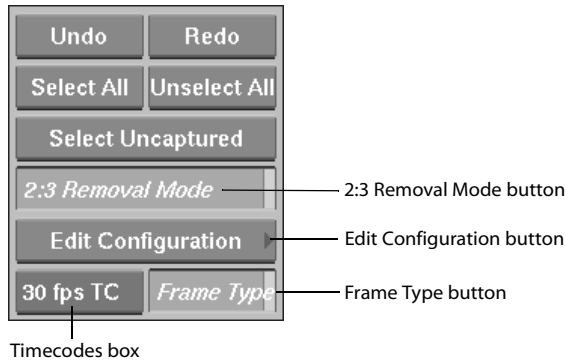
The 2:3 Removal Mode is used for 24p mastering with EDLs. When enabled, the 2:3 sequence is automatically removed from all clips in the EDL that you specify during Auto-Capture. For example, if some tapes contain film-based material with the 2:3 sequence and other tapes contain video material, you can customize a 2:3 configuration for your EDL so that the 2:3 sequence is only removed for those tapes that contain film-based material. Use this mode when:

- Your source material is 30 fps with 2:3 pulldown.
- The offline editing was done at 24 fps (not 30 fps).
- The EDLs describe the 30 fps film-based source material with 2:3 pulldown.
- The record timecodes were generated from the timecodes of the 24 fps master that was converted to 30 fps.

NOTE: If your project does not meet the above criteria, the 2:3 Removal Mode may result in video or audio gaps. For example, if the edit was performed with 30 fps sources, wherever the first frame of a splice is a jitter frame, the resulting clip will have a video and audio gap of one frame (a black frame) after capture. If this occurs, you must remove the gaps and correct the sync manually. For this reason, it is recommended that the original EDL be edited offline at 24 fps.

To capture material for 24p mastering:

1. Open the EDL module and close any EDLs that are open.
2. Make sure that the 2:3 Removal Mode button is enabled. It is enabled by default when a 24p template is selected at project creation.



3. Load the EDL(s). For details on options when opening multiple EDLs, see Chapter 14, “EDLs.”

NOTE: Once the EDL is loaded, you can use the Timecodes box to view timecodes at either 24 or 30 fps. You can change this anytime — it does not affect the EDL.

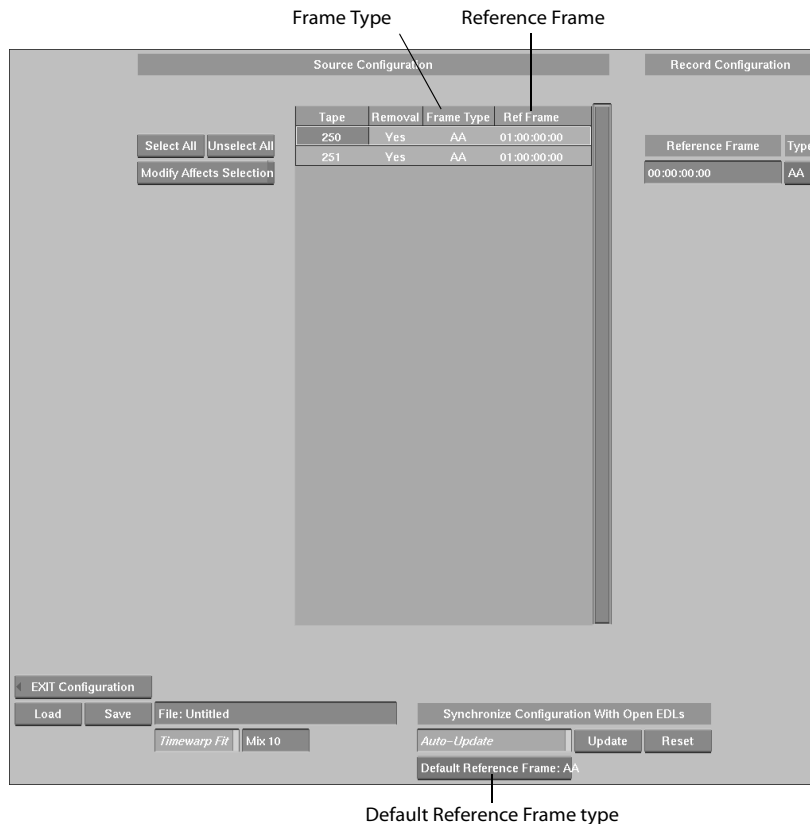
4. Modify the EDL if necessary. You can display or hide the display of the frame type in the EDL using the Frame Type button.

NOTE: To modify the EDL, 30 fps timecodes must be displayed.

5. Click the Edit Configuration button.

The 2:3 Configuration menu appears with a window listing all tapes from open EDLs. In this window, you can specify a different 2:3 sequence frame for the reference frame of any tape,

and also highlight any tapes that you do not want to undergo 2:3 removal (for tapes containing video-based material).



6. Use the Removal column to specify which tapes you want to undergo 2:3 removal. By default all tapes are set to get removal (indicated by “Yes”). To change this setting for a particular tape, place the cursor over the field and drag to the left.

The entry is highlighted and the field changes to “No” (for no removal).

7. The Frame Type column indicates the 2:3 sequence frame for the current reference frame. If the reference frame for any tape is not an A-frame, change the value in the Frame Type column by dragging the cursor over the current value to the right.

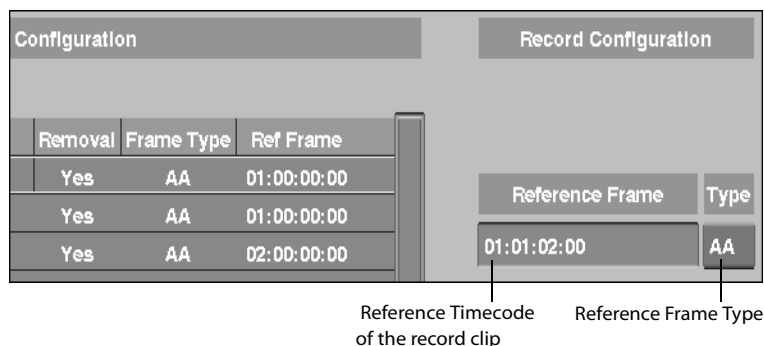
Change the value to BB, BC, CD or DD as needed.

NOTE: You can change the default reference frame type in the Default Reference Frame box.

8. The Ref Frame column shows the timecode of the reference frame on the tape. **flame** automatically sets this timecode by taking the lowest event Source In timecode for each tape

and then rounding down to the nearest hour. To manually change the Reference In timecode, click the entry you want to change and enter the new timecode.

9. If you need to change the Record Reference Frame type or timecode, you can change the values in the Record Configuration area. For example, if you want the first frame of the record clip to be 01:00:00:00 and a B frame, set the record Reference Frame timecode to 01:00:00:00 and the Reference Frame Type to BB.



10. Tapes marked with “No” in the Removal column will not receive pulldown removal. To apply a timewarp to the clips on these tapes, enable the Timewarp Fit button and set a mix value for the timewarp. The same mix value is applied to all the tapes getting a timewarp.

NOTE: You can also apply a timewarp manually after capture. In this case, do not enable the Timewarp Fit button.

11. If you will need to use the 2:3 configuration another time (for example, to re-capture the material), you can save it using the Save button. Click Save, enter a name, then click the Save Configuration button. The file is given the extension .r23 and placed in the default EDL directory. You can change the extension if needed.

NOTE: By default, the current 2:3 configuration is automatically updated as you open an EDL. You can temporarily disable this feature. For more information, see “Updating 2:3 Configurations to Open EDLs” on page 315.

12. Exit the 2:3 Configuration menu.

Any events excluded from 2:3 removal are highlighted.

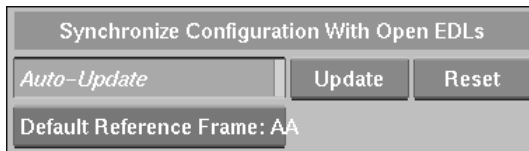
13. Open the Auto-Capture EDL menu and capture the material as described in “Auto-Capturing an EDL” on page 287.

NOTE: When capturing material with 2:3 Removal Mode, only one set of Capture Optimization options can be used. The option, found in the EDL Preferences menu, is greyed out when 2:3 Removal Mode is enabled, and set to have no VTR Preroll and to create one clip per event. For a description of this option, see “Library Optimization” on page 286.

Removal of 2:3 pulldown and timewarps, if requested, are performed as the material is captured. If an event starts at a non A-frame (for example, B or C), the number of handles of the event for capture is modified so that the capture starts at an A-frame. This results in a maximum of three extra frames per event. These frames are added to the number of specified capture handles and do not affect the in and out points of the event.

Updating 2:3 Configurations to Open EDLs

There are several update options for the 2:3 configuration. You may want the configuration to update every time you open an EDL, for example, if you are going to capture or assemble material from every open EDL. However, you may want to open an EDL for viewing purposes only and not affect your current 2:3 configuration. The following update options are available in the 2:3 Configuration menu.



Select:	To:
Auto-Update	Automatically update the configuration every time you open one or more new EDLs. The configuration is updated with any new information from the EDLs, including source and record reference frame. If you want to open an EDL without affecting the current 2:3 configuration, disable the Auto-Update button and then open the EDLs. The configuration is not affected. By default, this option is enabled.
Update	Update the configuration to reflect all open EDLs, even if Auto-Update is disabled.
Reset	Reset the configuration to the default values. NOTE: When you exit the EDL module, the current reference frame settings (AA or BB and timecode) are stored in the preferences as "default". When you click Reset, your latest preferences for these settings are restored.

To temporarily disable the 2:3 configuration update:

1. In the Edit Configuration menu, disable the Auto-Update button.
2. Click the Exit Configuration button.
3. Open any EDLs you want to view.
The 2:3 configuration is not affected by the EDLs you open.
4. You can now close the EDLs, or apply the 2:3 configuration to these EDLs. For more information, see the next section.

To update the 2:3 configuration with all open EDLs:

In the Edit Configuration menu, click the Update button.

The current 2:3 configuration is updated to reflect all open EDLs.

NOTE: The configuration is updated even if the Auto-Update button is disabled.

Updating vs. Resetting the Current Configuration

When you update the configuration, either by having Auto-Update enabled, or by clicking the Update button, information from all open EDLs is added to the current 2:3 configuration.

When you reset a configuration, default values are reapplied to the current configuration. Some values in the configuration are affected differently for update and reset.

Item:	Update:	Reset:
Tape	The tape reel names of all newly opened EDLs are added to the configuration.	The tape reel names of all newly opened EDLs are added to the configuration.
Removal	Existing entries are not affected. Newly opened EDLs added to the configuration are set to Yes by default.	All entries are reset to Yes.
Frame Type	Existing entries are not affected. Newly opened EDLs added to the configuration are set to the default value.*	All entries are reset to the default.*
Ref Frame (Source)	If the lowest Source In timecode is lower than the current value, the current reference frame is automatically set to the hour before that Source In timecode.	The current reference frame is automatically set to the hour before the lowest Source In timecode.
Ref Frame (Record)	If the lowest Record In timecode is lower than the current value, the current reference frame is automatically set to the hour before that Record In timecode.	The current reference frame is automatically set to the hour before the lowest Record In timecode.

* The default value for the Frame Type is reset every time you exit the EDL module. The value set in the Default Reference Frame box is prepared for the next session.

Working with Tapes with Inconsistent Pulldown

If the tape has inconsistent pulldown, you need to manually remove the pulldown before you can assemble the EDL. If the first portion of the tape has a Source Reference Frame of AA and the second portion of the tape has a Source Reference Frame of BB, then you can simply treat that tape as two tapes. Rename the reel for all events on the second portion of the tape and apply the configuration as normal. However, if the pulldown is inconsistent throughout the entire tape, you need to perform the following procedure.

If your tape has inconsistent pulldown:

1. Capture the EDL at 30 fps.
2. Use the Film Compress option in the Format menu to remove the pulldown for each clip as described in “Compressing Clips from 30 to 24 fps” on page 618.
3. Save the new clips to a reel in the clip library and name the reel.
4. Reassign the EDL reel name for every clip for which you have removed the pulldown, as described in “Renaming a Clip’s Tape in List View” on page 210.
5. Go back to the EDL and point the EDL to the new clip library reel.



Playing 24 fps Clips

The Player in **flame** does not currently support real-time 2:3 insertion, which would enable viewing your 24 fps clips at the 29.97 project framerate. However, there are several different methods you can use to view your 24 fps clips:

- Use 2:3 insertion in the Output Clip menu to view the clip (without outputting it).
- Use the Film Expand command in the Format menu to apply 2:3 insertion to individual clips.
- Temporarily change the monitor refresh rate and project framerate to settings that are suitable for playing 24 fps clips.

Each of these methods is described in detail next. Read through the entire procedure before actually performing it.

To play a 24 fps clip in the Output Clip menu:

1. From the Library menu, click Output Clip.
2. Select the clip.
3. In the Device Name box, select No VTR Selected.
4. Open the Engineering menu and enable the 2:3 Insertion button. Use the default setting for the A-frame reference timecode.
5. Go back to the Output Clip menu and click the Process button.

The clip plays in the image window with 2:3 insertion applied to it.

To apply 2:3 Insertion using the Film Expand command:

1. Follow the instructions for 2:3 insertion in the section “Expanding Clips from 24 to 30 fps” on page 621.
2. Play the clip.
3. Discard the clip.



WARNING: Do not apply effects to this clip or output it. It is important not to use this clip in your 24p master project, as 2:3 insertion will be applied during clip output.

To temporarily change the project framerate and monitor refresh rate:

1. Exit **flame**.
2. Go to the project directory:
`/usr/discreet/project/effects/<projectname>/cfg`
 where `<projectname>` is the name of the project you created.
3. Use the UNIX command *jot* to open the project configuration file.
4. Change the Hires token from 30Hzf to 72Hz, and the Framerate token from 29.97 to 24 fps. See “Project Configuration Files” in the “Configuration Files” chapter of the *flame Installation Guide* for details.

NOTE: You can also change the framerate within **flame**, using the Playrate field found in the Setup menu of the Player. Changing the framerate alone (without changing the refresh rate) will not provide accurate playback.

5. Save the file and reopen **flame**.
6. Play your 24 fps clips.
7. Before outputting your project, go back to the project configuration file and return the Hires and Framerate tokens to their original settings.



WARNING: You cannot output 24 fps clips with 2:3 insertion to a VTR when using the project configuration file settings described above for playing clips.

Archiving 24p Projects

You can archive 24p projects to either 24 fps or 30 fps VTRs. When you archive a 24p project to a 30 fps VTR, clips are archived at the original speed (24 fps) — no pulldown is added.

Therefore, when you see the VTR archive video segment dump, playback appears sped up (since you are playing 24 fps clips at 30 fps). The advantage is that the required archive tape space is reduced by about 20%.

PAL Mastering

You can use your 24p master to create a PAL master:

1. Create your 24p master as described in this document.
2. Copy the material to a PAL partition.
3. Make sure Word Sync is selected as the Input Source in Audio Preferences.
4. Change the sample rate on the word sync device to 50kHz.

The Audio Rate should appear as 50000 Hz in the Audio Preferences menu.

This performs a slight timewarp on the audio to match the video at 25 fps.

NOTE: You can only use analog audio for PAL mastering.

[illegible]

16

Image Import and Export

Picture this!

*You can import images into **flame** and export images to the file system using a number of different file formats. This chapter guides you through these procedures, and also demonstrates how to use Lookup Tables (LUTs) and create LUTs.*

Summary

In this chapter, you learn about:

- “File Formats and File Extensions” on page 322
- “Importing Images” on page 322
- “Using a Lookup Table” on page 330
- “Lookup Table Editor” on page 332
- “Exporting Images” on page 338
- “Removing Images” on page 343

About Image Import and Export

Clips can be stored as a sequence of image files in a directory in the UNIX file system. Use the Import Image command to import images from the file system to the desktop reels, and the Export Image command to export a clip or a single frame from the desktop reels to the file system. When you export a clip, each frame of the clip is stored in a separate image file in the selected directory.

NOTE: The default directory for imported and exported files is set in the Environment Directory Pathnames section of the project configuration file. For more information, refer to the **flame** *Installation Guide*.

You can use a Lookup Table (LUT) when importing or exporting a clip, and create a custom LUT for Cineon or Dpx (Spirit) images using the LUT Editor.

File Formats and File Extensions

You can export and import image files using a number of different file formats. The following file formats are supported for both import and export:

File Format	Default File Extension	Bit Depth (for Export)
Alias	als	8 bits
Cineon	(no extension)	10 bits
Dpx (Spirit)	(no extension)	8 or 10 bits
Jpeg	jpg	8 bits
Pict (Macintosh)	pict	8 bits
Pixar	picio	8 bits
Sgi	sgi	8 or 16 bits
Softimage	pic	8 bits
Targa	tga	8 bits
Tdi/Maya	iff	8 or 16 bits
Tiff	tif	8 or 16 bits
Wavefront	rla	8 or 16 bits

As shown in the table, **flame** provides a default file extension for each file format. You can use the default extensions as you are importing or exporting images, or change the extension temporarily as needed. You can also edit the default extensions in the *init.cfg* configuration file.

Importing Images

You can import images created in other applications such as a 3D rendering program, as well as images that have been previously exported using the Export Image menu. Images may originate from other platforms (for example, a Macintosh or PC).

When importing a file, you can specify a LUT to remap specific pixel values. This will most likely be needed for Cineon or Dpx files, unless you are using a specific display LUT. For more information, see “Using a Lookup Table” on page 330.

Image Sequences

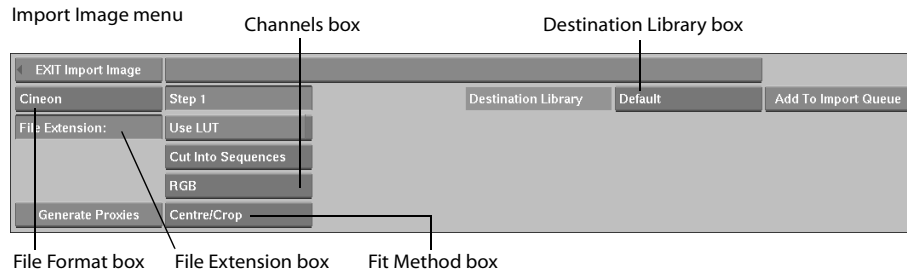
When clips with multiple frames are exported from **flame**, the frames are saved as separate image files, each of which is assigned a sequential number. In the file browser, these image files appear under a parent entry representing the entire sequence of images. Similarly, if another application created a series of images with sequential numbering appended, these images will appear under a parent entry in the file browser.

You can import all the images in a sequence or only a selection of the images.

To import an image:

1. Click the Import Image button in the Library menu.
2. Select the destination reel.

The Import Image menu and file browser appear.



3. By default, the file browser points to the images directory for the project. (This is set in the Image Token section of the project configuration file.)

If the files to import are located elsewhere, change the current directory (see “The File Browser” on page 61 for details).

NOTE: If your computer has a floppy drive, you can import images directly from a diskette.

4. Select a format from the File Format box.

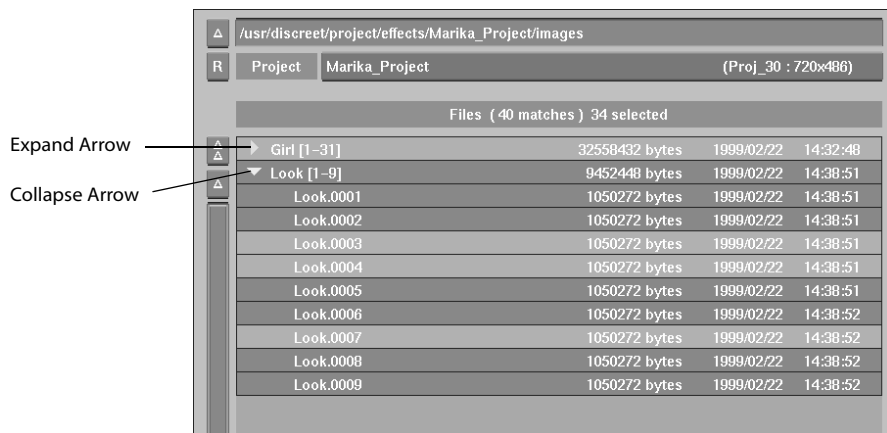
All available images of the selected file format appear in the file browser.

NOTE: If the file(s) you want to import do not use the default extension, they do not appear in the browser. In this case, click the File Extension box; the on-screen keyboard appears. Enter the extension of the file(s). They will now appear in the browser. To view all files in the current directory, enter a blank extension.

5. Set the following options: Force 12 bits (Sgi format only), Step Factor, Load Method, Channels, Fit Method, Use DPX Timecode (Spirit DPX format only) and Proxy Generation. See the section “Import Image Options” on page 327 for descriptions of these options.
6. Enable the Use LUT button if you want to apply a Lookup Table to the image files. See “Loading a Predefined LUT” on page 332 for details.

NOTE: If you choose Cineon or Dpx (Spirit) as the file format, you can use the LUT Editor. For more information, see “Lookup Table Editor” on page 332.

7. Select the image(s) to import.



An entry that has an expand or collapse arrow is a parent entry. When a parent entry is collapsed, the number of images in the sequence is shown in brackets. When a parent entry is expanded, you see the images indented below it. To expand or collapse a parent entry, click the arrow.

- To select all of the images in a sequence, click the parent entry. It changes to light grey.
- To select a range, **SHIFT**-click the first and last entries in the desired range. Alternatively, simply drag the cursor over the entries.
- To select or deselect several entries, **CTRL**-click them.

For more details, see “The File Browser” on page 61.

8. Select the destination library in the Destination Library box.
9. Click the Load button.

Background Import of Images

You can now import one or more image files in the background while you perform other tasks. To import several files, you select them and place them in a queue, then process them as a batch. You can assign different clip options (such as Fit Method or LUT file) to each file that you add to the queue.

When importing files as a background process, you import them to a designated clip library rather than to the desktop.

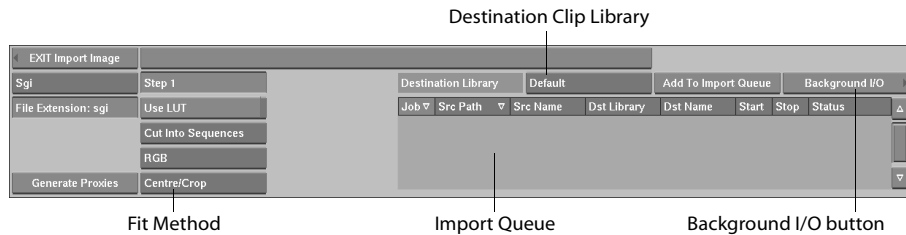
Use the Import Image menu to import images in the background.

HINT: To simultaneously import and export files in the background, simply start one of the processes, then go into the other menu (either the Import Image menu or the Export Image menu) and start that process.

To import images as a background process:

1. Click the Import Image button in the Library menu.

The Import Image menu appears.



2. In the Destination Clip Library box, select the clip library to which you want to import the clips.

NOTE: You can also create a new clip library by selecting <New>.

3. In the file browser, select the first file you want to import.
4. Set options such as Fit Method, channels to import, and so on. See “Import Image Options” on page 327 for details.
5. Click the Add To Import Queue button.

The clip appears in the Import Queue.

6. Repeat steps 3 to 5 for any other clips that you want to import.
7. Click the Background I/O button or press the **F11** hot key.

The Background Status menu appears.



Here you can check to see that your selections are in the queue.

NOTE: You can change the order of the items in the Import Queue and in the Active Import Queue by clicking on the label of the column that you want to sort by.

8. If the Import Queue contains items you do not want to process, you can remove them from the queue:
 - Click the Clear Entry button to remove the highlighted entry.
 - Click the Clear Queue button to remove all items from the queue.
9. To start the import process in the background, click the Background button.

The files currently in the queue are moved to the Active Import Queue and the import process starts. The progress of the import is indicated in the queue: the Status column temporarily changes to WORKING and then it shows the number of the frame it is processing. When a file is finished, the Status column changes to DONE.

10. Click the Exit Background I/O button or press the **F11** key again to go back to the Import Image menu.

You can now exit the Import Image menu and work anywhere in **flame**. To check on the status of your background jobs, press the **F11** key from any menu in **flame** and the Background Status menu appears, showing you the progress of the import. You can click the menu button to view your export jobs.

NOTE: You can perform any of the commands in the Background Status menu any time you access it, but to select items to import, you must access the menu from the Import Image menu.

Aborting the process

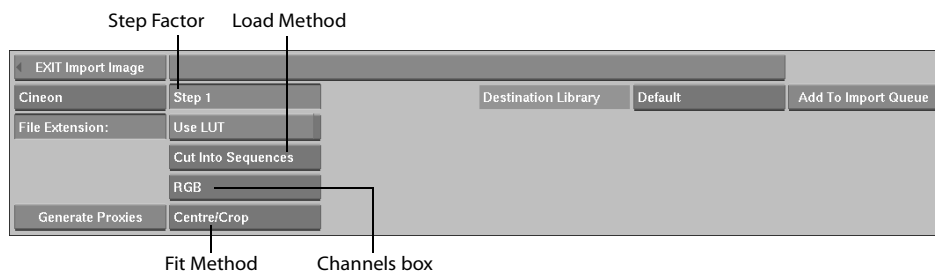
You can abort the job that is currently processing by clicking the Abort button. The frames that have already been processed are retained. For example, if the Status column reads 20 of 44 when you click Abort, you will have 20 of the 44 frames of the clip. If there are more items in the queue, processing of the next item begins.

Starting a new set of jobs

Once all the jobs in the Active Import Queue are done, you can start a new set. You can add items to the Import Queue any time, but you must wait until all the jobs in the previous set are done before clicking the Background button. Starting a new set clears the Active Import Queue of all completed jobs.

Import Image Options

Use the following options to control how images are imported.



Step Factor

When importing a selection of images (which may be part of a sequence or not), you can opt to skip frames, according to a factor that you specify. For example, if you set a step factor of two, every second frame will be imported, a step factor of 3 imports every third frame, etc. A step factor of 1 imports all frames.

When setting a step factor greater than 1, choose the appropriate Load Method:

- Load as One Clip creates one clip.
- Load as Single Frames creates individual frames for every imported image.
- Cut into Sequences does not create individual frames; it looks for a common base name for files, and creates individual clips based on the name.

See “Load Method” on page 328 for more details.

Use LUT

See “Using a Lookup Table” on page 330.

Create LUT

When you select Cineon or Dpx (Spirit) as the file type, the Create LUT button appears. This option allows you to create a custom LUT. For more information, see “Lookup Table Editor” on page 332.

Load Method

There are several methods you can use to load selected images.

Select:	To:
Load as One Clip	Load all selected images into a single clip.
Cut Into Sequences	Load all selected sequences and individual images into separate clips. Note: Selected images within a sequence are loaded as one clip.
Load as Single Frames	Load all selected images into separate, one-frame clips.

Channels

If the image has an alpha channel, you can load it with the image, either as is, or inverted. Select an option from the Channels box.

Select:	To:
RGB	Load the image file with no alpha channel.
RGB + Matte	Load the image and its alpha channel. Two clips are created: the first clip contains the RGB image, and the second clip contains the greyscale alpha channel image. Note: If the image does not contain an alpha channel, a single clip will be created.
RGB + Inv. Matte	Load the image and its alpha channel (inverted). Two clips are created: the first clip contains the RGB image, and the second clip contains the greyscale inverted alpha channel image.

Fit Method

If an image has a different size from the frame size of the partition, you can specify the method used to make the images fit in the current frame size. For example, when importing a film resolution image into a video resolution partition, the image size will be different. The images can be scaled, cropped, or borders can be added. Various combinations of these operations are available.

You can also specify the fit method when exporting images.

Centre/Crop and Pan — Center/Crop and Pan both copy pixels straight from the image to the framestore with no resizing.

- Center/Crop is the default option for importing images. If an imported image is larger than the frame size, the image is centered in the frame and cropped to size. If the image is smaller than the frame size, the image is centered and a black border is added.

NOTE: When exporting images, you can specify the border colour. See “Exporting Images” on page 338 for details.

- The Pan option lets you apply an offset (in pixels) to the X and Y positions of the image within the frame, so that it is no longer centered in the frame. An X/Y offset of 0,0 (no offset) aligns the lower-left corners of the image and the frame. Unused areas of the frame caused by the offset are filled with black (or another specified colour, when exporting images). If the image is larger than the frame, it is cropped. You can use positive or negative offset values.

Letterbox and Crop Edges — The Letterbox and Crop Edges options both use proportional scaling to scale the X and Y dimensions of the image, maintaining the correct aspect ratio. The imported image is either enlarged or reduced in size, as needed.

- The Letterbox option scales the longest edge of the image to fit into the frame and fills any resulting unused portion of the frame with black. You see the entire content of the original image, giving you a letterbox effect.
- The Crop Edges option scales the shortest edge of the image to fit into the frame and crops the longer edge. The frame will be filled and part of the original image will be missing (comparable to a pan and scan film transfer).

Fill — This option scales the X and Y dimensions of the image non-proportionally to make it fit in the current framestore partition. Note that this can change the aspect ratio, resulting in a distorted image.

Quality — When Letterbox, Crop Edges or Fill are selected, the Quality box appears, allowing you to choose the quality of scaling. The Coarse option gives the lowest quality with the fastest transfer time. The Medium and Quality options offer better image quality, but at slower rates of transfer. Bicubic is useful when importing an image with a lower resolution than that of the current partition. It reduces blur in the resulting image.

Using a Lookup Table

A Lookup Table (LUT) is an ASCII file that describes specific pixel values used in image display or conversion. You can use a LUT when you import and export an image. LUTs are most useful to convert a logarithmic image (such as a film image in Cineon or Dpx (Spirit) format) to a linear image (needed to work within **flame**) while maintaining colour accuracy.

There are three types of LUTs.

Use:	For:
Monitor LUTs	Viewing
Import LUTs	Conversion
Export LUTs	Conversion

Usually, the Export LUT you use corresponds to the exact inverse of the Import LUT. This ensures the colour accuracy of the exported images—it will be as if they had not been through a conversion. Import LUTs and Export LUTs are explained in the following sections.

Import LUTs

When you import an image, you may want to use an Import LUT to ensure that the image you work with resembles the final output. When you export the end result, use the corresponding Export LUT.

To use an Import LUT, you can:

- Use one of the default Import LUTs included with **flame**.
- Create one manually using an ASCII text editor.
- Create one interactively using the LUT Editor when importing Cineon or Dpx (Spirit) images. This is described in “Lookup Table Editor” on page 332.

Default LUTs

There are three default Import LUTs included with **flame**:

Use this LUT file:	To:
10logto12	Convert a 10-bit logarithmic Cineon image to a 12-bit linear image. Use this LUT if you are using a 12-bit partition.
10logto8	Convert a 10-bit logarithmic Cineon image to an 8-bit linear image. Use this LUT if you are using an 8-bit partition.
invertRGB	Convert an image by inverting the RGB values.

When you import a Cineon image, try using one of the default Import LUTs. If the default LUT does not provide the result you want, you may need to create a custom LUT. For more information, see “Lookup Table Editor” on page 332.

NOTE: If your image looks too dark, it may indicate that you need to adjust the gamma of your monitor.

Creating a Lookup Table Manually

You may want to create the LUT manually, using Jot or a comparable ASCII text editor.

The first non-commented line of a LUT file must use the following format:

```
LUT: < number of tables > < number of entries per table >
```

The total number of intensity values in the file must be equal to the number of tables multiplied by the number of entries per table. For example, for three tables of 256 entries each, the LUT file would contain a single column of 768 intensity values. Each entry must be a single integer, with one entry per line. The range of this entry may vary depending on whether you are mapping 8 bits to 8 bits (0-255), 8 bits to 10 bits (0-1023), or 8 bits to 12 bits (0-4095).

For example, to declare three tables (for Red, Green, and Blue) of 256 entries each, the first line in the LUT file would read:

```
LUT: 3 256
```

The Red channel (R) is mapped through the first table, the Green channel (G) is mapped through the second table, and the Blue channel (B) is mapped through the third channel. If there are four channels, R is mapped through the first table, G is mapped through the second table, B is mapped through the third table, and the alpha channel (A) is mapped through the fourth table. If there is only one table, this single 256-entry table is used for each of the Red, Green, Blue, and potentially alpha, channels.

A sample LUT is included in the `~/lut` subdirectory of the **flame** directory. (You can change the default LUT directory in the Environment Directory Pathnames section of the project configuration file.)

Export LUTs

When you export an image, you may want to use an Export LUT to ensure the image you export resembles the original image. This is necessary, for example, if you loaded an image with an Import LUT. The Export LUT you use should be the inverse of the Import LUT you used to import the image. For example, if you imported the image using the default 10logto8 LUT, then you should export the image using the default 8to10log LUT.

When you use the LUT Editor to create a custom LUT, **flame** automatically creates an Export LUT which is the inverse of the LUT you created. You should export the image using this Export LUT. For more information, see “Lookup Table Editor” on page 332.

There are two default Export LUTs included with **flame**:

Use this LUT file: To:

8to10log	Convert an 8-bit linear image to a 10-bit logarithmic Cineon image. Use this LUT if you are using an 8-bit partition.
12to10log	Convert a 12-bit linear image to a 10-bit logarithmic Cineon image. Use this LUT if you are using an 12-bit partition.

If you imported a Cineon image with a default Import LUT, you should export the image with the inverse default Export LUT.

Loading a Predefined LUT

You can load previously defined LUTs for use on clips that you are exporting, or on images that you are importing.

To load a predefined LUT:

1. In the Import Image menu or Export Image menu, click the Use LUT button.
2. Click the LUT button.

Existing LUTs appear in the file browser.

3. Click on a LUT to select it.

The Export Image or Import Image menu reappears and the name of the selected LUT appears on the LUT button. The LUT will be applied to the clip or image when exported or imported.

Lookup Table Editor

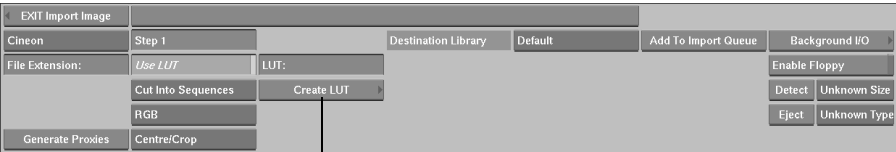
When importing Cineon or Dpx (Spirit) images, you can choose the Create LUT option in the Import Image menu to create a custom LUT interactively.

Once you create and save the LUT, you can use it for importing subsequent images. When you save the LUT, **flame** automatically creates an Export LUT which is the inverse of the custom Import LUT you created.

Before you create custom LUTs, you should make sure the monitor is properly calibrated. For information on calibrating your monitor.

To open the LUT Editor:

1. Open the clip library in which you want to store the images you are importing.
2. Click the Import Image button.
3. Select Cineon or Dpx (Spirit) in the File Format box.
4. Click Use Lut.



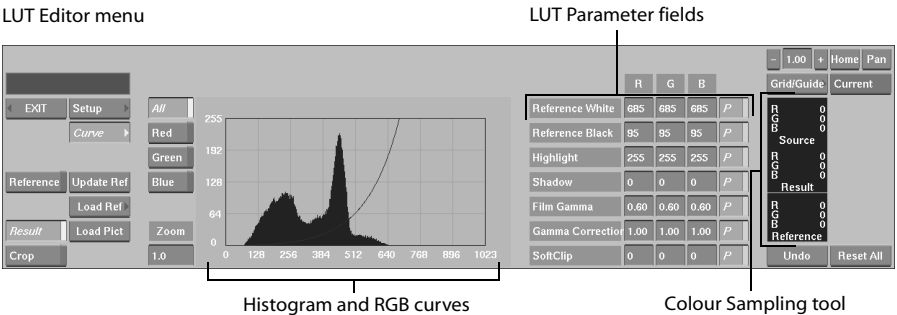
Create LUT button

5. Click the Create LUT button.
The file browser appears.
6. Use the Fit Method option box at the bottom of the screen to select the type of scaling or cropping you want to apply to the image. For a description of the available options, see “Fit Method” on page 328.
7. Select the Cineon or Dpx (Spirit) image you want to import.

NOTE: You should use an image that is representative of the scene. If possible, try to use a reference image that contains the darkest and brightest values in the shot.

The LUT Editor appears.

The image that appears in the image window is the result of the 10-to-8-bit conversion (or 10-to-12-bit conversion, if you are using a 12-bit partition). This conversion takes place according to the default LUT parameters. Modify the LUT Editor parameters until you get the result you want in the image window.



To create a custom LUT:

1. The image that first appears in the window is the Result image, using the default LUT parameters. For more information, see “Using the Result Image” on page 334. Display the Reference image by clicking the Reference button.

The image that is initially loaded as the Reference is the raw 10-bit Cineon or Dpx image scaled to 12-bit or truncated to 8-bit, depending on your current partition. For more information, see “Using a Reference Image” on page 334.

2. Click the Update Ref button.

The Reference image is updated with the default Result image. Now you can make changes to the LUT using the Result image, and compare the result with the default in the reference buffer.

3. Click the Result button.
4. Create a crop box so that you can see both the Reference and Result images at the same time. See “Using the Crop Tool” on page 335.
5. Modify the Ref. White and Ref. Black values, if necessary. See “Adjusting Reference White and Reference Black” on page 336.
6. Adjust the Film Gamma, if necessary. See “Adjusting the Film Gamma” on page 336
7. Adjust the RGB SoftClip values, if necessary. See “Adjusting the RGB SoftClip values” on page 336.
8. Adjust the Highlight, Gamma (midtones) and Shadow areas, if necessary. See “Adjusting Highlights, Gamma, and Shadows” on page 337.
9. Save the LUT.

flame automatically saves your custom Import LUT and the corresponding Export LUT.

Using the Result Image

When the Result button is enabled, the Import LUT is applied to the original Cineon image in the image window. By default, when you load an image into the LUT Editor for the first time, **flame** uses the default Import LUT. When you modify any parameter in the LUT Editor menu, the change is applied to the Result image.

To return to the original Import LUT, click the Reset All button. To use a different Result image, click Load Picture and select the file from the file browser.

Using a Reference Image

When the Reference button is enabled, the Reference image appears, which initially corresponds to the raw image adapted to the current partition depth (either truncated to 8 bits or scaled to 12 bits).

Use the Update Ref button to replace the Reference image with the Result image.

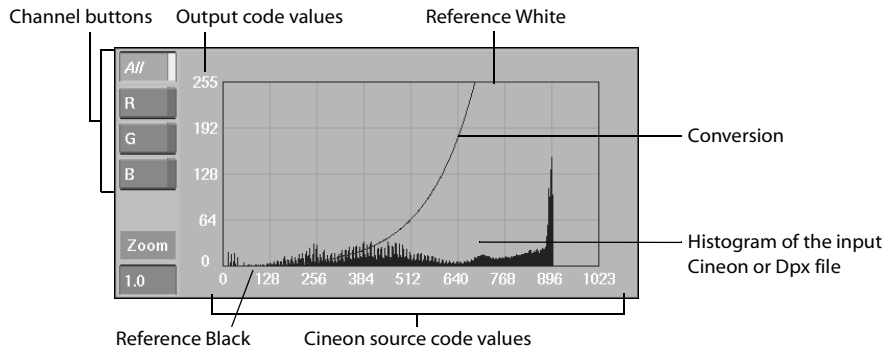
To use a different Reference image, click Load Ref and select the new Reference image from the desktop.

Using the Crop Tool

You can have parts of both the Reference image and the Result image visible at the same time by clicking Crop and drawing a box over a part of the image. The area inside the crop box corresponds to the Result image, and the area outside the crop box corresponds to the Reference image.

Using the Histogram and RGB Curves

The RGB curves are a visual representation of the logarithmic-to-linear image conversion LUT. The histogram shows the distribution of pixels in the Cineon or Dpx source file buffer.



The logarithmic Cineon source image is represented along the horizontal axis scale from 0 to 1023 (10 bits).

The output linear image is represented along the vertical axis scale from 0 to 255 (8 bits), or from 0 to 4095 for 12 bits.

The curves represent the mapping of the logarithmic data to the linear data (in other words, the LUT content). For example, in the previous figure, pixels in the logarithmic image with a code value of 512 are mapped to a value of 64 in the linear image.

The point at which the curve reaches a value of 0 on the vertical axis is called the Reference Black. Any values in the logarithmic image which are below the Reference Black value are mapped to black.

The point at which the curve reaches a value of 255 (or 4095 for 12-bit partitions) on the vertical axis is called the Reference White. Any values in the logarithmic image which are above the Reference White value are mapped to white.

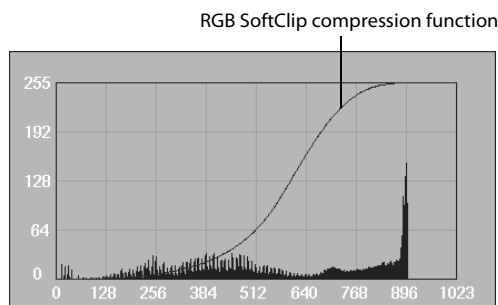
Adjusting Reference White and Reference Black

In converting from the full-range 10-bit digital negative to an 8-bit or 12-bit linear representation, the method is to limit the density range that is translated to that of a normal exposure with a scene contrast range of 100:1.

You must specify the code value of the reference white in the scene. The default value for a normal exposure is 685. The default value for reference black is 95 which is the code value for *Dmin* in the calibration of the Cineon scanner.

Adjusting the RGB SoftClip values

Once you set the Reference White value, you can soften the shoulder of the conversion curve by adjusting the RGB SoftClip value for any or all of the RGB channels. When you soften the shoulder of the conversion curve, you soften the transition of colours toward Reference White.



To adjust the RGB SoftClip values:

1. To modify all channels at the same time, enable the P (Proportional) button.
2. Change the RGB SoftClip value you want to modify.

The shoulder of the selected R, G, or B curve changes dynamically.

Adjusting the Film Gamma

The Gamma of a typical film negative is 0.6. You might want to adjust the Film Gamma. For example, if you know that the negative was developed to a Gamma different from 0.6 or if you want the film to look like it was push processed when it was not, you might want to set the Film Gamma to a value other than 0.6.

If you want to use a different Film Gamma, enter the new value in the Film Gamma field.

If you find that you must increase the Film Gamma to between 1.1 and 1.3 to be satisfied with your result, this means you are compensating for the gamma of your monitor during the conversion. To avoid such compensation during the conversion, you need to calibrate the monitor.

Adjusting Highlights, Gamma, and Shadows

Adjust the brightness (luma) of selected areas of the image by modifying the Shadow, Gamma and Highlight parameters.

To adjust Highlights, Gamma and Shadows:

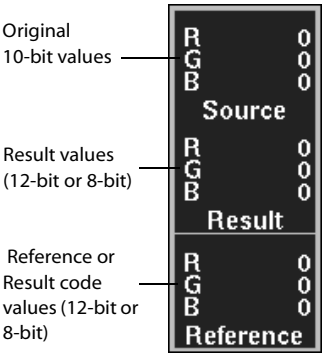
- 1. To modify all channels at the same time, enable the P (Proportional) button.
- 2. Change the Highlight, Gamma or Shadow value you want to modify.

The selected R, G, or B curve changes dynamically.

Using the Colour Sampling Tool

Use the Colour Sampling tool to compare pixel values in the Original (10-bit), Result, and Reference images. It works much like the Colour Sampling tool in the Colour Corrector.

Colour Sampling Tool



When you sample with the uppermost boxes, the Original 10-bit and Result values are shown in the top and middles boxes. The lower box operates independently, and can be used as a reference. It displays either the Reference or Result values, depending on whether the Reference or Result image is selected.

To sample a value in the image:

- 1. Click one of the two sampling boxes.
The cursor changes to a color picker.
- 2. Click on the image. The RGB pixel values and colour are displayed.

Saving a Custom LUT

Click the Save button to save the current parameters to a LUT file. You are prompted to enter a file name. When saving a LUT, two versions are created: an Import LUT and an Export LUT.

flame adds the designations “10to12” and “12to10” (or “10to8” and “8to10” for 8-bit partitions), respectively, to the file name that you specify, along with the extension *.lut*. For example, for an 8-bit partition, if you enter *mylut*, the following two LUT files are created and stored in the LUT directory: *mylut_10to8.lut* (Import LUT) and *mylut_8to10.lut* (Export LUT).

Loading a Custom LUT

To load a previously defined LUT into the LUT Editor, click the Load button. Existing LUTs appear in the file browser. Select a LUT. You are returned to the LUT Editor.

NOTE: Only LUTs that were provided with **flame** or created using the LUT Editor can be used because they contain data specific to parameters used for the conversion computation. If you select a LUT that is not compatible, you will receive an error message.

Removing a Custom LUT

If you wish to delete a previously defined LUT, click the Load button. The file browser appears. Select the LUT you want to delete and click the Remove button.

Exporting Images

Use the Export Image command to export a clip or a single frame from the desktop reels to the file system. You can export clips to a number of different formats. See “File Formats and File Extensions” on page 322 for a list of formats to which you can export. Exported clips can be used on other platforms (for example, a Macintosh or PC).

Image Sequences

When you export a clip to the file system, each frame in the clip is saved as a separate image file. The images are assigned a sequential number which is appended to the file name. (You can modify the numbering format.) In the file browser, the sequence of image files appears under a parent entry that represents the entire sequence. Similar to the way parent entries work in clip libraries, you can expand or collapse the parent entry to view or hide the entire sequence, and select the whole sequence by clicking on the parent entry.

To export an image:

1. Click the Export Image button in the Library menu.
The cursor changes to a red selection arrow and the Front/Matte and Clip/Range boxes appear.
2. Choose Front to export RGB images or Front/Matte to export RGBA images (where the format permits).

3. Select Clip to export an entire clip, or Range to export a range of frames within a clip.
4. Select the Front clip that you want to export. When exporting a range of frames within a clip, select the first frame in the range.
5. If you selected Range, the cursor changes to a green arrow. Select the last frame in the range of frames to export.
6. If you selected Front/Matte, the cursor changes to a blue arrow. Select the matte to be exported as the alpha channel.

The Export Image menu and file browser appear.

Export Image menu

File Format box Fit Method Border Colour box

7. If the file browser is not pointing to the directory to which you want to export the clip, change the current directory (see “The File Browser” on page 61 for details).

NOTE: If your computer has a floppy drive, you can export images directly to a diskette.

8. Select the required file format from the File format box. Change the default file extension if required.
9. Specify whether you are exporting a single frame or an entire clip:
 - Select Single Frame to export a clip composed of one frame.
 - Select Clip to export a clip composed of several frames.

NOTE: If you select Single Frame and the clip has more than one frame, only the first frame is exported.

When you enable the Clip option, two Numbering Format fields appear. You can change the numbering format for the image files as described in “Changing the Numbering Format” on page 342.

10. If you want to change the name of the exported image file, click the Name field, type a name using the on-screen keyboard, and press **ENTER**.
 - In Clip mode, a sequential number is appended to each file name (see “Changing the Numbering Format” on page 342).
 - In Single Frame mode, the file name has no number appended to it.

11. To use a predefined LUT, load it with the Use LUT button. For details, see “Loading a Predefined LUT” on page 332.
12. Select the fit method. For more information, see “Fit Method” on page 328. If you wish, change the colour of unused portions of the frame using the Border Colour box.
13. If needed, use the Width and Height fields to change the size of the exported images. By default, the image will have the dimensions of the current partition.
14. When all settings are to your liking, click the Export Images button to start exporting.
When the export is finished, you are returned to the Library menu and the desktop reels.

Background Export of Images

Using the Export Image menu, you can export one or more image files in the background while you perform other tasks. To export several files, you select them and place them in a queue, then process them as a batch. You can assign different clip options (such as Fit Method) to each file that you add to the queue.

HINT: To simultaneously import and export files in the background, simply start one of the processes, then go into the other menu (either the Import Image menu or the Export Image menu) and start that process.

To export clips as a background process:

1. In the Library menu, click the Export Image button.
2. Select the first clip you want to export.

The Export Image menu appears.



NOTE: To view the floppy drive buttons, tap the swipe bar on the left or right side of the menu.

3. Point the file browser to the directory where you want to save the clip(s).
4. Specify the Fit Method, and other options. See “Export Image Options” on page 342.

NOTE: Bicubic quality is not available when using background export.

5. Click the Export to Queue button.

The clip appears in the Export Queue.

6. Exit to the desktop and repeat steps 1-4 for any other clips you want to export.
7. Click the Background I/O button or press the **F11** hot key.

The Background Status menu appears.



8. Verify that all the clips you want to export are in the Export Queue. If needed, you can remove items from the queue by selecting them and clicking the Clear Entry button.

NOTE: You can change the order of the items in the Export Queue and in the Active Export Queue by clicking on the label of the column that you want to sort by.

9. To start the export process in the background, click the Background button.

The files currently in the queue are moved to the Active Export Queue and the export process starts. The Status column indicates the progress of the export.

10. Click the Exit Background button or press the **F11** key again to go back to the Export Image menu.

You can now exit the Export Image menu and work anywhere in **flame**. To check on the status of your background jobs, press the **F11** key from any menu in **flame** and the Background Status menu appears, showing you the progress of the export. You can click the Menu button to view your import jobs.

Export Image Options

Use the following options to control how images are exported.

NOTE: The Fit Method option is described in “Fit Method” on page 328.

JPEG Compression

When you export clips to jpeg images, you can use the Quality field to specify the quality factor. A value of 0 gives the lowest quality (and greatest compression), and a value of 100 gives the best quality (no compression).

Lossless Compression

When you export clips to Sgi and Tiff images, you have the option of applying lossless compression. Enable the Compression button to apply compression.

Bit Depth

Some file formats support multiple bit depths. For example, when exporting to Sgi files, you can select 8 or 16 bits. The default bit depth is that which best matches the bit depth of the current partition. See “File Formats and File Extensions” on page 322 for information on supported bit depths for various file formats.

Generating Proxies for Exported Images

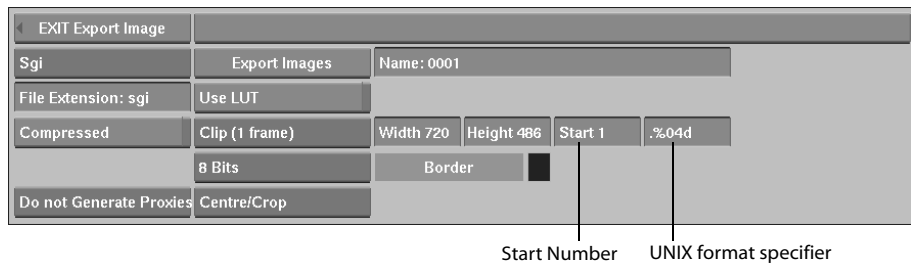
You can generate proxies when you export a clip. You see the proxies when viewing the image files in the file browser using Proxies mode. If a proxy has not been generated for an image file, the proxy appears grey.

To generate proxies for an exported clip, specify a Proxy Generation option:

Select:	To:
Do not Generate Proxies	Not generate a proxy for the exported clip.
One Proxy Per Clip	Generate one proxy for the exported clip.
One Proxy Per Frame	Generate a proxy for every frame of the exported clip.

Changing the Numbering Format

When you export a clip, the individual image files are numbered sequentially so that the order of frames in the clip can be maintained if you import the clip. You can specify the first number in the sequence using the Start Number field. The default is 1.



You can also use the UNIX format specifier to add leading zeroes before the number. These ensure that the images are listed and stored in the correct order. Since the files are stored in numeric order, it is a good idea to use leading zeroes. Without the leading zeroes, files would be stored in the following order: 1, 10, 11, 12,..., 18, 19, 2, 20, 21, and so on.

For example, to add leading zeroes to the numbering format of a clip called *testclip*, you could use the UNIX format specifier “`:%04d`”. The resulting file names would be:

```
testclip.0001
testclip.0002
testclip.0003
testclip.0004. . .
```

The “4” in the format specifier tells the UNIX operating system to use four digits in the numbering format. You could use 3, 5, or another number instead. However, four digits is sufficient for most clips, as it provides correct numbering in clips up to 9999 frames long.

The “0” in the format specifier tells the operating system to place leading zeroes before the file number when there are fewer than four digits in the sequence number.

Removing Images

You can use the file browser from the Import Image menu to remove image files from the file system.

To remove images:

1. Open the Import Image menu.
2. Display the directory containing the image files you want to remove.
3. Select the files you want to remove.
4. Click the Remove button and confirm.

[illegible]

OMF eliminates the need to perform time-consuming translations when transferring files between incompatible applications.

Summary

In this chapter, you learn about:

- “Exporting a Clip as an OMF File” on page 345
- “Importing a Clip from an OMF File” on page 349

About OMF Files

Open Media Framework (OMF) Interchange is a standard file format used for the exchange of digital multimedia data between applications and across platforms. In **flame**, you can import and export OMF files that contain digital video data. **flame** supports most Avid Video Resolution (AVR) formats.

Exporting a Clip as an OMF File

When you export a clip as an OMF file, the file is saved in a directory in the file system. You can load an OMF file from the directory into another application, or transfer it to another platform.

NOTE: The default directory for OMF files is set in the Environment Directory Pathnames section of the project configuration file. This is the same default directory as for Import and Export Image. For more information, refer to the **flame** *Installation Guide*.

A single OMF file can contain more than one clip, although it is unlikely that you will want to save more than two clips in the same file. Make sure that you are aware of the import and export capabilities of your OMF-compatible software. In **flame**, you can export more than one clip to an OMF file, but you can only import the first clip in any OMF file.

If you are exporting the clip for use with Avid® software products, make sure you select the appropriate Avid Video Resolution (AVR). For more information, refer to your Avid documentation and to “Avid Video Resolutions” on page 348.

To export a clip to an OMF file:

1. Click the Export OMF button in the Library menu.
2. Select the clip that you want to export.

The Export OMF menu appears.

File Name field Clip Name field

EXIT Export OMF	File: /usr/discreet/project/effects/Marika_Project/images/cat.omf		Fit Width	Box Width 720
Process	Clip: cat		Fit Height	Box Height 486
	Overwrite	AVR : None	Fit Frame	Box X 0
		Use JPEG	Reset	Box Y 0

Append/Overwrite box

By default, the name of the selected clip is used as the file name. The file extension *.omf* is added automatically to the file name. You cannot change or delete the file extension.

3. To enter a new name or change the default directory for the OMF file, click the File Name field.

The file browser appears.

- To save the clip in a new file, use the keyboard to enter the file name and click Enter. You are returned to the Export OMF menu.
 - To overwrite an existing file, select the file that you want to overwrite in the file browser and click Enter. You are returned to the Export OMF menu. Select the Overwrite option in the Append/Overwrite box.
 - To append the new clip to an existing file, select the file in the file browser and click Enter. You are returned to the file browser. Select the Append option in the Append/Overwrite box.
4. The name of the selected clip appears in the Clip Name field. If you want to change the name, click the Clip Name field and enter the new name.
 5. If you will be using the OMF file in an Avid product, select an option from the AVR option box. For more information, see “Avid Video Resolutions” on page 348.

EXIT Export OMF	File: /usr/discreet/project/effects/Marika_Project/images/cat.omf		Fit Width	Box Width 720
Process	Clip: cat		Fit Height	Box Height 486
	Overwrite	AVR : Studio 1 field	Fit Frame	Box X 0
		AVR2s	Reset	Box Y 0

AVR option box

- 6. If the AVR option box is set to “AVR: None”, you can compress the file to save space by enabling the Use JPEG button. JPEG compression achieves a very high compression rate, but some detail is lost. JPEG compression is always used with AVR.
- 7. You can export all or part of a clip. Enter the timecode or frame number of the first frame in the field to the left of the timeline. Enter the number of the last frame in the right field.



- 8. Click Process to export the clip.

Cropping the Image

Usually, you will want to export an entire image. However, if you want to export only a part of an image, use the crop box to select part of the image and discard the unselected area. If you change the size of the crop box, the selected region of the image is resized to the AVR when it is exported to an OMF file.

The crop box is the red outline that appears around the perimeter of the frame. You can resize and move the crop box to include any rectangular area of the frame. You can manipulate the crop box interactively or by using the menu controls.

The crop box cannot be made active on a frame-by-frame basis. The setting you use at any one frame is used for every frame in the clip.

Fit Width	Box Width 720
Fit Height	Box Height 486
Fit Frame	Box X 0
Reset	Box Y 0

Moving the crop box — To move the crop box to a new position while maintaining its current dimensions, click and hold inside the crop box and drag it to a new location. As you move the crop box, the X and Y coordinates of the lower-left corner of the crop box are updated in the Box X and Box Y fields.

Resizing the crop box — To change the size of the selection, click and hold on an edge or corner of the crop box and drag until the crop box is the required size. The width and height of the crop box are updated in the Box Width and Box Height fields. Or, enter the width and height directly using these fields.

You can also set the width and height of the crop box automatically (see the table below).

Click:	To:
Fit Width	Match the width of the crop box to the width of the frame. The height is adjusted to preserve the aspect ratio of the crop box.
Fit Height	Match the height of the crop box to the height of the frame. The width is adjusted to preserve the aspect ratio of the crop box.
Fit Frame	Fit the crop box to the entire frame.
Reset	Reset the crop box to its default size and position. The default size and position may vary according to the selected AVR option.

Avid Video Resolutions

When you export a clip to an OMF file, you can select an AVR to maintain compatibility with Avid software products. Select a video resolution from the AVR option box before you export the clip.

Select:	For:
AVR: None	Any image size, with or without JPEG compression. Select this option if you do not intend to use the clip with Avid software products.
AVR: NuVista 1 Field	Square pixel images, only one field.
AVR: NuVista 2 Field	Square pixel images, both fields.
AVR: Studio 1 Field	Non-square pixel images (601 format), one field.
AVR: Studio 2 Field	Non-square pixel images (601 format), both fields.
AVR: Studio multicam	Non-square pixel images (601 format), half resolution.

When you select an AVR format, a box with a list of sub-options appears:

The AVR format:	Has these sub-options:
NuVista 1 Field	AVR 1-5, and AVR 1e-6e
NuVista 2 Field	AVR 25-27
Studio 1 Field	AVR 2s, 3s, 4s, 5s, 6s, 8s, 9s
Studio 2 Field	AVR 12, 70, 71, 75, 77
Studio multicam	AVR 2m-6m

Generally, a higher AVR results in a lower compression ratio and a higher quality image. For example, in Studio 2 Field, AVR 77 will result in a higher quality image than AVR 12.

When you export an OMF, the image file size depends on the AVR format you select and the current video standard in which you are working.

For this standard and AVR: Image sizes are:

NTSC - NuVista	640 x 480 pixels
NTSC - Studio	720 x 486 pixels
PAL -NuVista	640 x 576 pixels
PAL -Studio	720 x 576 pixels

If you change the size of the crop box, the selected region of the image is resized to the AVR resolution when it is exported to an OMF file.

JPEG Compression

If you want to save file space, you can compress the stored image by clicking the Use JPEG button. JPEG compression achieves a very high compression rate, but some detail is lost.

The Use JPEG button is only available when the option “AVR: None” is selected. JPEG compression is used automatically at all AVRs.

Importing a Clip from an OMF File

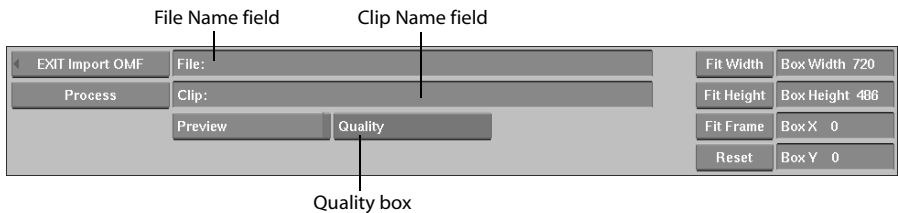
Use the Import OMF command to load a clip from an OMF file to the desktop reels. Although several clips can be contained in a single OMF file, most metadata is not imported and neither is audio. In **flame**, you can only import the first clip in the file.

When you import a clip, JPEG decompression is used automatically if the data in the file was compressed. AVRs are also handled automatically.

To import a clip from an OMF file:

1. Click the Import OMF button in the Library menu.
2. Select a destination reel for the imported clip.

The Import OMF menu appears.



3. Click the File Name field.
The file browser appears.

4. Select the file that contains the clip you want to import.

The first frame of the first clip in the file appears in the image window. In **flame**, you can only import the first clip in the file.

5. To change the name of the clip as it will appear on the desktop reels, click the Clip Name field and enter a new name.
6. Choose the quality of the imported image by selecting an option in the Quality box.
7. You can scroll through the clip using the image window controls. Since the clip is being played from the file system rather than from your framestore, playback may be slow.

You can import all or part of a clip. Enter the frame number or timecode of the first frame in the field to the left of the timeline. Enter the number or timecode of the last frame in the right field.

NOTE: The display of frames or timecodes is controlled in the Preferences menu.

8. Click Process.

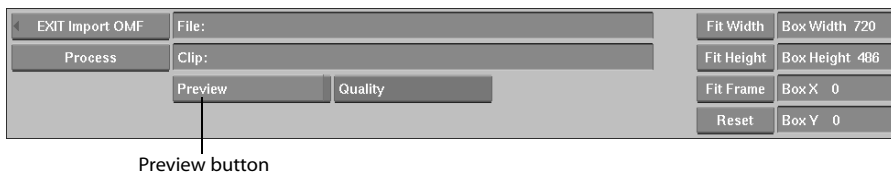
The clip is loaded on the destination reel.

Changing the Position or Size of the Image

You can use the crop box to change the size or position of the imported image within the frame. Initially, the crop box is the same size as the image stored in the OMF file. For information on how to move and resize the crop box, see “Cropping the Image” on page 347.

Unlike Export OMF, if you change the size of the crop box, the image is scaled to fit the crop box (unless you use Fit Width or Fit Height).

You can preview the cropped clip before importing it to the desktop reels by clicking Preview.



Use archiving for off-line storage of clips, backing up projects, exchanging content between systems, and freeing up space on the framestore.

Summary

In this chapter, you learn about:

- “Archive Compatibility” on page 352
- “Accessing the Archive Menu” on page 353
- “Archive Structure” on page 354
- “Archive Table of Contents” on page 354
- “Auto Testing” on page 355
- “Estimating Archive Size” on page 355
- “Creating an Archive” on page 356
- “Opening an Archive” on page 362
- “Saving to an Archive” on page 364
- “Loading Entries from an Archive” on page 369
- “Removing Entries from an Archive” on page 371
- “Closing an Archive” on page 372
- “Recovering an Archive” on page 372
- “Using an ASCII TOC to View an Archive” on page 374
- “Deleting Tables of Contents” on page 374
- “Archiving Setups” on page 375

About Archiving

Archiving is the process of copying film or video clips to a file. You can archive and restore individual clips, soft clips as well as desktops, reels. In addition, you can archive and restore a project's setups.

The Archive menu provides an organized way of archiving and restoring clip libraries, desktops, reels, soft clips, and individual clips.

You can create an archive on the following devices:

- VTRs (digital with little or no compression such as Digital Betacam or D1).
- UNIX tape devices such as a Exabyte, DAT, DLT, and DST (with version 3.1.2 or later of the DST driver).
- File (to any IRIX-supported file system). You can also use NFS (Network File System) to archive to a remote file system.

The devices that you use for archiving must be defined in the CLIPMGTDVICE KEYWORD section of the *init.cfg* configuration file. The devices that are enabled in this file will be available in **flame**. For information on how to enable or disable devices, refer to the ***flame** Installation Guide*.

NOTE: For information on archiving 24p projects, see “Archiving 24p Projects” on page 319.

Archive Compatibility

Using **flame** 7.0, you can restore clip libraries, desktops, EditDesks, reels, Source areas, Record areas, soft clips, or individual clips from archives created with the following Discreet products:

- **smoke** 1.x and later
- **fire** 2.x and later
- **flame** 4.0.2, 4.4IR, 4.5 and later
- **flint** on Octane SE 6.0 and later
- **flint** on Indigo2 6.0
- **flint** 4.0.2, 4.4I, and 5.0
- **effect** 5.5 and later
- **inferno** 1.0, 1.1, 1.4IR, 1.5 and later

NOTE: Older archive formats are opened in read-only mode—you cannot append to an older archive format from **flame** 7.0.

Accessing the Archive Menu

Use the Archive menu to:

- Format an archive.
- Select an archiving device.
- Open an archive in order to save or restore clips.

To access the Archive menu:

1. Open the Library module.

<i>LIBRARY</i>				
EDITING	Load	Import Image	Input Clip	Delete
PROCESSING	Save	Export Image	Output Clip	Name
EFFECTS	Archive	Import OMF	DDR Library	Move
FORMAT	Network	Export OMF	Tape Library	Copy
SYSTEM	Spirit Import	HDCAM Codec		Search

2. Click the Archive button.

<i>LIBRARY</i>				
EDITING	Load	Import Image	Input Clip	Delete
PROCESSING	Save	Export Image	Output Clip	Name
EFFECTS	Archive	Import OMF	DDR Library	Move
FORMAT	Network	Export OMF	Tape Library	Copy
SYSTEM		HDCAM Codec		Search

The Archive menu appears.

EXIT Archive				
Clips				Delete
Device : Tape	Open	Open Using OTOC	Tape Size Estimate	Name
Tape : DAT				Move
Format	Tape : /dev/rmt/tps0d3ns			Copy
				Search

Archive Structure

The first entry of an archive is the archive itself. The archive entry is the parent entry for all the other entries in the archive. Entries within the archive entry are structured in the same way as entries in a clip library.



Entries in an archive can be selected, expanded, and collapsed like those in a clip library. Indentation is used to indicate the parent and child relationship among archive entries. For more information, see “Clip Library Structure” on page 203.

In addition to clip library, desktop, reel, soft clip, and clip entries, an archive can also contain partition entries. You can only save a partition entry for the partition you are currently using.

Archive Table of Contents

When you create an archive, **flame** creates a table of contents, or header, on the medium. This table of contents contains a list of the contents of the archive, as well as other information such as the order in which clips are assembled on the archive, clip IDs, transitions, and timecodes. When you retrieve an archive, **flame** needs this information to successfully retrieve the material.

Online and ASCII Tables of Contents

flame also saves a copy of the table of contents in the UNIX file system. This copy is referred to as the Online Table of Contents, or OTOC. You can open an archive in read-only mode using the OTOC. In read-only mode, you can load entries from the archive. With VTR archives, you can

open an archive in read-write mode using the OTOC, and save and remove entries from the archive.

The OTOC is most useful to:

- Recover material in an archive when the table of contents on the medium is corrupted. For more information, see “Recovering an Archive” on page 372.
- Open an archive more quickly.

When an OTOC is created, **flame** also creates an ASCII text copy of it (the ATOC). Use the ASCII copy to view the contents of an archive without opening it. For example, use it to view the contents of a VTR archive without connecting to the VTR.

The OTOC and ATOC are saved to the pathname specified in the ArchiveLibrary token in the *init.cfg* configuration file. If the ArchiveLibrary token is disabled, the OTOC and ATOC are saved to */usr/discreet/archive*.

The OTOC is updated each time you close the archive.

Auto Testing

Auto testing of archives is performed when a VTR archive is formatted and when it is opened in read/write mode. The test scans binary and play/record delay information to verify the quality and accuracy of the video signal.

When you close an archive, **flame** writes and verifies three copies of the table of contents information and the binary information, checking whether it can write, read, and verify the integrity of the archived material. It updates the slates whether or not errors are found. Three copies are used in case one becomes corrupted (for example, from tape damage).

Estimating Archive Size

This feature provides an estimate of the media space (in megabytes) needed to archive selected material. The size estimate takes header information into account.

The space required to archive material varies by archiving device. The following settings also influence the amount of space required for an archive:

- Include Audio
- Compact/Normal archive

To estimate tape size:

1. Select the device type from the Archive Device box.
2. Select a VTR device from the Device Name box. The device name box is present for VTR and Tape devices other than DST. It is not displayed when only one device of a given type can be specified in the *init.cfg*.

3. Click Archive in the Library menu.
4. Click Tape Size Estimate.
The clip library appears.
5. Select one or more clips in the clip library.
6. Click Tape Size Estimate.

The estimate appears in the field immediately to the right of the Tape Size Estimate button.

Creating an Archive

Before you can save entries to an archive you must open an existing archive or create a new one. The steps for creating an archive depend on your archiving device you are using. Refer to the section corresponding to your archiving device:

- “Creating a VTR Archive” on page 356.
- “Creating a Tape Archive” on page 359.
- “Creating a File Archive” on page 360.

Creating a VTR Archive

To archive to a VTR, the VTR must be declared in the *init.cfg* configuration file; otherwise, it will not appear as an option in the Device Name box. For information on modifying the *init.cfg* configuration file, refer to the **flame** *Installation Guide*. Make sure you have black striped tape with continuous non-drop timecode before creating a VTR archive.

You can create a VTR archive on a tape that already contains material or add material after an archive on a tape. For example, on a 70-minute tape, you could allocate 20 minutes to an archive by setting its start timecode at 01:00:00:00 and defining the length of the archive as 20 minutes in the Length field. You could then add any other material you want starting after 01:20:00:00.

All VTR archives contain a Header Info Slate, a single frame providing information on the archive. Located on the archive before the table of contents, the Header Info Slate provides the name of the archive, the name of the OTOC, start time code, creation date and time, modification date and time, and the minimum size of the clip library buffer (in MB) required to restore the archive (see MaxLibSize init.cfg token).

To create a VTR archive:

1. Open the Archive menu.
2. Select VTR from the Archive Device box.

The options for archiving to a VTR device are displayed.

Archive Device box

EXIT Archive				
Clips				Delete
Device : VTR	Open	Open Using OTOC	Tape Size Estimate	Name
VTR : DigBeta	Open Read/Write			Move
Format		Rewind On Close	Load Old D1	Copy
Engineering	Auto Start			Search

Device Name box

Manual Start Option

3. Select a VTR device from the Device Name box.

NOTE: If the VTR you want to use does not appear in the list, open the *init.cfg* configuration file to make sure it is declared. You can declare several VTR devices. For information on modifying the *init.cfg* configuration file, see the *flame Installation Guide*.

4. Insert a black striped tape with continuous non-drop timecode in the VTR.
5. Click the Engineering button to set specific parameters such as Timecode Source and Precision in the Engineering menu:
 - The recommended Timecode Source for archiving is LTC.
 - For Precision, if you are using an Octane, use 8-bit for D1 VTRs and 10-bit for digibeta and D5 VTRs.



For more information, see “Setting Engineering Menu Options” on page 240.

NOTE: When you open an archive in read/write mode, play delay and pre- and post-roll values are verified and set in the auto test that is run before archiving begins; therefore, you do not need to set these options.

6. If you want to set the start timecode for the archive, select the Manual Start option and enter a timecode.

The start timecode that appears by default is set in the *init.cfg* configuration file.

If the tape already contains material, find out what the start timecode for the archive should be by playing the tape in the Input Clip menu. For more information, see “Playing the VTR” on page 237.

NOTE: If you are using an Ampex DCT D1 deck, you will find that the VTR cues to the wrong timecode (the next ascending minute) when archiving. To work around this problem, select the Manual Start option and enter the desired timecode.

7. Click Format.

The Format menu appears.

Length field

8. Enter the name for the archive entry and any additional comments in the Name and Comments fields.
9. Enter the available space after the Start Time Code in the Length field. Enter the length in minutes.

NOTE: If you are creating the archive on a tape that already contains material, make sure you define the length of the archive as the space you want the archive to use, not the total length of the tape.

10. Click Format and then Confirm to continue.

The new archive is created on the VTR. Click Open to open the archive and save the contents of the current partition, or clip libraries, desktops, reels, soft edits, or individual clips. For more information, see “Saving to an Archive” on page 364.

HINT: Write down the archive name and creation date on the tape label. This will help you to select the appropriate table of contents file, if you ever need to restore your archive.

Tips for Better VTR Archiving

Use the following tips to get better results when archiving:

- Set the system reference of the VTR to External Analog.
- Check **flame**’s field dominance and make it consistent with your VTR and VCP settings.
- Track the VTR to get the best RF (Reference) value and error-rate level. If your VTR has an auto-tracking feature, disable it and track manually, or place it on “one shot.”
- Ensure that the IRIX “videod” configuration flag is set to “on.” You can check it by using the “chkconfig” command. If the flag is set to “off,” log in as root and enter the command:
chkconfig videod on.
- Click the Engineering button and check the following Engineering menu setting:
 - Timecode type: Make sure the VITC & LTC timecodes are the same. If they are not, set timecode type to LTC.
 - Input format and sync: Make sure that input format and sync match the device.
 - Output format and sync: “House” is the recommended output sync.

For more information, see “Setting Engineering Menu Options” on page 240.

NOTE: For more tips on archiving to VTR, see the *Archiving to VTR Troubleshooting Guide*, an article in the Knowledge Base on the Discreet Support website (www.discreet.com). To access the guide, do a search in the Reference category using “VTR” as the keyword.

Creating a Tape Archive

Before archiving to a DAT, Exabyte, DST, or DLT device, make sure:

- The selected device is a fixed-block-size device, such as tps0d3ns or tps0d7ns.8500.
- The tape device is declared in the CLIPMGTDVICE KEYWORD section of the *init.cfg* configuration file. For more information, see the *flame Installation Guide*.

NOTE: You must have version 3.1.2 (or a later version) of the DST driver to use the DST drive with **flame**.

To create a tape archive:

1. Open the Archive menu.
2. Place a tape in the tape device.
3. From the Archive Device box, select Tape as the type of device you are archiving to.
Options specific to the tape device are displayed.

Archive Device box

EXIT Archive				
Clips				Delete
Device : Tape	Open	Open Using OTOC	Tape Size Estimate	Name
Tape : DAT				Move
Format	Tape : /dev/rmt/tps0d3ns			Copy
				Search

Device Name box

4. The Device Name box lists the names of the tape devices that are defined in the CLIPMGTDVICE KEYWORD section of the *init.cfg* configuration file and that are connected to your workstation. Tape devices are detected automatically when you first enter the Archive menu. If needed, select a different tape device.
5. Click the Format button.
The Format menu appears.

File Name field Capacity box

6. Use the File Name and Comment fields to enter a name for the archive and any additional comments.
7. Select the capacity of the tape from the Capacity box.
If you cannot find the description of the tape, select Other as the capacity and use the adjacent field to enter the capacity of the tape in MB.
8. Press the Block Size box and select Default to use the default value from the *init.cfg* configuration file or specify an appropriate block size in the adjacent field.
9. Click Format and then Confirm to continue.

A new archive is created on the tape. Click Open to open the archive and save the current partition, or clip libraries, desktops, reels, soft edits, or individual clips. For more information, see “Saving to an Archive” on page 364.

NOTE: Write down the archive name and creation date on the tape label. This will help you to select the appropriate OTOC if you ever need to restore the archive.

Creating a File Archive

You can create a file archive on any IRIX-supported file system. When you format the archive, **flame** determines the available space and creates a table of contents file where this information is recorded. The archive itself is in a separate file, whose size is determined by the amount of material archived within it.

NOTE: The table of contents and archive files are both given the name that you specify when you create the file, but the archive file has the extension *.seg*.

To create a file archive:

1. Select File from the Archive Device box.
By default, the File Path field shows the name and path of the archive file declared using the ClipMgtDevice File token in the *init.cfg* configuration file.

File Path field

You can:

- Use the default archive file, “archive” in the */usr/tmp* directory (skip to step 3). However, make sure there is no previously archived material in it before formatting.
 - Create a new archive file.
 - Select an existing file archive and reformat it, thereby losing all previously archived entries within it.
2. To create a new archive file or select an existing one, click the File Path field. The file browser appears. If needed, change the directory, and then either:
- Use the keyboard to enter a name for a new archive file and click Enter.
 - Select an existing archive file.

You are returned to the Archive menu and the name and path of the new or selected archive file appears in the File Path field.

3. Click Format.

The Format menu opens.

File Name field

4. Use File Name and Comment fields to enter a name for archive entry and any additional comments. (Each archive file contains one archive entry within which all other archived entries are saved.)
5. Click the Format button. A message appears asking you to confirm that you want to create a new archive on the selected device. Click Confirm.



WARNING: If you selected an existing archive, any material within it is deleted when you reformat it.

The new file archive is created and the Archive menu is automatically displayed. You can now save partitions, clip libraries, desktops, reels, soft edits, and clips in the archive.

Opening an Archive

Once you create an archive, you can add entries to it at any time. For example, you can save entries to an archive after you create it, close it, and open the same archive later to add new entries. In the case of file and VTR archives, you can also remove entries.

flame creates online and ASCII tables of contents for any archive that you open, including archives that were created in older versions of the software.

To open an archive, use either the table of contents on the medium or the OTOC. Opening an archive with the OTOC is faster because the information is read from the computer hard drive rather than from the tape. You can load entries from an archive using the OTOC.

For VTR archives, you can save, load, and remove entries from the archive using the OTOC.

You can open a VTR archive in read/write or read-only mode. Archives created with a different format (from older versions) are opened in read-only mode.

To open a VTR archive:

1. Select VTR in the Archive Device box.
2. Select the VTR device.
3. Insert the archive tape in the VTR.
4. Select the start timecode by doing one of the following:
 - Select the Autostart option if you selected the Autostart timecode option when you created the archive (the archive rewinds the tape to find the start timecode).
 - Select the Manual Start option and enter the appropriate timecode if you selected this option when you created the archive.

EXIT Archive				
Clips				Delete
Device : VTR	Open	Open Using OTOC	Tape Size Estimate	Name
VTR : DigBeta	Open Read/Write	Verify on Write		Move
Format		Rewind On Close	Load Old D1	Copy
Engineering	Manual Start	01 00 00 00		Search

Manual Start option

Timecode fields

5. Do one of the following:

- To use the table of contents on the tape, click the Open button. Select Open Read/Write or Open Read Only from the Open Mode box.

NOTE: When you try to open a VTR archive in read/write mode, **flame** automatically checks for machine errors (for example, dirty heads). If any problem is detected, the archive will not open. If the tape is write-protected, the archive will open in read-only mode.

- To use the OTOC, click the Open Using OTOC button and select the OTOC in the file browser. You will be given the option to read slates from the tape. Answer 'Y' if you wish to see clip proxies.

To open a tape archive:

1. Select Tape from the Archive device box.
2. Select the tape device.
3. Insert the archive tape in the tape device.
4. Do one of the following:
 - To use the table of contents on the tape, click Open.
 - To use the online table of contents, click the Open Using OTOC button and select the online table of contents in the file browser. The OTOC does not contain slates hence archive proxies will show up as black.

To open a file archive:

1. Select File from the Archive Device box.
2. Select the file archive.
3. Do one of the following:
 - To use the table of contents in the archive, click Open.
 - To use the OTOC, click the Open Using OTOC button and select the OTOC in the file browser.

NOTE: Use the OTOC if the archive table of contents was corrupted or if you want to use a shortcut. For more information on recovering an archive, see "Recovering an Archive" on page 372.

Saving to an Archive

You can save entries from a clip library or from the desktop.

To save clips to an archive, you must open the archive using the table of contents on the medium (that is, using the Open button). For VTR archives, you can also save clips using the OTOC (with the Open Using OTOC button).

Archiving Entries from a Clip Library

Archiving entries directly from a clip library is the recommended method of archiving entries because it obliges you to organize your entries in a clip library before archiving. You can archive from the desktop, but this gives you less control over the organization of your entries before archiving.

When you save from a clip library, **flame** provides two methods for organizing the entries: Normal and Compact. The Compact method creates a backup set, which acts like a container that holds the archived material. It appears in the archive as an entry called Backup Set, and all selected entries are within it. Each time you use Compact, a new Backup Set is created. When entries are archived using the Normal option, the entries appear directly in the archive.

Both the Normal and Compact archiving methods perform some optimization to save space in the archive. The differences and advantages of the two methods are described.

Normal Archives

With Normal archives, if a frame appears more than once within a source clip (non soft-edit clip), the frame is archived only once. No image data for the duplicate frames is archived. This saves space on the archive, especially if your clips contain a lot of duplicate frames, such as still shots.

NOTE: If the same frame appears in two different source clips, the frame will be archived twice. Only duplicate frames within the same source clip are optimized on the archive.

When you create a Normal archive, each source clip is archived individually. This makes it easier and faster to restore individual clips. This guarantees that frames will be output to tape in play order, making the clips viewable on a VTR archive.

Compact Archives

Compact archives add a higher level of optimization during archiving than you would achieve using a Normal archive. With Compact archives, if a frame appears more than once in *any clip* being archived, the frame is archived only once. No image data for the duplicate frames is archived. This saves a significant amount of space on the archive, especially if your clips contain a lot of duplicate frames.

As mentioned previously, when you create a Compact archive, the archived material is stored in a Backup set. The Backup Set can be expanded in the archive library to view and select the clips

within it. You can restore selected clips from a Compact Backup Set without having to restore the entire archive.

NOTE: When you open an archive, all entries appear exactly as they did when you selected them, even if entire clips were composed of duplicate frames and not archived. The optimization has been done on the tape itself, and references to all original clips are kept so that the clips can be restored properly.

Additional Optimization

Both Normal and Compact archives add a second level of optimization by saving only one copy of clips. If a clip appears more than once in the archive, the clip is archived only once. Also, if you append to an archive, and a clip already exists in the archive, the clip is not added to the archive—that is, no image data for the duplicate clip is archived.

To archive entries from a clip library:

- 1. Open the archive if it is not already open (see “Opening an Archive” on page 362).
The available space in the archive appears.

EXIT Archive

Clips

Device : File

Load

Save

Remove

File : /usr/tmp/archive

Normal

Save From Library

Close

Delete

Name

Move

Copy

Search

4% full

Space for 2402 frames

Include Audio

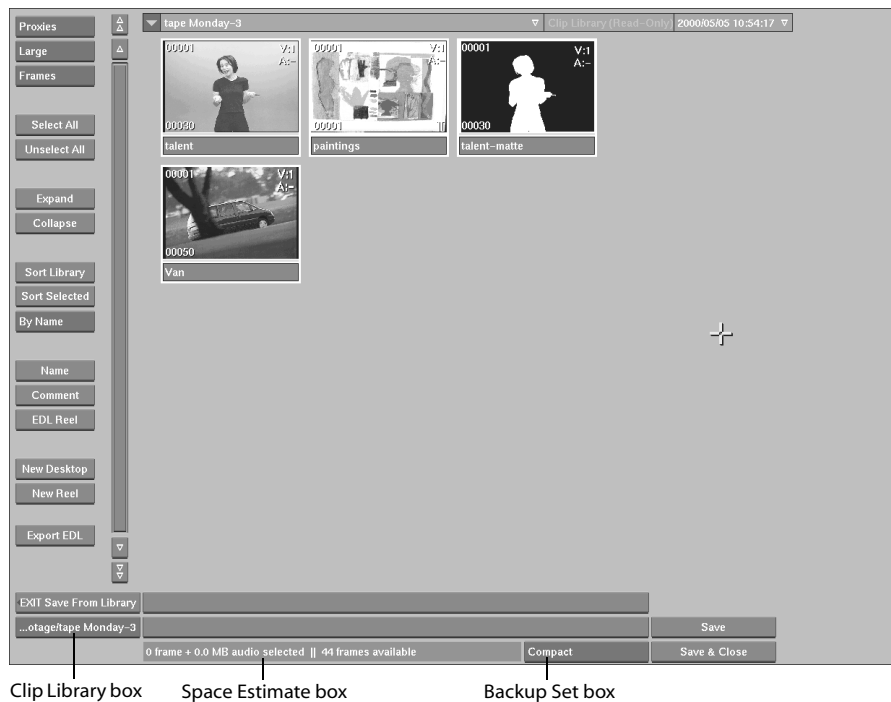
Available space

Include Audio button

- 2. Enable the Include Audio button if you want audio data to be archived with the clips.

NOTE: if you disable the Include Audio button upon archiving, when you restore the clips, the audio track information and the audio data will be lost.

- 3. Click the Save From Library button in the Archive menu.
The current clip library appears.



4. Select the required clip library from the Clip Library box.
5. Select the backup set from the Backup Set box.

Select: **To:**

- | | |
|---------|--|
| Compact | Maintain a highly organized archive, and save space on the device. |
| Normal | Archive without using a backup set. Entries are archived directly to the table of contents. Source clips remain in play order on tape. |

6. Select the entries that you want to archive.
Click an entry to select it. When an entry is selected, it is highlighted. For more information, see “Selecting Entries in a Clip Library” on page 212.

NOTE: If you deselect a clip that belongs to a soft edit, the soft edit is deselected as well. This prevents a soft edit from being archived without all of its source clips.

7. Click Save.
The selected entries are saved to the archive.

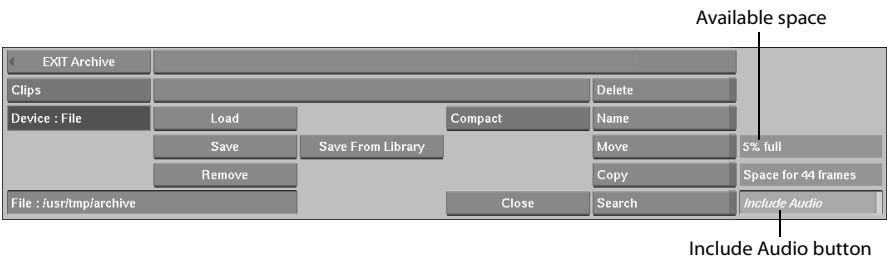
NOTE: To abort the archiving process, click anywhere in the grey area of the screen.

Archiving Entries from the Desktop

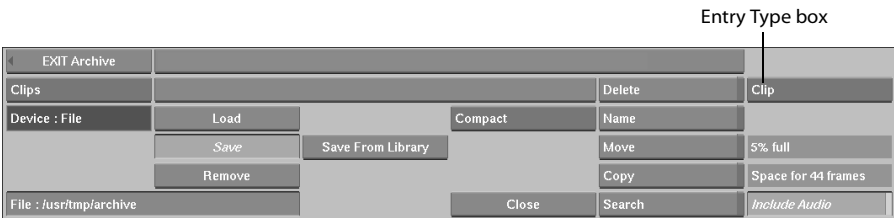
You can save a partition, clip library, desktop, reel, soft clip, or clip to the selected archive.

To save entries from the desktop:

- 1. Open the archive if it is not already opened.
The available space in the archive appears.



- 2. Enable the Include Audio button if you want audio data to be archived with the clips.
NOTE: if you disable the Include Audio button upon archiving, when you restore the clips, the audio track information and the audio data will be lost.
- 3. Click Save in the Archive menu.
The Entry Type box appears.



- 4. Select an option from the Entry Type box.

Select:	To:
Partition	Save the desktop and all clip libraries to the archive.
Clip Library	Save the current clip library to the archive. Before selecting this entry, make sure that you have chosen the appropriate clip library using the Clip Library box in the Archive menu.
Desktop	Save the desktop to the archive.
Reel	Save a work reel to the archive.
Clip	Save a clip or soft edit to the archive.

5. Select the clip or reel to be saved, or click Save:
 - If you are saving a partition, clip library, or desktop, a Save button automatically appears. Click Save to save the partition, clip library, or desktop. Click elsewhere to cancel.
 - If you are saving a reel, select a clip on the reel that you want to save.
 - If you are saving a clip, select the clip that you want to save.

The keyboard appears.

6. If the entry that you are saving already has a name, the name appears in the Keyboard field. Otherwise, use the keyboard to enter a name for the entry.
7. Click Enter.

If an entry of the same type and with the same name already exists in the archive, the Add, Replace, and Rename buttons appear. For more information, see “Adding, Replacing, and Renaming while Archiving” on page 368.

Otherwise, the partition, clip library, desktop, reel, or clip is archived under the specified name. If necessary, the name of the clip, reel, or desktop is changed on the desktop.

NOTE: flame checks for space availability on the selected medium before commencing the archive. If there is insufficient space, an error message appears and the process is terminated.

A time estimate appears once **flame** starts saving to the archive (for tape and VTR archives, the time estimate appears after the tape is rewound).

8. Repeat the procedure from step 4 for any other entries you want to save to the archive.
9. When you are finished, close the archive. For more information, see “Closing an Archive” on page 372.

Adding, Replacing, and Renaming while Archiving

If you try to save a clip that has the same name as a clip already in the archive, and the two clips are different, the Add, Replace, and Rename appear.



Click:	To:
Add	Add the entry to the current archive. Two entries will have the same name.
Replace	Replace all of the entries of the same type and with the same name as the partition, clip library, desktop, reel, or clip you are archiving.
Rename	Enter a different name for the entry you are archiving.

To cancel the operation, click anywhere else in the menu.

Loading Entries from an Archive

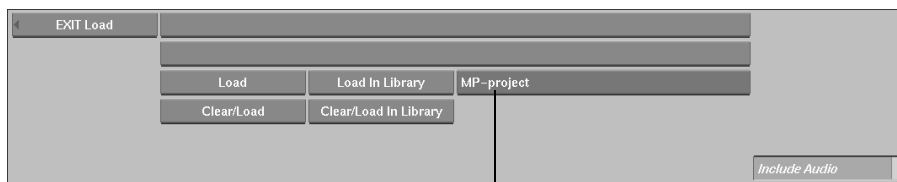
When you load entries from an archive, **flame** reads the table of contents from the medium. To speed up the process, click the Open Using OTOC (online table of contents). The MaxLibrarySize token in the *init.cfg* configuration file specifies the maximum size (in MB) that clip libraries can be. To successfully restore entries from an archive, the MaxLibrarySize token must allow the current clip library to grow sufficiently to accommodate the entries you are restoring. When you archive to a VTR, the clip library size required to restore the archive is recorded in the Header Info Slate. For information on modifying the MaxLibrarySize token, refer to the **flame** *Installation Guide*.

When you load a VTR archive, a confirmation is required before any action can be aborted.

To load entries from an archive:

1. Open the archive if it is not already opened.
2. Click Load in the Archive menu.
3. Select the destination.

The Load menu and the archive appear.



Clip Library box

4. Select the entries you want to load.
Click an entry to select it. When an entry is selected, it is highlighted.
5. Click Load.

The entries selected determine where the clips are loaded.

Entry	Destination
Partition	If you select a partition entry, its desktop and clip libraries are merged with the current partition.
Clip Library	If you select a clip library entry, its desktop, reel, and clip entries are merged with the clip library with the same name. If there is no matching clip library, a new clip library is created.
Desktop	If you select a desktop entry, its reels and clips are merged with the current desktop. Each desktop entry clip is loaded on the reel it was on when the desktop was saved.
Reel	If you select a reel entry, its clips are loaded on the destination reel.
Clip	If you select a clip entry, it is loaded on the destination reel.

Clear/Load

The Clear/Load, Load in Library, and the Clear/Load in Library provide different choices when you load entries from an archive.

The Clear/Load button deletes the destination before loading the selected entry. For example, if you are loading a reel from an archive, the destination reel is deleted before the archived reel is loaded.



WARNING: Since the Clear/Load button may delete a clip library, desktop, reel, or clip, use this button with caution.

To clear and load an entry:

1. Click Load in the Archive menu.
2. Select the destination.
3. Select the entry that you want to load.
4. Click the Clear/Load button, then Confirm.

The destination is replaced by the selected entry. For example, when loading a desktop, the current desktop is replaced by the new one.

Load in Library

The Load in Library button is similar to the Load button, except that the Load in Library button lets you to load entries directly into a clip library.

To load entries into a clip library:

1. Click Load in the Archive menu.
2. Select the destination.

3. Select the entries to load.

The Load menu and the archive appear.

4. Click the Clip Library box and select the clip library where you want to load the selected entries.

The “<new>” option is used to create a new clip library. For more information, see “Creating Clip Libraries” on page 218.

5. Click the Load in Library button.

The selected entries are loaded into the selected clip library.

Clear/Load in Library

The Clear/Load in Library button deletes the contents of the selected clip library before loading the selected entries. To use the Clear/Load in Library button, follow the same steps as for the Load in Library button in the previous procedure.



WARNING: Since the Clear/Load in Library command deletes the selected clip library, use this button with caution.

Loading a VTR Archive from flame 3.9.10

You should only use the Load Old D1 button to load an old archive created with **flame** 3.9.10. This button only appears if a Sirius, DIVO, or OctaneVideo card is installed and a VTR is the selected archive device.

Removing Entries from an Archive

Use the Remove button to remove the selected entries or use the Keep button to keep the selected entries and remove the unselected ones.

NOTE: Because of physical restrictions in the way information is stored on Exabyte, DAT, and other tape drives, you cannot remove entries from a tape archive.

To remove entries:

1. Click the Remove button in the Archive menu.

The Remove menu and the archive appear.

Remove menu



2. Select the entries to remove.

Click an entry to select it. When an entry is selected, it is highlighted.

3. Click Remove and then Confirm.
The selected entries are removed from the archive.
4. Click Exit Remove to return to the Archive menu.

To keep entries:

1. Click Remove in the Archive menu.
The Remove menu and the archive appear.
2. Select the entries to keep.
Click an entry to select it. When an entry is selected, it is highlighted.
3. Click Keep and then Confirm.
The selected entries are kept and the unselected entries are removed from the archive.
4. Click Exit Remove to return to the Archive menu.

Closing an Archive

Click the Close button to update the table of contents and close the device. The Close button appears in the Archive menu once an archive is open. Before you can eject a tape from a tape drive, you must first click the Close button.

If you are archiving to a VTR, you can automatically rewind to the beginning of the tape when you exit the Archive menu. To do this, enable the Rewind on Close button. By default, this button is disabled.

NOTE: If you attempt to exit the Archive menu, you are prompted to close the device. Click Confirm to close the device, or elsewhere to cancel.

Recovering an Archive

If the table of contents of an archive on tape gets corrupted, **flame** may not be able to read it. If this happens, open the archive using the OTOC. OTOCs have been available for archives since version 4.5 and higher.

Once you open the archive, you can restore the entire contents of the archive and save it to a new tape. Do this, for example, if the tape has been damaged.

For VTR archives, you can also overwrite the corrupted table of contents by saving a single frame to the archive. If the table of contents is successfully overwritten, you can open the archive using the Open button.

To restore the archive:

1. Click Open to open the archive. If the table of contents is corrupted, the following message appears:

CLIP MGT: Warning! Cannot read archive header. Select online TOC?

2. Click Confirm to open the OTOC.
The file browser appears.
3. Select the OTOC for the archive you are trying to open and click Confirm.
4. If the archive is on a VTR, a prompt asks if you wish to retrieve the slates (proxies). Press Y to see proxies in the archive or N to have no proxy images.
After a moment, the archive appears.

NOTE: When you open an archive with the OTOC, there are no proxy images (unless the archive is on a VTR, and you answered Yes to the prompt). Proxies appear as black images.

5. Select the entire contents of the archive. Make sure you have sufficient space on the framestore for the contents of the archive.
6. Click Restore.
7. When the restore is complete, click Close.
The restored clips appear in the selected clip library.
8. Re-archive the material onto a new tape as described in “Archiving Entries from a Clip Library” on page 364.

To overwrite a corrupted table of contents (VTR only):

1. Click Open to open the archive. If the table of contents is corrupted, the following message appears:

CLIP MGT: Warning! Cannot read archive header. Select online TOC?

2. Click Confirm to open the OTOC.
The file browser appears.
3. Select the OTOC for the archive you are trying to open and click Confirm.
4. A prompt asks if you wish to retrieve the slates (proxies). Press Y to see proxies in the archive or N to have no proxy images.
After a moment, the archive appears.

NOTE: When you open an archive with the OTOC, there are no proxy images (unless you answered Yes to the prompt described above). Proxies appear as black images.

5. Select a clip library from the Library Type box.

6. Select any single-frame clip and click the Archive button.
7. Click Close to close the archive.
8. Click Open to open the archive.

If the archive will still not open using the table of contents on the tape, restore the entire archive following the procedure in “Loading Entries from an Archive” on page 369.

Using an ASCII TOC to View an Archive

Open the ASCII version of the table of contents using *jot* to view the contents of an archive without opening it. ASCII TOC file names have the following format:

```
<archive name>_<creation date>.atoc.
```

By default, they are stored in the */archive* directory for your project. The contents of the ASCII files are labelled as follows:

Code:	Meaning:
P	Partition
L	Clip library
D	Desktop
R	Reel
C	Clip / soft-edit
E	Source clip

Deleting Tables of Contents

So that you do not lose data, **flame** never erases the tables of contents it makes when you create archives. Therefore, you must delete the tables of contents that are no longer useful, for example, after you have deleted an unneeded archive. **flame** uses the following name formats for the tables of contents:

Type	Format
Online	<archive_name>_<creation date>.otoc
ASCII	<archive_name>_<creation date>.atoc

The archive name is the name that you type in the Name field when you create the archive.

To delete tables of contents:

1. Make a backup copy of the tables of contents you want to delete on an Exabyte tape or by using your usual method for backing up UNIX files.
2. Use the **cd** command to go to the directory where the tables of contents are stored. By default, it is the */usr/discreet/archive* directory.
3. Use the **rm** command to remove the unwanted tables of contents.

Archiving Setups

You can archive and restore setups that have been saved in any **flame** module. For information on saving setups, see Chapter 7, “Saving Setups and Preferences.” When you save setups as you work in different modules, they are saved by default within the current project. When you archive setups, they are archived by project. Before archiving setups, make sure you have:

- Defined your own project. For more information on defining a project, see Chapter 3, “Project Management.”
- Achieved the desired effect(s) in the module menus.
- Saved the appropriate setups.

NOTE: Archives for setups can be file-based or you can use UNIX tape devices such as DAT or Exabyte.

Archive setups using the Archive Setups menu.

To open the Archive Setups menu:

1. Select Setups in the Archive Type box.
The Archive Setups menu appears.

Archive Type box

EXIT Archive	
Setups	Delete
Load	Name
Save	Move
	Copy
File : /usr/tmp/setups.tar	Search

File Name field

To archive a setup:

1. The File Name field contains the default pathname for the archive file, as specified in the SETUPARCHIVETAPE KEYWORD in the *init.cfg* configuration file. If you want to change the pathname, click the File Name field.
The file browser and on-screen keyboard appear.

2. If needed, select another directory using the file browser.
3. Type a new file name using the on-screen keyboard, and click Enter.
The file browser closes and the Archive Setups menu reappears. The new pathname appears in the File Name field.
4. Click Save, then click Confirm.
An archive file for your current project setups is created. It includes all setups that you saved in your module menus.
5. Click EXIT Archive.

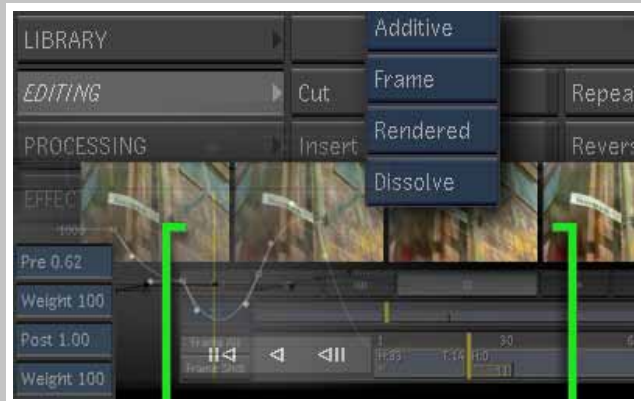
To restore a project's setups:

1. Open the Archive Setups menu (see “To open the Archive Setups menu:” on page 375).
2. The File Name field displays the last saved archive. If you want to select a different archive, click the File Name field.
The file browser appears.
3. If needed, select another directory using the file browser.
4. Select the archive you want to restore by clicking on it.
The file browser closes and the Archive Setups menu reappears.
5. Click Load button, then click Confirm.
The archive file containing your project setups is restored into the current project, that is, all the setups you saved in your module menus are loaded into the appropriate modules.

Tips for Better Archiving

When archiving, keep the following points in mind to obtain best results:

- Normal archives do not archive duplicate frames if they appear in the same clip.
- Compact archives do not archive any duplicate frames.
- You can restore an individual clip from a Normal or Compact archive.



Section 3: Editing

*Gestural,
picture-based
editing for
easy viewing
and intuitive
operation.*

19

Overview: The Editing Menu

The “cut” and dry of editing

Many editing operations are available to create timewarps and dissolves, and to trim and manipulate your clips. You can also perform editing operations on audio, such as fade-ins.

Summary

In this chapter, you learn about:

- “Editing Options” on page 380
- “The Editing Menu” on page 380
- “Gestural Editing” on page 382
- “Editing Video and Audio Simultaneously” on page 383
- “The EditReel” on page 386
- “Soft Edits” on page 389
- “Editing Timecodes” on page 390
- “Consolidating and Committing Soft Clips” on page 391
- “The Force Render Command” on page 392

Terminology

The following terms are used to refer to clips and editing commands:

- A *source clip* is either a clip loaded from a VTR, or a new clip created in **flame**.
- The *target clip* is the clip on the desktop that receives a source clip during an edit. For example, when splicing Clip A onto Clip B, Clip A is the source clip and Clip B is the target clip.
- A *soft edit* is an uncommitted edit that “remembers” the processes and the source clips involved in its creation.
- A *cut* is an edit created with the Cut command. The Cut command divides a clip into two smaller clips, or *cutouts*.
- A *timewarp* is a soft edit that changes the timing of a clip. Timewarps are created with the Timewarp command.

- A *transition* joins two clips to make a soft edit. Transitions are created with the Splice, Dissolve, Insert, and Replace commands.
- A *transition handle* is the yellow line that marks a splice or either of two green brackets that mark the beginning and end of a dissolve.
- A *shot* is the part of a soft edit between two transitions, or between a transition and the start or end of an edit.
- An *outgoing* clip and an *incoming* clip are the fade-out and fade-in clips in a dissolve.

Editing Options

flame gives you several different editing options:

- The traditional menu-based option is used to perform basic editing functions from the buttons on the Editing menu.
- The cursor-based drag and drop “gestural” option allows you to perform some basic desktop editing functions without using the Editing menu buttons. For more information, see “Gestural Editing” on page 382.
- The EditReel is used to perform advanced editing functions such as trimming, transition editing, and timewarp editing. For more information, see “The EditReel” on page 386.

The Editing Menu

To display the Editing menu, click the Editing button in the Main menu. Several operations that are performed with Editing menu items can also be performed using other editing options. All editing operations apply to audio unless otherwise noted.

EDITING ▶	Cut	Repeat	Dissolve	Delete
PROCESSING ▶	Insert	Reverse	Audio TimeStretch	Name
EFFECTS ▶	Splice	Swap Shot	TimeWarp	Move
FORMAT ▶	Replace	Consolidate	Commit	Copy
SYSTEM ▶	Match Source	EDL	Force Render	Search

Cut — Divides a single clip into two or more clips. Select a frame in the source clip and the clip is cut after the selected frame. See “Cutting Clips” on page 396.

Insert — Inserts a partial or complete source clip after a frame in the target clip. The Insert command creates a soft edit. See “Inserting Clips Gesturally” on page 405.

Splice — Splices two or more source clips together. Select the frames in the two source clips where the splice is made. The Splice command creates a soft edit. See “Splicing Clips” on page 400.

Replace — Replaces frames in the target clip with a partial or complete source clip. The Replace command creates a soft edit. See “Replacing Frames and Clips” on page 406.

Match Source — Finds matching frames in two source clips or matches a soft edit with its source clip. See “Matching a Soft Edit with Its Source Clips” on page 408.

Repeat — Repeats a frame, a clip, or the frames within a clip any number of times. The repeated frames or clips are joined end to end to make a new clip. See “Repeating Frames in a Clip” on page 410.

Reverse — Reverses the order of the frames in a clip. See “Reversing Frame Order in a Clip” on page 411.

Swap Shot — Replaces a shot between two transitions, or between a transition and the start or end of a soft edit. See “Swapping Shots” on page 412.

Consolidate — Sets an upper limit on soft clip handles (heads and tails). You can consolidate an element, a single clip, a range, a reel, or the entire desktop.

NOTE: Consolidate is not applied to audio.

EDL — Automatically assembles the clips in an Edit Decision List (EDL). See Chapter 14, “EDLs.”

Dissolve — Fades between two source clips (dissolve or crossfade), or one source clip and a black source (fade-in or fade-out). You can also mix clips together. The Dissolve command creates a soft edit. See “The Dissolve Command” on page 422. Dissolves can also be applied to audio.

Audio TimeStretch — Creates a timewarp for your audio. See “Audio Timewarps” on page 481.

TimeWarp — Changes the overall timing in a clip. You can create complex timing changes or add an image trail with the Timing Editor menu. The Timewarp command creates a soft edit. See “Soft Timewarps” on page 390.

NOTE: Timewarp is not applied to audio. You need to use Audio Timestretch to timewarp audio.

Commit — Commits a soft edit. See “Consolidating and Committing Soft Clips” on page 391.

Force Render — Renders a clip again, all the clips on a reel, or all the clips on the desktop. See “The Force Render Command” on page 392.

NOTE: Force Render is not applied to audio.

Delete — Deletes a single frame from a clip, a clip from a reel, all the clips on a reel, or the entire desktop. You can also delete an audio track from a clip. See “Deleting Clips” on page 414.

Name — Changes the name of a clip. See “Naming and Renaming Clips” on page 416.

Move — Moves a single frame, a clip, or all clips on a reel to another reel. You can also move an audio track from one clip to another. See “Moving Clips” on page 416.

Copy — Makes a copy of a single frame, a clip, or all clips on a reel. You can also copy audio tracks between clips. See “Copying Clips” on page 418.

Search — Searches for clips on the desktop or in a clip library. See “Searching for Clips” on page 46.

Undo — Reverses the last command. See “Searching for Clips” on page 46.

Gestural Editing


Gestural editing is a technique in which you use your mouse or stylus to move clips among reels on the desktop, and to perform cut, copy, insert, and replace edits. For details on how to perform these tasks, refer to Chapter 20, “Basic Editing.”




Gestural Editing Cursors

When you drag a source clip over a target clip, the cursor changes as the source clip moves over different areas of the target clip. The colour and type of cursor indicates the type of edit that will occur if you drop the source clip on that spot.

A green cursor is used for insert (non-destructive) edits. A red cursor is used for replace (destructive) edits.

To perform a gestural insert or replace edit, drag the source clip over the frame in the target clip where you want to make the edit. The insertion point is the space after the selected frame. When you see the appropriate edit cursor, drop the source clip. The different edit cursors are described as follows.

Cursor:	Appears when:	Use to:
 Insert (Green)	The source clip is positioned between frames on the target clip.	Perform an insert edit.

Cursor:	Appears when:	Use to:
 Replace (Red)	The source clip is grabbed at any frame other than its first or last frame, and is positioned over a frame on the target clip.	Perform a replace edit.
 Replace ahead	The source clip is grabbed at its first frame and is positioned directly over a frame on the target clip.	Perform a replace edit starting at the frame under the cursor and moving in the direction of the arrow.
 Replace back	The source clip is grabbed at its last frame and is positioned directly over a frame on the target clip.	Perform a replace edit starting at the frame under the cursor and moving in the direction of the arrow.

Enabling Gestural Editing

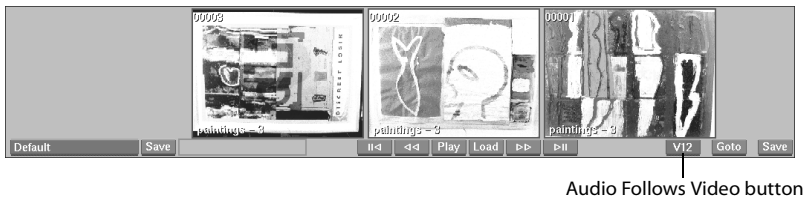
To enable gestural editing, choose Gestural Editing from the Editing option box in the System Preferences menu. For more information, see “Gestural Editing Options” on page 119.

NOTE: Gestural editing can be applied to audio depending on the status of the Audio Follows Video option in the System Preferences menu. For more information, see “Editing Video and Audio Simultaneously” on page 383.

Editing Video and Audio Simultaneously

If you are gesturally editing a clip that has audio, you can specify if the audio is affected by the edit you are performing. Gestural editing of audio is enabled or disabled with the Audio Follows Video button beside the Goto button.

NOTE: You can also enable or disable the Audio Follows Video option in the System Preferences menu.



HINT: Hold the **A** hotkey to temporarily change the status of the Audio Follows Video button for the current gestural edit.

When V12 is enabled, the video and both audio tracks are edited simultaneously. When V is enabled, only video is affected during the gestural edit.

If you access an editing operation from the Editing menu, a Video/Audio option button appears.



Video and Audio button

When Video and Audio is enabled, the video and both audio tracks are edited simultaneously. When Video is enabled, only video is affected during the edit.

The following sections explain the effect of enabling or disabling editing audio simultaneously with video for both gestural and non-gestural editing.

Gesturally Editing Audio

Use the Audio Follows Video button in the System Preferences menu to specify how audio is edited when you modify video elements. When the Audio Follows Video button is enabled, edits such as dissolves are applied to both video and audio tracks. When the Audio Follows Video button is disabled, edits such as dissolves are applied to video elements only. The following table explains which editing functions are affected by the status of the Audio Follows Video button.

Function:	Audio Follows Video enabled:	Audio Follows Video disabled:
Cut	Cuts are placed in audio tracks at the same location as the video track, and same portions are removed.	Audio tracks are not cut and the complete audio tracks appear on both parts of the clip.
Splice	Splice points are created in audio tracks at the same place as in the video track. If the source clip does not contain audio, the target clip's audio is overwritten with silence.	Audio in the target clip is unaffected, audio on the source clip is ignored.
Insert	Splice points are created in audio tracks at the same place as in the video track. If the source clip does not contain audio, the target clip's audio is overwritten with silence.	Audio in the target clip is unaffected, audio on the source clip is ignored.
Replace	Splice points are created in audio tracks at the same place as in the video track. If the source clip does not contain audio, the target clip's audio is overwritten with silence.	Audio in the target clip is unaffected, audio on the source clip is ignored.

Editing Audio with the Editing Commands

When you edit a clip with audio by using a command from the Editing menu, a Video/Audio button appears. Use this button to specify how audio is edited when you modify video elements. When the Video option is enabled, edits such as dissolves are applied to both video elements only. When the Video & Audio option is enabled, edits such as dissolves are applied to video and audio tracks. The following table explains which editing functions are affected by the status of the Video/Audio button.

Function:	Video & Audio option is enabled:	Video option is enabled:
Cut	Audio tracks are cut at the same location as the video.	Audio tracks and timing remain intact on both parts of the clip.
Insert	Insertion is performed in all tracks. If the source clip does not contain audio, the target clip's audio is overwritten with silence.	Audio in the target clip is unaffected, audio in the source clip is ignored.
Splice	Splice points are created in audio tracks at the same place as in the video track. If the source clip does not contain audio, the target clip's audio is overwritten with silence.	Audio in the outgoing clip is kept.
Replace	Splice points are created in audio tracks at the same place as in the video track. If the source clip does not contain audio, the target clip's audio is overwritten with silence.	Audio in the target clip is unaffected, audio in the source clip is ignored.
Swap Shot	The portion of audio intersecting the shot being swapped in is carried over.	Audio in the target clip is unaffected, audio in the source clip is ignored.
Dissolve	Audio in the incoming and outgoing clips is dissolved in sync with video.	Audio in the outgoing clip is unaffected, audio in the incoming clip is ignored.
Copy (Range/Frame)	Only audio in the selected range is copied.	The complete audio tracks are copied regardless of the selected range.
Move (Range/Frame)	Only audio in the selected range is moved. Cuts are inserted on the source clip for video and audio tracks at the frames delimited by the range selection.	The complete audio tracks are moved, regardless of the selected range. The source clip is unaffected.
Delete (Range/Frame)	Only audio in the selected range is deleted.	All audio is deleted, regardless of the selected range.

The EditReel

The EditReel is accessed through the Player. For information about the Player, see “The Player” on page 77.



The EditReel menu contains the EditReel, a timeline, and items that are used for advanced editing, such as trimming, dissolve curve editing, and timewarp editing. Refer to the appropriate chapter in this user’s guide for details.

NOTE: The reel direction of the EditReel is left-to-right.

The EditReel has a timeline view for working with video and audio soft edits as segments. Every soft edit appears as a different segment on the timeline. The timeline supports one video track and two audio tracks.

To use the timeline:

1. In the Player, select EditReel.
The EditReel appears.
2. Select an option from the EditReel view box:

Select:	To:
Wave	View the waveforms for the audio in the EditReel.
EditWave	View a timeline for the video and audio tracks. Video and audio elements appear as segments in the EditReel. Audio waveforms appear within each audio segment. Soft edits appear as multiple segments in the timeline.
Editing	View a timeline for the video and audio tracks. Video and audio elements appear as segments in the EditReel. No waveforms appear for the audio. Soft edits appear as multiple segments in the timeline.
No Waves	View a timeline for the video tracks only.

NOTE: When you use Editing and EditWave mode, use the Frame All button to toggle between “show all video elements” mode and “show all video and audio elements” mode.

Editing Audio in the EditReel

You can edit video and its associated audio at the same time to avoid putting them out of sync. Use the Link Audio button to simultaneously edit video and audio while trimming, slipping, sliding, or changing the type of transition (for example, changing a splice to a dissolve or a dissolve to a splice).

To edit video and audio simultaneously:

1. View the timeline for the clip.
2. Move the positioner over the video element you want to edit.

A yellow border appears around the current video element.

NOTE: If you centre the positioner over a transition, both outgoing and incoming video elements have a yellow border.

3. Enable the Link Audio button.

A yellow border appears around the audio elements that are now linked to the video element.

4. Edit the video element.

The same edit is applied to both video and audio tracks. For example, if Link Audio is enabled and you trim the clip by 5 frames, the video and audio tracks are trimmed by 5 frames.



WARNING: Because you cannot edit audio independently of video, you may apply an edit to a clip that cannot be reversed with the editing commands. If this occurs, use the Undo button to undo the last action, the Reset button to reset the entire clip, or the Reset Audio button to reset only the audio tracks.

When you turn the EditReel off and view an audio clip, the waveform for the audio head and tail frames appear dark grey. You do not hear audio head and tail frames during playback because they are not part of the clip. To hear the complete audio clip, enable Play Audio Only.

Video Rendering in the EditReel

When you edit a blending transition in the EditReel, it can be either rendered or unrendered. The Rendered/Unrendered box that appears for transitions in the EditReel is a global setting and does not apply on a segment by segment basis. The Rendered or Unrendered setting applies to all transitions when they are centred in the EditReel. For example, if you select Rendered for one transition, that setting applies every time you move the positioner over a transition.

If you select Unrendered and then center a transition in the EditReel that has not yet been rendered, a Render V. button appears in the lower left corner of the EditReel. Click the Render V. button to render the currently selected (centred) video element.

You can also click the Render All button to render the entire clip in the EditReel.

Audio Rendering in the EditReel

If you trim audio in the EditReel, the waveform needs to be rebuilt. You can manually rebuild the waveforms, or have them rebuilt automatically.

When you trim audio, a Render Au. button appears beneath the Link Audio button. Click the Render Au. button to manually rebuild the waveforms. This method is recommended for long clips that may take a while to render and you do not want rendered automatically every time you perform a trim operation.

NOTE: Playback only plays the most recently rendered version of the audio. If you want to play back your latest changes, you must click the Render Au. button to render the audio.

To automatically rebuild the waveform after every trim operation, enable the Auto Load button beneath the Audio Lib button.

Virtual Sources in the EditReel

Virtual sources are sources that can be trimmed indefinitely without creating new frames. A coloured frame is an example of a virtual source. Virtual sources are indicated by an asterisk in the Head and Tail information portion of the element instead of a number (H* or T*).

The gap between two audio elements in a clip is also a virtual source. Gaps can be trimmed indefinitely because they represent silence. Gaps between audio elements also have an asterisk in the Head and Tail information on the element.

Playing Clips in the EditReel

- If you are using SGI audio, clips containing soft edits can be played; a hard-committed copy of the original will be used (thus leaving the original untouched).

NOTE: If you modify the clip in the EditReel, the audio is automatically re-committed.

- If you are using Discreet Audio, clips containing soft edits can be played directly (**flame**/**inferno** only).

EditReel Setups

Select Setup in the Player to see the EditReel Setup menu. Use the EditReel Setup menu to set preferences for working in the EditReel. For more information on this and other Player options, see “Player Setup Options” on page 78.

Soft Edits

The Splice, Dissolve, Insert, Replace, and Timewarp commands, as well as all gestural edits, generate soft edits. A soft edit is an uncommitted modifiable edit. All soft edits are marked by a “[” label on the proxy.

NOTE: Clips brought in from an Editing product are considered soft edits and have the “[” label.

For example, if you use the Dissolve command to dissolve between two source clips, the result is a soft edit with green brackets that mark the beginning and end of the dissolve. You can modify the soft edit by sliding the location of the dissolve, changing the length of the dissolve, and so on.

You can modify a soft edit because it “remembers” the steps and source clips involved in its creation. There are three kinds of soft edits: soft transitions, soft timewarps, and soft cutouts.

To use soft edits, you must enable the Soft Edits button in the System Preferences menu.

Soft Cuts

There is a Soft Cuts button in the System Preferences menu. You can only use Soft Cuts when the Soft Edit button is enabled. Use the Soft Cuts button in conjunction with the Soft Edits button, as follows:

Status:	Result:
Soft Cuts enabled	Handles (head and tail frames) are kept when clips are cut.
Soft Cuts disabled	Handles are lost when clips are cut.

Soft Transitions

Soft transitions are created with the Splice, Insert, Replace, and Dissolve commands. They are characterized by yellow transition handles (splice, insert, and replace) that mark where the two clips have been joined, or green brackets (dissolve) that delimit the frames in a transition. A label within parentheses at the bottom of a frame indicates if it is part of a dissolve.

Label:	Description:
“D(<i>n</i> / <i>m</i>)”	Describes a dissolve where <i>n</i> is the frame’s position in the dissolve and <i>m</i> is the length of the dissolve. For example, for a 10-frame dissolve, the first frame is labelled D(1/10) and the last, D(10/10).

Keep/Remove Sources Option Box

When you use the Splice or Dissolve command, the Keep/Remove Sources option box enables you to keep or remove the source clips after creating the transition.

Select:	To:
Keep Sources	Keep source clips when using the Splice and Dissolve commands. This default mode is reset between flame sessions.
Remove Sources	Remove source clips from the desktop when using the Splice and Dissolve commands to keep your desktop from becoming cluttered.

Soft Timewarps

Soft timewarps are created with the TimeWarp command. All frames in a timewarped or reversed soft edit are identified with the “TW” label.

Soft Cutouts

Soft cutouts are created by cutting a soft edit, and are identified by a “[]” label. Soft cutouts can be modified. A soft cutout still “remembers” its source clip. To view its source clip, use the Match command in Source Clip mode. See “Matching a Soft Edit with Its Source Clips” on page 408.

NOTE: You should consolidate a soft cutout before saving it to the clip library or an archive to avoid saving the soft cutout’s source clip. For example, an unconsolidated 5-frame cutout linked to a 500-frame source clip will be archived as a total of 505 frames. Consolidating a cutout removes the link to its source clip and archives the 5-frame cutout and specified handles only.

Editing Timecodes

While viewing clips on the desktop, you can view two timecodes for each clip by selecting Src and Rec Timecode from the Frame ID display in the System menu. For more information, see Chapter 8, “System Utilities and Preferences.”

The source timecode (left) is always the timecode for the source clip used in the soft edit. A source clip is either loaded from a VTR or created in **flame**. Source clips loaded from a VTR start with the timecode from the VTR cassette. Source clips created in **flame** begin at timecode 00:00:00:00.

The record timecode (right) is the timecode of the soft edit. When you use an editing command that creates a soft edit, the record timecode shows a new timecode renumbered starting at 00:00:00:00.

NOTE: The source and record timecodes for source can be changed using the Change Timecode option.

Consolidating and Committing Soft Clips

Use the Consolidate and Commit commands to convert a soft clip to a source clip. This removes the clip's soft properties, meaning it no longer "remembers" the source clips used to create it.

If you Consolidate the clip, you can specify a number of handles to keep at the head and tail of the clip. If you Commit the clip, you do not keep any handles. Keeping handles is useful if you want to modify the clip at a later date (for example, you want to trim or slip the shot).

You consolidate or commit a soft clip when you archive the clip or when you are satisfied with its results. Use the Consolidate or Commit commands to consolidate or commit:

- All the soft edits in a clip.
- All the soft edits on a reel.
- All the soft edits on the desktop.
- All the soft edits in a range that you select.



WARNING: Once a soft edit is committed, its links to source clips cannot be re-established.

To use the Consolidate command:

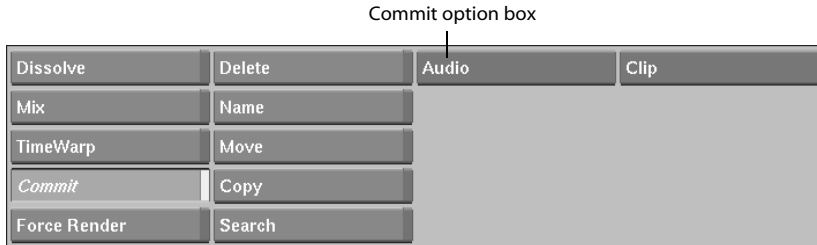
1. Click the Consolidate button in the Editing menu.
2. The Consolidate option box appears.
3. Enter the number of handles to keep in the Handles box.
4. Select the item you want to consolidate.

Item:	Selection method:
Clip	Click the clip with the red arrow cursor.
Reel	Click anywhere on the reel.
Desktop	Click the confirm button.
Element	Click on the frame immediately before the element.
Range	Click at the start of the range with the red arrow cursor, and at the end of the range with the green arrow cursor.

To use the Commit command:

1. Click the Commit button in the Editing menu.

The Commit option box appears.



2. Select the type of item you want to commit from the Commit option box.
3. Select the item you want to commit.

Item: Selection method:

Clip Click the clip with the red arrow cursor.

Reel Click anywhere on the reel.

Desktop Click the confirm button.

Element Click on the frame immediately before the element.

Range Click at the start of the range with the red arrow cursor, and at the end of the range with the green arrow cursor.

NOTE: If you select Audio, an Audio Commit option box appears. Select the item using the same methods in the above table.

4. If elements such as timewarps or dissolves have not been rendered, they are automatically rendered before the commit.

The Force Render Command

Use the Force Render command to render:

- A range of transitions or timewarps in a soft edit
- A single transition or timewarp in a soft edit
- An entire soft edit
- All the soft edits on a reel
- All the soft edits on the desktop

A soft edit may need to be rendered if it is the result of an aborted timewarp or dissolve. Soft edits that need to be rendered again contain one or more frames displaying the message

“Unrendered Frame”. You can also force render a dissolve created in Do Not Render mode. For more information, see “Render Mode Options” on page 423.

Force Rendering a Range of Frames

Use the Range mode to force render all transitions or timewarps within a specified range of frames.

To force render a range of frames:

1. Click the Force Render button in the Editing menu.

The Force Render option box appears.



2. Select Range from the Force Render option box.
3. Select the starting frame in the soft edit.
4. Select the ending frame in the soft edit.

The transitions or timewarps within or which cross the selected range of frames are force rendered.

Force Rendering a Single Dissolve or Timewarp

Use the Element option to force render a single dissolve or timewarp in a soft edit.

To force render a single dissolve or timewarp:

1. Click the Force Render button in the Editing menu.

The Force Render option box appears.

2. Select Element from the Force Render option box.
3. Select a frame within the dissolve or timewarp that you want to force render.

The selected dissolve or timewarp is force rendered.

Force Rendering Entire Soft Edits

Use the Clip, Reel, and Desktop options to force render an entire soft edit, all soft edits on a reel, or all soft edits on the desktop.

To force render entire soft edits:

1. Click the Force Render button in the Editing menu.
The Force Render option box appears.
2. Select an option from the Force Render option box.
3. Select a clip, a clip on a reel, or any clip on the desktop.

The basic editing tools can help you assemble the final cuts for your sequences.

Summary

In this chapter, you learn about:

- “Editing Collapsed Clips” on page 396
- “Cutting Clips” on page 396
- “Splicing Clips” on page 400
- “Inserting Frames and Clips” on page 403
- “Replacing Frames and Clips” on page 406
- “Gestural Splice, Insert and Replace Edits” on page 408
- “Matching a Soft Edit with Its Source Clips” on page 408
- “Repeating Frames in a Clip” on page 410
- “Reversing Frame Order in a Clip” on page 411
- “Swapping Shots” on page 412
- “Consolidating Clips” on page 414
- “Deleting Clips” on page 414
- “Naming and Renaming Clips” on page 416
- “Moving Clips” on page 416
- “Copying Clips” on page 418

About Basic Editing

This chapter describes basic editing techniques and functions. Some editing functions, such as Splice, Insert and Replace, create soft transitions, which are easy to modify. For more information on soft transitions, see “Soft Edits” on page 389.



Some of the basic editing techniques can be accomplished by both the traditional menu-based method, and by the “drag and drop” gestural method. In such cases, both methods are described, and the gestural method is marked with the “gestural editing” icon in the margin, as seen here on the left.

Editing Collapsed Clips

The editing procedures in this chapter describe basic editing done on uncollapsed clips. Most of the same operations can be done on collapsed clips. The only difference is that you must select your in-point, out-point, and target frames by scrubbing through the collapsed clip until the desired frame is the one that is visible at the top of the collapsed clip proxy.

For more information about collapsed clips, see “Collapsing Clips” on page 39.

Cutting Clips

The Cut command divides one clip into two (or more) smaller cuts. There are three cut modes to choose from. Choose the cut mode in the Cut Mode option box, which appears when the Cut button is pressed.

Choose:

After selected frame

At splices

Every Nth frame

To:

Cut the clip after the frame that you select when in Cut mode.

Cut the clip at every soft splice that occurs on the clip.

Cut the clip at the intervals that you specify in the numeric field that appears when you select this option.

To cut a clip after the selected frame:

1. Click the Cut button in the Editing menu.

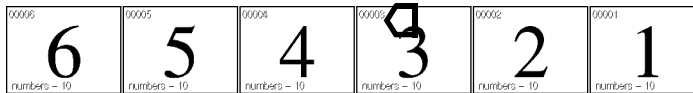
The Cut menu appears.



Cut Mode option box

2. Choose “After selected frame” in the Cut Mode option box.
3. Select the frame before the place where you want to make the cut in the source clip.

The clip is divided into two cuts after the selected frame. The frames in the second cutout are renumbered starting at frame number 00001. For more information, see “Soft Cutouts” on page 390.



Action: Select the source clip



Result: The clip is cut between the third and fourth frames

To cut a clip at every Nth frame:

1. Click the Cut button in the Editing menu.
2. Choose “Every Nth Frame” in the Cut Mode option box.
3. Enter the number of frames to be contained in each cut in the Frame Number field.
4. Select the source clip.

The clip is cut into segments of the selected number of frames.

To cut a clip at its splices:

1. Click the cut button in the Editing menu.
2. Choose “At splices” in the Cut Mode option box.
3. Select the source clip.

The clip is cut into segments at every soft splice point. If the clip contained no soft splices, no cuts are made.

Cutting Dissolves and Timewarps

Performing a cut on an uncommitted dissolve or timewarp divides the soft edit into two soft cuts. See “Soft Cutouts” on page 390.

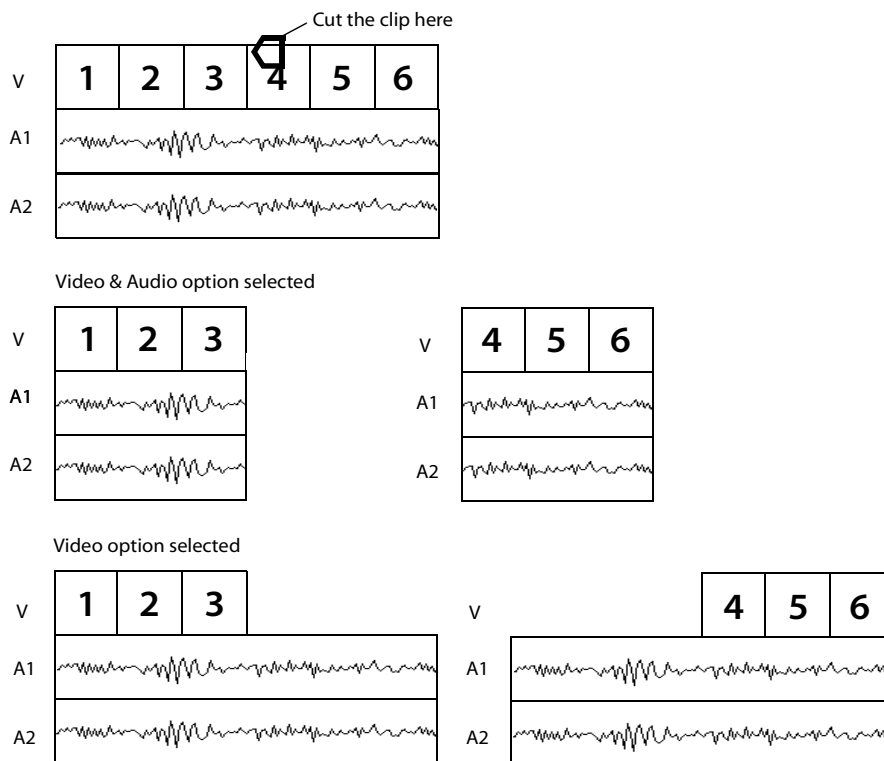
If the cut is performed on a frame within a dissolve, the dissolve is committed before the cut is made. The two resulting cuts are soft edits linked to the original source clips.

If the cut is performed on a frame within a timewarp, two clips with unrendered timewarps are created.

NOTE: You should consolidate a soft cutout before saving it to the Clip Library or an archive to avoid saving the soft cutout’s source clip.

Cutting Clips with Audio Tracks

If you cut a clip with audio, use the Video/Audio option box to specify how the audio is affected. When you use the Video & Audio option, the audio tracks are cut at the same location as the video. When you use the Video option, the audio tracks are not cut and the entire audio tracks are included on both new clips.

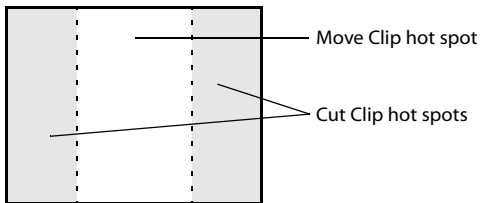




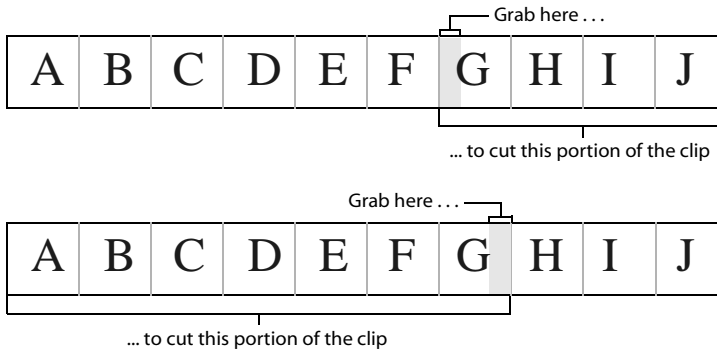
Cutting Clips Gesturally

When gestural editing is enabled, each frame in a clip contains two types of “hot spots”. The Move Clip hot spot is the part of the clip that is used to move clips around on the desktop. The Cut Clip hot spots are the parts of the clips that are used to cut the clip. The cursor changes to indicate the type of hot spot that it is currently over. For more information on gestural editing cursors, see “Gestural Editing Cursors” on page 382.

NOTE: The hot spots are not marked. You can only tell when you move from one hot spot to the next by the change in the shape of the cursor.

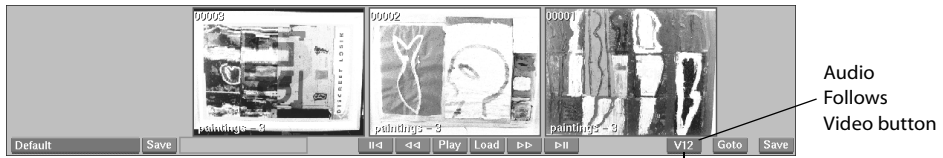


To cut a clip gesturally, position the cursor over the frame where you want to make the cut. Place the cursor in the Cut Clip hot spot on the frame edge closest to the cutting point. Grab the frame and drag up or down to cut it from the work reel.



Cutting Audio Gesturally

If you gesturally cut a clip with audio, use the Audio Follows Video button to specify how audio is affected.



When V12 is enabled, the video and both audio tracks are edited simultaneously.
When V is enabled, only video is affected during the gestural edit.

HINT: Hold the **A** hotkey to temporarily change the status of the Audio Follows Video button for the current gestural edit.

You can also change this option in the Desktop Preferences menu. If the Audio Follows Video button is enabled, the audio tracks are cut at the same location as the video. If the Audio Follows Video button is disabled, the audio tracks are not cut and the entire audio tracks are included on both new clips. The result is the same as cutting a clip from the Editing menu. See the illustration in “Cutting Clips with Audio Tracks” on page 398.

Splicing Clips

The Splice command creates a splice by joining two source clips together.

To splice two clips together:

1. Click the Splice button in the Editing menu. The Splice menu appears.



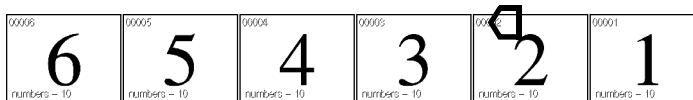
2. Select Src In/Rec Out from the Splice Type box. Select either Keep Sources or Remove Sources. For more information, see “Keep/Remove Sources Option Box” on page 390.
3. In the source clip, select the last frame before the splice. The frames from the beginning of the clip up to and including the selected frame will be contained in the new clip.
4. In the record clip, select the first frame after the splice. The frames from the selected frame up to the last frame in the clip will be contained in the new clip.
5. Select the destination reel.

The two clips are spliced together to form a clip that is added to the destination reel. A yellow transition handle indicates where the two clips were spliced together. To remove the yellow

handle, commit the clip. For information on committing clips, see “Consolidating and Committing Soft Clips” on page 391.

If you are showing source and record timecodes, the first timecode (left) is the original timecode of each source clip. The second timecode (right) is a new timecode for the splice starting at the record in timecode of the outgoing clip. For more information, see “Editing Timecodes” on page 390.

After creating the splice, the cursor reverts to a red arrow so that you can repeat the Splice command using another set of clips.



Action: In the first source clip, select the last frame before the splice



Action: In the second source clip, select the first frame after the splice



Result: The clips are spliced together

To splice together all clips on a reel:

1. Click the Splice button in the Editing menu. The Splice menu appears.
2. Select Reel from the Splice Type box. Select either Keep Sources or Remove Sources. For more information, see “Keep/Remove Sources Option Box” on page 390.
3. Click the reel containing the clips you want to splice together.
4. Click the destination reel.

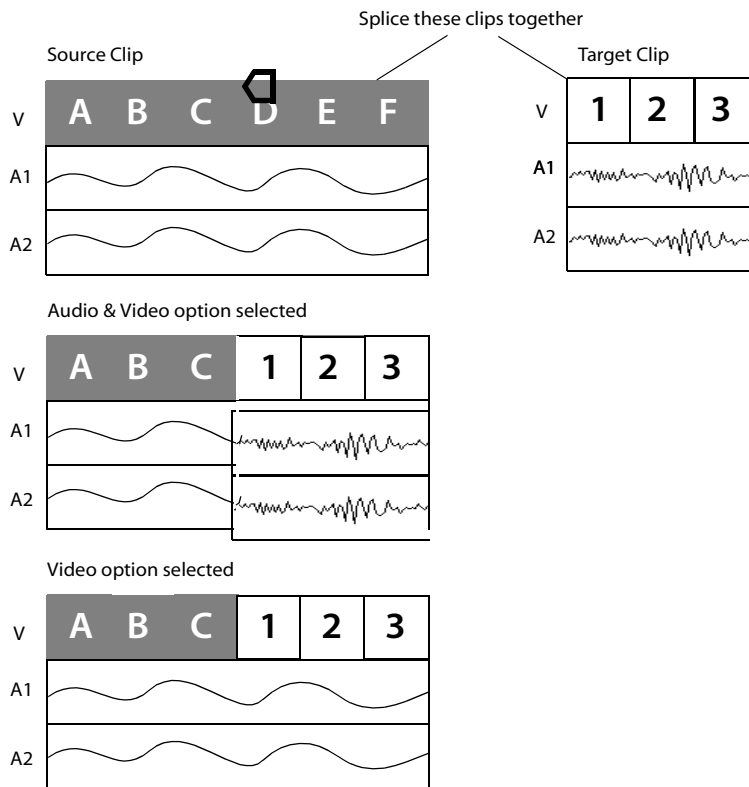
All clips on the reel will be spliced together into one clip and placed on the destination reel.

Yellow transition handles indicates where the clips were spliced together. To remove the yellow handle, commit the clip. For information on committing clips, see “Consolidating and Committing Soft Clips” on page 391.

NOTE: After splicing together an entire reel, you can access the Edit Reel to trim head and tail frames and create dissolves. For information on editing in the Edit Reel, see “The EditReel” on page 386.

Splicing Clips with Audio

If you splice two clips with audio, use the Video/Audio option box to specify how the audio is affected. When you use the Video & Audio option, audio is spliced at the same location as video and the source audio tracks overwrite the target audio tracks. If the source clip has no audio tracks, the target clip's audio tracks are overwritten with silence. If you use the Video option, the audio tracks in the source clip are ignored and the audio tracks on the target clip are unaffected.



Splicing Clips Gesturally

For information on gestural splicing, see “Gestural Splice, Insert and Replace Edits” on page 408.

Inserting Frames and Clips

Use the Insert command to to:

- Insert a portion of a source clip after a frame in the target clip
- Insert a source clip after a frame in the target clip

The option you select in the Insert Mode option box determines whether you insert frames or clips. This option box appears when you click the Insert command in the Editing menu.



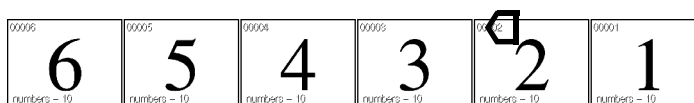
Choose:	To:
Src In/Out	Insert a portion of the source clip after the selected frame in the target clip. See “Inserting Frames” on page 403.
Rec In	
Clip	Insert the source clip after a selected frame in the target clip. This is the default mode which is reset between flame sessions. See “Inserting Clips” on page 404.

Inserting Frames

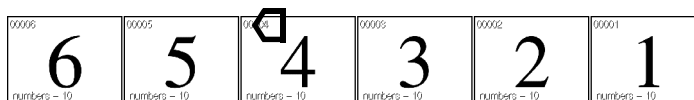
Select the Src In/Out Rec In mode to insert a portion of the source clip after a selected frame in the target clip.

To insert frames:

1. Click the Insert button on the Editing menu.
The Insert Mode option box appears.
2. Choose Src In/Out Rec In mode in the Insert Mode option box.
3. Select in-point of the source clip (the starting frame of the segment that you want to insert).
4. Select the out-point of the source clip (ending frame of the segment that you want to insert).
5. Select an in-point frame in the target clip.
The frames between the in-point and out-point of the source clip are inserted after the selected frame (the in-point) in the target clip. The remaining frames of the target clip are placed at the end of the inserted frames.



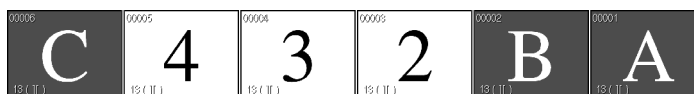
Action: Select the in-point of the source clip



Action: Select the out-point of the source clip



Action: Select an in-point frame on the target clip



Result: The selected frames are inserted into the target clip

Inserting Clips

Select Clip mode in the Insert Mode option box to insert an entire clip after the selected frame in the target clip.

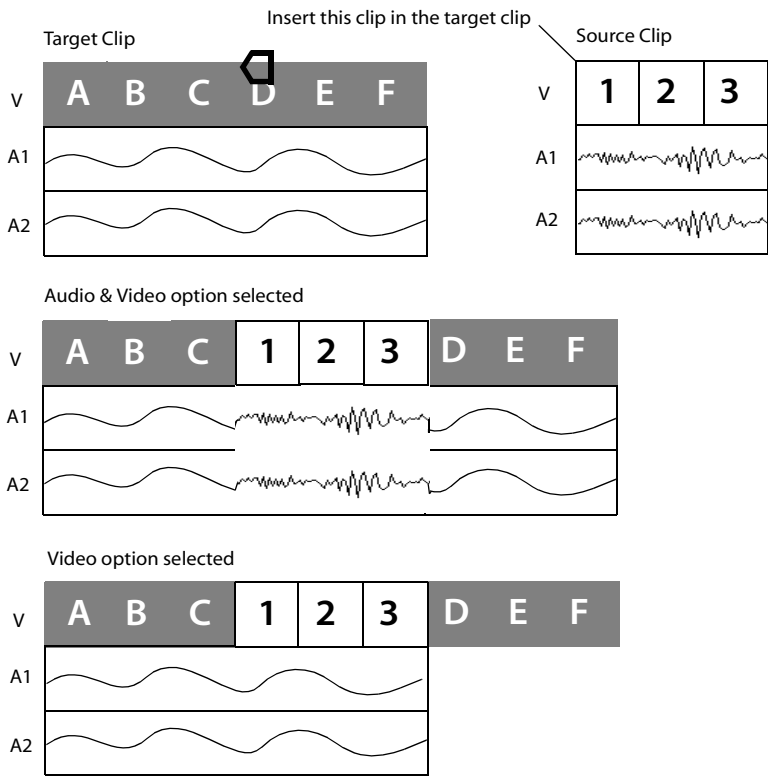
To insert a clip:

1. Click the Insert button in the Editing menu.
The Insert Mode option box appears.
2. Choose Clip mode in the Insert Mode option box.
3. Select the source clip.
4. Select an insertion-point frame in the target clip.

The source clip is inserted after the selected frame in the target clip. The remaining frames of the target clip are placed at the end of the inserted clip.

Inserting Clips with Audio

If you insert a clip with audio, use the Video/Audio option box to specify how the audio is affected. When you use the Video & Audio option, the source audio tracks overwrite the target audio tracks. If the source clip has no audio tracks, the target clip's audio tracks are overwritten with silence. If you use the Video option, the audio tracks in the source clip are ignored and the audio tracks on the target clip are unaffected.



Inserting Clips Gesturally

For information on gestural insert edits, see “Gestural Splice, Insert and Replace Edits” on page 408.

Replacing Frames and Clips

Use the Replace command to:

- Replace frames in the target clip with frames from a source clip
- Replace frames in the target clip with a source clip

The option you select in the Replace Mode box determines whether you are replacing frames with a series of frames or an entire clip. To view this box, click the Replace command in the Editing menu.



Replace Mode box

Select: **To:**

Src In/Out Rec In Replace frames in the target clip with a portion of the source clip. See "Replacing Frames" on page 406.

Clip Replace frames in the target clip with the source clip. This is the default mode which is reset between **flame** sessions. See "Replacing Clips" on page 407.

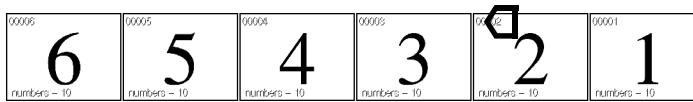
Replacing Frames

Select the Src In/Out Rec In to replace frames in the target clip with frames from the source clip.

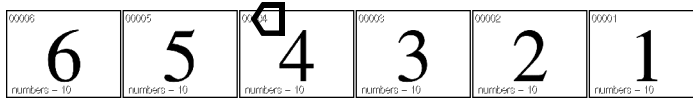
To replace frames:

1. Click the Replace button in the Editing menu.
The Replace Mode box appears.
2. Select Src In/Out Rec In mode in the Replace Mode box.
3. Select the in-point of the source clip. The in-point is the first frame of the segment that you want to insert.
4. Select the out-point of the source clip. The out-point is the last frame of the segment that you want to insert.
5. Select a frame in the target clip.

The frames after and including the selected frame in the target clip are replaced by the frames between the in-point and out-point of the source clip. An example is shown below.



Action: Select the in-point of the source clip



Action: Select the out-point of the source clip



Action: Select an in-point frame in the target clip



Result: The selected frames replace frames in the target clip

Replacing Clips

Select the Clip mode to replace frames in a clip with an entire clip.

To replace frames with a clip:

1. Click the Replace button in the Editing menu.

The Replace Mode box appears.

2. Click the Replace Mode box to switch to Clip mode.

3. Select an Audio follows video option:

- If you select Audio & Video, audio in the source clip replaces the specified audio in the target clip.
- If you select Video, audio in the target clip is unaffected.

NOTE: If you select Audio & Video and the source clip contains no audio tracks, the specified audio in the target clip is overwritten with silence.

4. Select the source clip.
5. Select a frame in the target clip.

The source clip replaces the frames after and including the selected frame in the target clip.

Gestural Splice, Insert and Replace Edits



All gestural splice, insert and replace edits ripple the edit sequence (and many change the duration of the target clip to accommodate the spliced, inserted, or replaced clip). The exception to this rule is when you replace a clip with another clip of the same duration.

The type of edit that occurs when you drop a source clip onto a target clip depends on the type of cursor that is visible when you drop the clip. The cursor type that appears depends on where you have grabbed the source clip, and where it is positioned over the target clip. For more information about gestural editing cursors, see “Gestural Editing Cursors” on page 382.

To insert a clip gesturally:

1. Drag the source clip over the target clip.
2. Position the source clip over the frame before the place where you want the insert to occur.
3. Drop the source clip when it is correctly positioned and the Insert cursor appears.

NOTE: Note that a splice is simply an insert at the end or beginning of a clip.

To perform a gestural replace edit:

1. Drag the source clip over the target clip.
2. Position the source clip where you want to make the replace edit, such that the Replace cursor appears.

The cursor that appears depends on how you grabbed the source clip. For more information, see “Gestural Editing Cursors” on page 382.

3. Drop the source clip when it is correctly positioned and the Insert cursor appears.

Matching a Soft Edit with Its Source Clips

Use the Match Source command to match a frame in a soft edit with its source clip, even if the source clip is not on the desktop. This is because **flame** keeps a history of all clips used to create a soft edit, and is always able to find a source clip in a soft edit’s history.

You can specify if **flame** will search the desktop and the clip library for the original source, or just the desktop when matching back to the original source. Limiting the search to the desktop is useful if you have many large local clip libraries as they may greatly increase the time taken to locate the original source clip.

If you select Desktop, only clips on the desktop are searched. If the original source cannot be found, a new source is created on the desktop that contains only the material for which you are searching. However, the newly created clip will not contain the original clip name or record timecode information.

If you select Desktop & Library, the desktop is searched first for the original source. If the original source is not found, all local clip libraries are searched. Searching both the desktop and

the clip libraries increases the likelihood of finding the original source but may also result in longer search times with large clip libraries. If the original source is not found either on the desktop or in the clip library, a new source is created on the desktop that contains only the material for which you are searching. However, the newly created clip will not contain the original clip name or record timecode information.

If you are matching a single-element clip that has been trimmed, the Match Source will create a new clip on the desktop and remove all of the handles. This provides an untrimmed version of the clip with the original name and record timecode information.

To match a soft edit with its source clips:

1. Click the Match Source button.
2. Select Desktop or Desktop & Library.
3. Select the frame in the soft edit that you want to match. The work reels are searched for the source clip that contains the selected frame:
 - If the source clip is on the desktop, it is centred on its reel and the reel number is shown in the message bar.
 - If the source clip is not on the desktop, the cursor changes to a white arrow. Select a destination reel. The source clip appears on the selected reel.

NOTE: If the original source did not exist on either the desktop or clip library, the new clip will not contain the original name or record timecode information.

EDITING	Cut	Repeat	Dissolve	Delete
PROCESSING	Insert	Reverse	Audio TimeStretch	Name
EFFECTS	Splice	Swap Shot	TimeWarp	Move
FORMAT	Replace	Consolidate	Commit	Copy
SYSTEM	Match Source	EDL	Force Render	Search

Match Source button

Repeating Frames in a Clip

The Repeat command makes a number of copies of a single frame, an entire clip, or each frame in a selected clip and joins the repeated frames or clips end to end.

To use the Repeat command:

1. Click the Repeat button in the Editing menu.

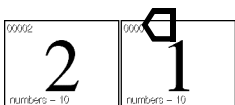
The Repeat command menu appears.



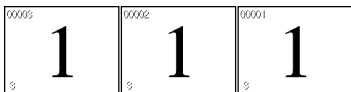
2. Use the Repeat Mode box to choose the type of item that you want to repeat. Press the box and choose one of the options.

Choose:	To:
Frame	Repeat a single frame. Frames are joined end to end to produce one new clip.
Clip	Repeat an entire clip. Clips are joined end to end to produce one new clip.
Frames in Clip	Repeat each frame in a clip. For example, if you set number of repetitions to three, then result clip contains first frame repeated three times, followed by second frame repeated three times, and so on.

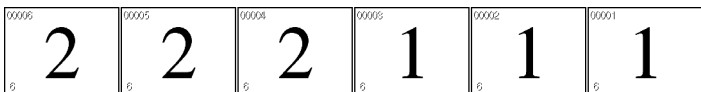
3. Enter the number of times that you want to repeat the frame or clip in the Repeat Number field. For the examples below, the Repeat Number is set to 3.
4. Select either the clip or the frame that you want to repeat.
5. Select the destination reel.
- A new clip containing the repeated frame(s) or clip appears and is added to destination reel.



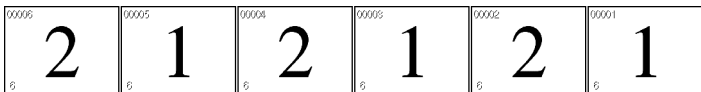
Action: Select the source clip or frame



Result: Repeating a frame



Result: Repeating the frames in a clip



Result: Repeating the clip

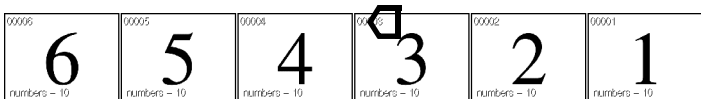
Reversing Frame Order in a Clip

The Reverse command reverses the order of frames in a clip.

To use the Reverse command:

1. Click the Reverse button in the Editing menu.
2. Select the source clip.
3. Select the destination reel.

A new clip in which the order of frames is reversed appears on the destination reel.



Action: Select the source clip



Result: The order of the frames is reversed

Swapping Shots

Use the Swap Shot command to replace a shot between two soft edit transitions, or between a transition and the start or end of a soft edit.

To swap a shot in a soft edit:

1. Click the Swap Shot button in the Editing menu.

The Shot and Force Render boxes appear.



2. If you are swapping a shot within a dissolve, use the Shot box to choose between swapping the outgoing or incoming shot.

Select:

To:

Outgoing Shot Select the outgoing shot when clicking on a frame within a dissolve.

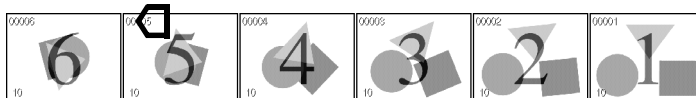
Incoming Shot Select the incoming shot when clicking on a frame within a dissolve.

3. Select the align frame in the source clip.

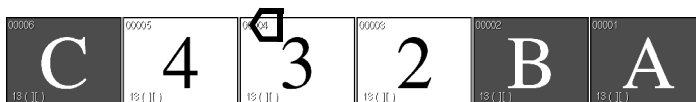
The align frame is used to align the source clip and shot being swapped.

4. Select the align frame in the shot that you want to swap.

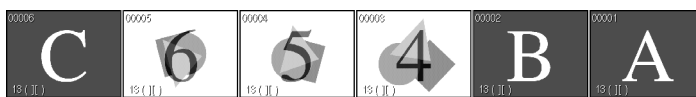
The shot and source clip are lined-up, using align frames, then swapped. If there are not enough frames before or after align frame in source clip to fit before or after align frame in selected shot, the error message “Selected Shot Does Not Fit at Destination” appears in message bar.



Action: Select the align frame in the source clip that will swap the shot



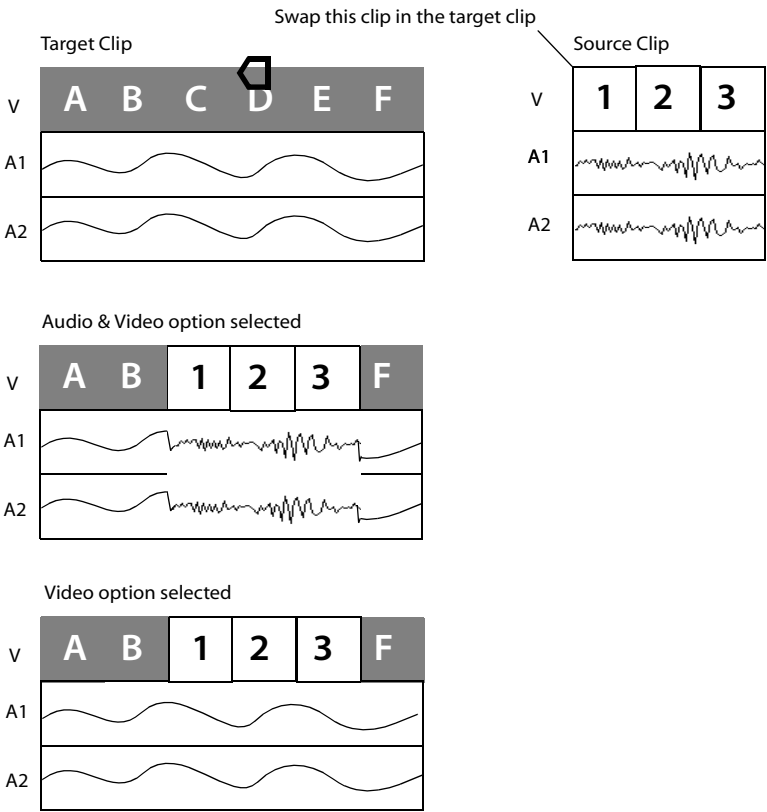
Action: Select the align frame in the shot you want to replace



Result: The shot is swapped with the source clip

Swapping Shots with Audio

If you swap a clip with audio, use the Video/Audio option box to specify how the audio is affected. When you use the Video & Audio option, the source audio tracks overwrite the target audio tracks. If the source clip has no audio tracks, the target clip's audio tracks are overwritten with silence. If you use the Video option, the audio tracks in the source clip are ignored and the audio tracks on the target clip are unaffected.



Consolidating Clips

When consolidating, you enter a value in the Consolidation field. The value represents upper handles limit. Thus, for example, if have a clip with 40 head frames and 10 tail frames, and you consolidate it with a value of 15, the resulting clip has 15 head frames and 10 tail frames.

You might want to consolidate clips with large amounts of heads and tails before archiving to avoid archiving unused material.

To limit heads and tails (consolidate):

1. Click the Consolidate button on the Editing menu.
2. Select either “Element”, “Clip”, “Range”, “Reel”, or “Desktop”, depending on what you want to consolidate.
3. Choose an upper handles limit.
4. With the red cursor, select the item you wish to consolidate.

Deleting Clips

The Delete command is available in all of the module menus. Use it to delete:

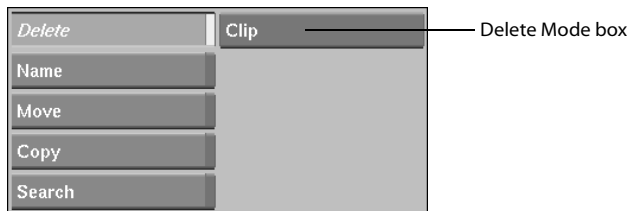
- A single frame from a clip
- A range
- A selected clip
- All clips on a selected work reel
- All clips on the desktop
- An audio track that is linked to a clip

Deleting a Frame, Range, Clip, or a Reel

To delete a frame, clip, or a reel:

1. Click the Delete button.

The Delete Mode box appears.



2. Press the Delete Mode box and select one of the options.

3. Select the frame, range, clip or reel that you want to delete. To select a reel, select any clip on that reel.

If you are deleting a frame, the selected frame is spliced out of the clip. The clip becomes a soft edit. For more information on splices, see “Splicing Clips” on page 400.

If you are deleting a clip or reel, the selected clip or reel is deleted.

NOTE: The cursor remains as a red arrow until you select another menu option to disable the Delete button. Be careful not to inadvertently delete additional frames, clips, or reels while the Delete button is enabled.

Deleting Frames or a Range with Audio

If you select Frame or Range when using the Delete command, an Audio/Video option box appears. If you select Audio & Video, both audio and video is deleted for the selected range. If you select Video, all audio is unaffected.

Deleting Audio Tracks

You can use the Delete command to delete audio tracks that are linked to a clip.

To delete an audio track:

1. Click the Delete button.
The Delete Mode box appears.
2. Press the Delete Mode box and select Audio.
3. Select the clip linked to the audio track that you want to delete.

Deleting a Desktop

You can use the Delete command to delete all the clips on your desktop.

To delete all the clips on the desktop:

1. Click the Delete button.
The Delete Mode box appears.
2. Press the Delete Mode box and select Desktop.
3. Click the Confirm button.
When the desktop is deleted, the desktop name is cleared.



Deleting Clips Gesturally

To delete a clip from the desktop or from the Clip Library, grab it and drag it toward the bottom of the screen until the cursor changes into a green “recycle” icon, then drop the clip.

Naming and Renaming Clips

Use the Name command to name or change the name of a clip at any time during a work session.

To name or rename a clip:

1. Click the Name button in the Editing menu.

The keyboard appears.

2. Select the clip to which you want to assign the name.
3. Enter the clip name.

If the display of clip names is enabled, the name appears on each frame of the selected clip.

HINT: Also, move the pointer over a clip, reel or desktop area and press the N hot key to name a clip, reel or desktop.

Moving Clips

The Move command is available in all of the module menus. Use it to move:

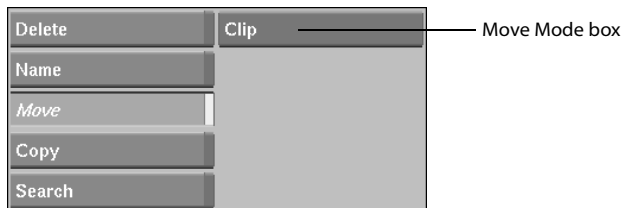
- A single frame
- A range
- A selected clip
- All clips on a selected work reel
- Audio tracks that are linked to a clip

Moving a Frame, Range, Clip, or Reel

To move a frame, clip, or reel:

1. Click the Move button in the Editing menu.

The Move Mode box appears.



2. Press the Move Mode box and select one of the options.
3. Select the frame, range, clips, or reel that you want to move.

If you are moving a frame, the selected frame is spliced out of the clip. The clip becomes a soft edit. For more information on splices, see “Splicing Clips” on page 400.

4. Select the destination reel.

The selected frame, clip, or reel is moved to the destination reel. If a single frame is moved, the timecodes (or frame numbers) in the source clip are updated.

Moving a Frame or Range with Audio

If you select Frame or Range while using Move command, an Audio/Video option box appears. If you select Audio & Video, audio and video tracks in selected range are both moved to the new location. If you select Video, the entire audio tracks are moved regardless of the selected range.

Moving Audio Tracks

You can use the Move command to move an audio track linked to a clip.

To move an audio track:

1. Click the Move button in the menu.

The Move Mode box appears.

2. Press the Move Mode box and choose the Audio option.
3. Select the source clip that is linked to the audio track.
4. Select the destination clip.

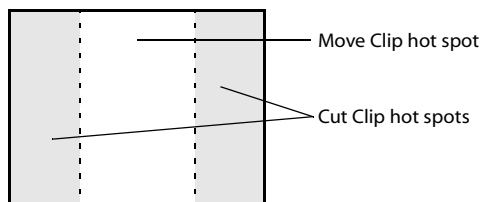
The audio track from the source clip is moved and linked to the destination clip. The source clip is no longer linked to the audio track.



Moving Clips Gesturally

You can move a clip from one reel to another, or from one location to another within a reel. To move a clip, grab the Move Clip hot spot of any frame in the clip and drag up or down to detach it from the work reel. Be careful where you grab the clip; each frame contains drag and drop hot spots for cutting as well as for moving. You can also move a single frame by pressing **CTRL**.

NOTE: The hot spots are invisible. You can only tell when you move from one hot spot to the next by the shape of the cursor.



Copying Clips

The Copy command is available in all of the module menus. Use it to copy:

- A single frame
- A range
- A selected clip
- All clips on a selected work reel
- Audio tracks that are linked to a clip

Copying a Frame, Range, Clip, or Reel

To use the Copy command:

1. Click the Copy button.

The Copy Mode box appears.



2. Press the Copy Mode box and select one of the options.
3. Select a frame, range clip, or reel.
4. Select the destination reel.

The frame, range, clip, or reel is copied to the destination reel.

Copying a Frame or Range with Audio

If you select Frame or Range while using the Copy command, an Audio/Video option box appears. If you select Audio & Video, the audio and video tracks in the selected range are both copied to the new location. If you select Video, the entire audio tracks are copied, regardless of the selected range.

Copying the Source Clip Out of an Edited Clip

To copy the source clip in it's original length out of an edited clip, hold down **SHIFT** and drag a frame of the clip you want to pull out of the soft-edit.

Copying Audio Tracks

You can use the Copy command to duplicate audio tracks linked to a clip.

To copy an audio track:

1. Click the Copy button in the menu.

The Copy Mode box appears.

2. Press the Copy Mode box and choose the Audio option.
3. Select the source clip linked to the audio tracks that you want to copy.
4. Select the destination clip.

A new link is made between the audio tracks and the destination clip. Note that the audio tracks themselves are not copied, but are now linked to both the source clip and the destination clip.

[illegible]

21

Dissolves

Transitioning

Dissolves can create smooth transitions between shots. You can also create fade-ins and fade-outs for both video and audio. Using the Dissolve Editor, you can create elaborate dissolves that follow a non-linear dissolve curve.

Summary

In this chapter, you learn about:

- “The Dissolve Command” on page 422
- “Creating Complex Dissolves” on page 428
- “Dissolving between Two Clips” on page 424
- “Crossfading between Two Clips” on page 425
- “Creating a Fade-in or Fade-out” on page 427
- “Creating Audio Dissolves and Fades” on page 428
- “Creating and Modifying Dissolves in the EditReel” on page 432

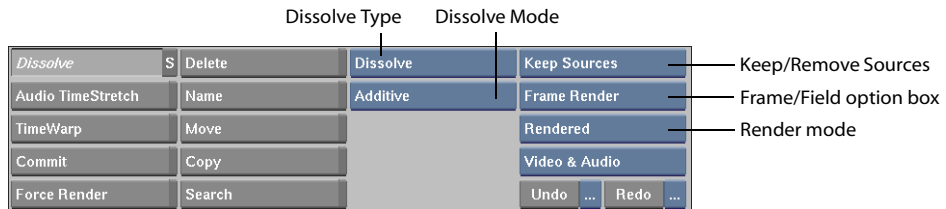
About Dissolves

A dissolve is a blending transition in which the first clip fades out at the same time as the second clip fades in. Dissolves are soft transitions created using the Dissolve command or the EditReel. For an overview of soft transitions, see “Soft Edits” on page 389. There are two ways to create a dissolve. You can use the Dissolve command on the desktop, or you can use the Timeline in the EditReel to convert a splice to a dissolve. You normally use the Dissolve command to create a dissolve for two separate clips on the desktop that are not currently spliced together. If you want to add a dissolve between two clips that are already splice together, you would use the EditReel to convert the splice to a dissolve. If you use the Audio Follows Video option, a dissolve is added to the audio tracks as well.

NOTE: Audio dissolves are linear and do not use the same curve that is applied to video dissolves in the Dissolve Editor.

The Dissolve Command

The Dissolve command in the Dissolve menu creates a dissolve that fades between two source clips (dissolve or crossfade) or one source clip and a black source (fade in or fade out). To view the Dissolve menu, click the Dissolve button in the Editing menu.



Dissolve Type Options

Use the Dissolve Type option box to select the type of dissolve that you want to create. The Dissolve and Crossfade options both dissolve between two clips, but the Crossfade option gives you more control over the dissolve rate and selection criteria.

Select:	To:
Dissolve	Create a dissolve between two clips. The dissolve rate is based on the frame you select in the incoming clip. See “Dissolving between Two Clips” on page 424.
CrossFade	Create a dissolve between two clips. You can choose the dissolve rate and the criteria used when selecting clips. See “Crossfading between Two Clips” on page 425.
Fade In	Create a dissolve that fades from a black source to a clip. See “Creating a Fade-in or Fade-out” on page 427.
Fade Out	Create a dissolve that fades from a clip to a black source. See “Creating a Fade-in or Fade-out” on page 427.
Mix	Mix the pixels between two clips based on a percentage you specify.

Frame/Field Options

Click the Frame/Field option box to render the dissolve in frames or fields.

Dissolve Mode Options

When creating a dissolve, you can specify how the pixels of the source clips are combined to produce the corresponding pixels of the new clip. The dissolve mode can be Additive, Nonadditive, or Inverse Nonadditive.

Additive — The colour value of a pixel in the first source clip is added to the colour value of the corresponding pixel in the second source clip. This value is assigned to the pixel placed in the resulting dissolve.

Nonadditive — Corresponding pixels in the first and second source clips are evaluated to determine which has the higher colour value. The pixel in the resulting dissolve is assigned the higher of the two values.

Inverse Nonadditive — Corresponding pixels in the first and second source clips are evaluated to determine which has the lower colour value. The pixel in the resulting dissolve is assigned the lower of the two values.

Alignment Options

The Dissolve Alignment box appears when you select CrossFade, Fade In, or Fade Out as the dissolve type. Select the alignment used when selecting the outgoing and incoming clips for the dissolve.

Select:	To:
Start Point	Select the starting frame of transition in both the outgoing and incoming clip.
Mid Point	Select the middle frame of transition in both the outgoing and incoming clip.
End Point	Select the end frame of transition in both the outgoing and incoming clip.

Dissolve Rate Field

The Dissolve Rate field appears when you select CrossFade, Fade In, or Fade Out as the dissolve type. Use this field to specify the number of frames over which the dissolve occurs. For example, if the dissolve rate is set to 10, then the dissolve takes place over 10 frames.

Render Mode Options

Use the Render Mode box to specify when dissolves are rendered.

Select:	To:
Rendered	Always render dissolves.
Unrendered	Never render dissolves. Dissolves will remain unprocessed until Force Render is selected. Select Unrendered if you want to edit transitions in the Soft Clip Editor without having to wait for dissolves to be rendered.

When you select Unrendered, a black frame with the message “Unrendered Frame” is substituted for a rendered frame whenever a dissolve is created or whenever it is modified in the Soft Clip Editor.

If your system is capable of creating real-time dissolves and you are modifying a dissolve in Unrendered mode, the dissolve is shown in the image window when the clip is played. The dissolve is not shown on the EditReel at the bottom of the Soft Clip Editor.

Keep/Remove Sources Options

Use the Keep/Remove Sources box to select whether you want to keep or remove the source clips after creating the dissolve. For more information, see “Keep/Remove Sources Option Box” on page 390.

Dissolving between Two Clips

The Dissolve command is used to dissolve between two clips. When you use the Dissolve command, the first source clip you select is the outgoing clip and the second source clip is the incoming clip.

The number of frames over which the dissolve occurs is specified by the dissolve rate. The dissolve rate is set when you select a frame in the incoming clip; the number of the selected frame relative to the first frame in the clip is used as the dissolve rate.

To create a dissolve:

1. Click the Dissolve button in the Editing menu.
2. Select the Dissolve option from the Dissolve Type box.
3. Set the Frame/Field, Dissolve Mode, Render Mode, and Keep/Remove Sources options.
4. Select the first source clip. The first source clip is the outgoing clip.
5. Select the second source clip. The second source clip is the incoming clip. The number of the frame that you select in the incoming clip specifies the dissolve rate.

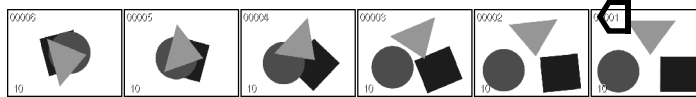
An “E” (for Editor) appears on the Dissolve button after you select the second clip. To use the Dissolve Editor, click the Dissolve button again. For more information, see “Creating Complex Dissolves” on page 428.

6. Select a destination reel.

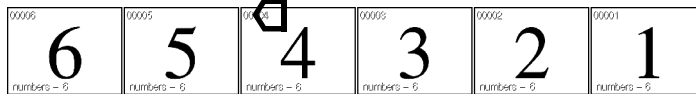
A new clip containing the dissolve appears on the destination reel. The length of the new clip can be determined using the following equation:

$$(length\ of\ outgoing\ clip) + (length\ of\ incoming\ clip) - (dissolve\ rate)$$

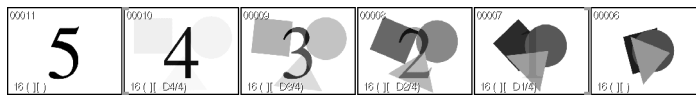
An example of a dissolve is illustrated below. In the example, the fourth frame in the incoming clip is selected. As a result, the dissolve rate is set to four.



Action: Select the outgoing clip



Action: Select the incoming clip. The dissolve rate is set to 4.



Result: The dissolve across four frames

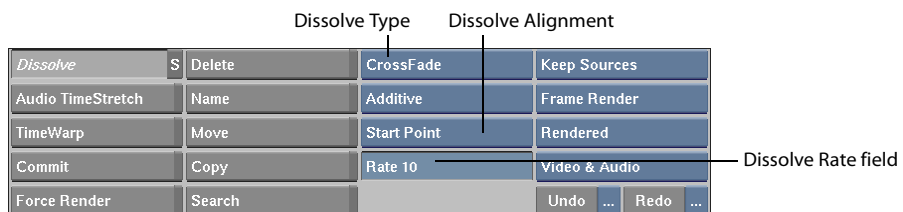
Crossfading between Two Clips

A crossfade is the same as a dissolve, but this option gives you more control over the duration of the dissolve, and the portions of the clips that are used for the crossfade.

To create a crossfade:

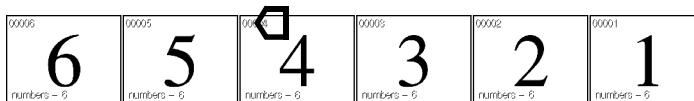
1. Click the Dissolve button in the Editing menu.
2. Select the Crossfade option from the Dissolve Type box.

The Dissolve Rate field and Dissolve Alignment box appear.



3. Set the Dissolve Alignment, Dissolve Rate, Frame/Field, Dissolve Mode, Render Mode, and Keep/Remove Sources options.
4. Select the frame in the outgoing clip that will be the start, middle, or end of the dissolve.

5. Select the frame in the incoming clip that will be the start, middle, or end of the dissolve. An “E” (for Editor) appears on the Dissolve button after you select the second clip. To use the Dissolve Editor, click the Dissolve button again. For more information, see “Creating Complex Dissolves” on page 428.
 6. Select the destination reel.
- If the “E” button is enabled, the Dissolve Editor appears. Otherwise, the crossfade is added to the destination reel. Examples of creating a crossfade using the start, middle, and end alignments are shown below.



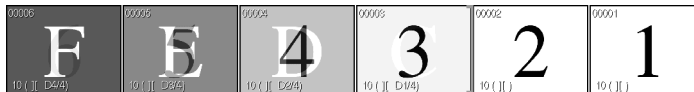
Action: Select the start, mid, or end point of the dissolve in the outgoing clip.



Action: Select the start, mid, or end point of the dissolve in the incoming clip.



Result: The dissolve using start point. The dissolve rate was set to 4.



Result: The dissolve using mid point. The dissolve rate was set to 4.



Result: The dissolve using end point. The dissolve rate was set to 4.

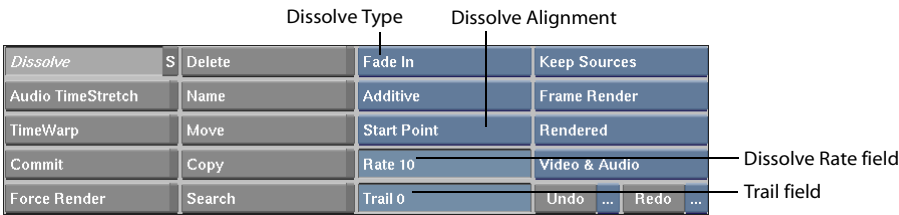
Creating a Fade-in or Fade-out

In a fade-in or fade-out, the outgoing clip (fade-in) or incoming clip (fade-out) is a black source.

To create a fade-to-black or fade-from-black:

- 1. Click the Dissolve button in the Editing menu.
- 2. Select either Fade In or Fade Out from the Dissolve Type box.

The Trail, Dissolve Rate, and Dissolve Alignment options appear.



- 3. Use the Trail field to enter the number of pure black frames to place before or after the fade-in or fade-out, respectively.
- 4. Set the Dissolve Alignment, Dissolve Rate, Frame/Field, Dissolve Mode, Render Mode, and Keep/Remove Sources fields.
- 5. Select the frame in the source clip that will be the start, middle, or end of the fade-in or fade-out.

An “E” (for Editor) appears on the Dissolve button after you select the source clip. To use the Dissolve Editor, click the Dissolve button again. For more information, see “Creating Complex Dissolves” on page 428.

- 6. Select the destination reel.

If the “E” button is enabled, the Dissolve Editor appears. Otherwise, the fade-in or fade-out is added to the destination reel. An example of creating a fade-in and fade-out is shown below.

Creating Audio Dissolves and Fades

If a clip has audio tracks, you can add a dissolve to both video and audio tracks at the same time. If you use the Video and Audio option when adding a dissolve, crossfade or fade-in/out, the dissolve or fade is added to both video and audio tracks. If you use the Video option, the dissolve or fade is only added to the video track.

NOTE: You cannot add a dissolve or fade to audio tracks independently of video tracks.

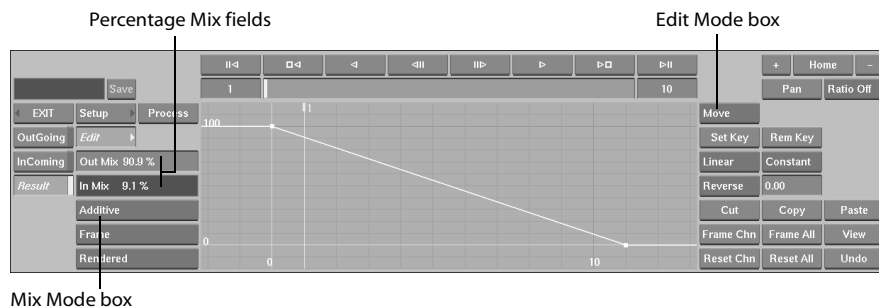
Creating Complex Dissolves

Use the Dissolve Editor to create more complex dissolves. For example, instead of having a gradual fade-in of the incoming clip, you can make your clip fade in, fade out, then fade in again.

To use the Dissolve Editor:

1. Select the clips for the dissolve as previously described.
An “E” (for Editor) appears on the Dissolve button.
2. Click the Dissolve button again.
3. Select the destination reel.

The Dissolve Editor appears.



The graph appearing in the menu plots the frame number of the result clip (on the horizontal axis) against the percentage mix (on the vertical axis). The default inverted-S curve represents a gradual dissolve across the number of frames specified by the dissolve rate.

4. The percentage mix value for one frame specifies the amount of the incoming and outgoing clip used in the mix for that frame. For example, if you enter 90% in the Out Mix field, 90% of the outgoing clip and 10% of the incoming clip are used in that frame. You can also enter mix values directly in the In Mix field.

You can edit the percentage mix in any frame using either of the following methods:

- Display the frame that you want to edit and change the value in either the Out Mix or In Mix field. If the AutoKey button in the Dissolve Setup menu is enabled, a control point is added to the curve at that frame.
- Add and move control points on the percentage mix curve itself. Use the Add and Move options in the Edit Mode box to edit the curve.

Keep the following limits in mind when you are adding control points to the curve:

- You can only add control points between the dissolve boundary keys (any control points that you add outside of the boundary keys will not be processed). The boundary keys are located at the first and last frames in the dissolve.

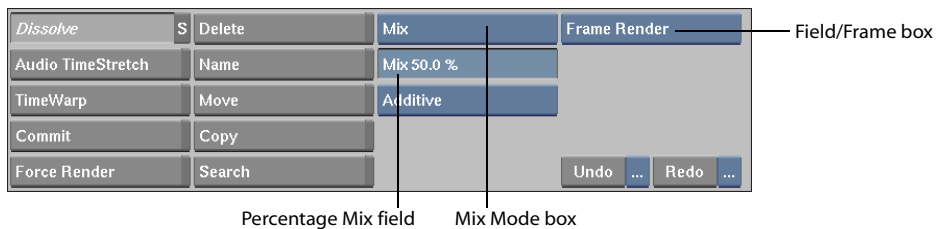
NOTE: When you enter a new rate, the boundary keys are automatically updated.

5. You can also use the Dissolve Editor to select processing in frames or fields and to load or save mix curves. Click the Setup button to display the Load and Save options.
6. When you are satisfied with the results, click Exit.

NOTE: The curve in the Dissolve Editor is applied to video dissolves only. Audio dissolves are always linear.

Mixing Source Clips

The Mix command generates a new clip from two source clips. Each frame in the generated clip is the result of mixing the image in a frame of the first source clip with the image in the corresponding frame of the second source clip. To view the Mix menu, select Mix in the Dissolve menu.

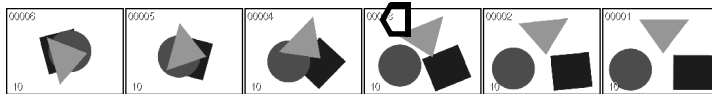


The Mix Mode

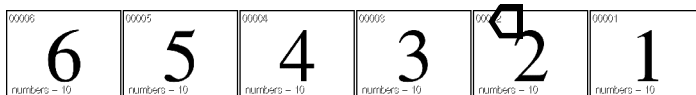
When you are using the Mix command, you can specify how the pixels of the source clips are combined to produce the corresponding pixels of the new clip. The mix mode can be additive, non-additive, or inverse non-additive.

Percentage Mix

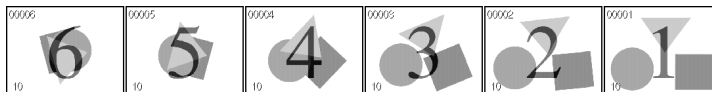
The Percentage Mix value specifies how much of each clip will be used in the mix. For example, if the Percentage Mix is set to 80%, the resulting mix will contain 80% of the first source clip and 20% of the second source clip.



Action: Select the first source clip



Action: Select the second source clip



Result: The result clip contains 50% of each source clip

To use the Mix command:

1. Select Mix in the Dissolve menu.
The Mix menu appears.
2. Enter the Percentage Mix value.
3. From the Mix Mode box, select a mix mode.
4. Select the first source clip.
5. Select the second source clip.
6. Select the destination reel.
The mixed clip appears on the destination reel.

The Mix Editor

You can edit the percentage mix across the length of the clip using the Mix Editor. For example, instead of having the same percentage mix across all frames in the clip, the percentage mix can be 50% in frames 1 through 5, 75% in frames 6 through 10, 25% in frames 11 through 15, and so on.

To display the Mix Editor:

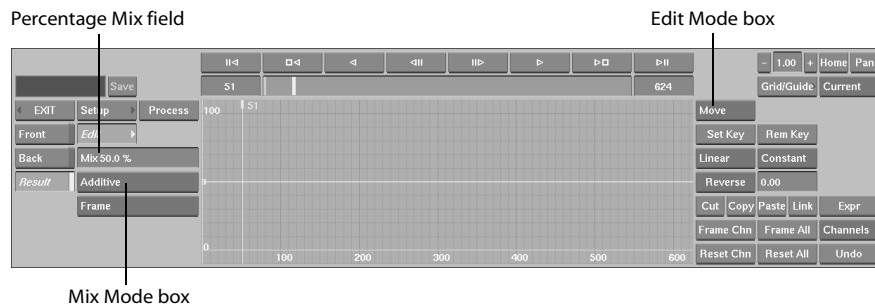
1. Select the clips for the mix as previously described.

An “E” (for Editor) appears on the Mix button.

2. Click the Mix button again.

3. Select the destination reel.

The Mix Editor appears.



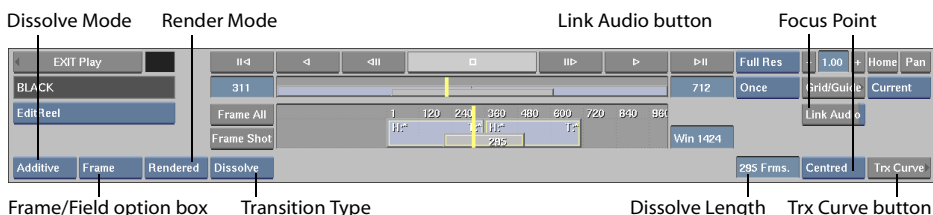
The graph that appears in the menu plots the frame number (on the horizontal axis) against the percentage mix (on the vertical axis). The default curve is a straight line, indicating that the percentage mix is the same (50%) in all frames of the clip.

To change the percentage mix in one or more frames of the clip:

1. Display the frame that you want to edit in the image window. This will allow you to see the result of changing the percentage mix value for that frame.
2. Select the Add option in the Edit Mode box. Add a point on the curve for the frame that you want to edit.
3. Select the Move option in the Edit Mode box. Move the new point up or down to the required percentage mix value.
4. Continue to add, move, or delete points on the curve until you achieve the required result.
5. You can also use the Mix Editor to select processing in frames or fields, and to load or save mix curves. Click the Setup button to display the Load and Save options.
6. Click Process to process the clip. When you are finished editing and processing the clip, click Exit to return to the desktop reels.

Creating and Modifying Dissolves in the EditReel

After creating a dissolve, you can use the EditReel to change the length, dissolve mode, and so on. When you centre a frame used in a dissolve (a frame with “Dn/m” in its lower-left corner), the Transition controls appear. You can also convert a splice into a dissolve in the EditReel.



Dissolve Length field — Shows the present length of the dissolve. Use this field to change the number of frames used in the dissolve.

Render Mode option box — Affects when dissolves are rerendered. For more information, see “Render Mode Options” on page 423.

Dissolve Mode option box — Affects how the pixels of the outgoing and incoming clips are combined to produce the corresponding pixels of the dissolve. The dissolve mode can be Additive, Non-additive, or Inverse Non-additive. For more information, see “Dissolve Mode Options” on page 422.

Trx Curve button — Click to access the Dissolve Editor and create a custom dissolve. For more information, see “Creating Complex Dissolves” on page 428.

Frame/Field option box — Click to render the dissolve in fields or frames.

Focus Point option box — Use this box to indicate where frames are taken from, or added to, when modifying dissolve lengths.

Select:	To:
At Start	Take or add frames to the start of the dissolve.
At End	Take or add frames from the end of the dissolve.
Centred	Take or add frames equally from the start and the end.

Transition Type option box — Click to set the current transition as either a dissolve or a splice.

You can enter default dissolve values in the EditReel Setup menu. These values are applied to a new dissolve when you change a splice to a dissolve using the Transition Type display. For information about the EditReel Setup menu, see “EditReel Setups” on page 388.

Link Audio button — Enable to modify the dissolves on the audio tracks at the same time you modify the dissolves on the video track.

To create (or modify) a dissolve in the EditReel:

1. Load a clip containing a soft edit (or a dissolve) into the Player.
2. Select EditReel.
The EditReel appears.
3. Position the soft edit handle (or one of the dissolve boundary handles) on the throne (the center point) of the EditReel.
The Transition controls appear.
4. If you want to add the dissolve to the audio tracks as well, enable Link Audio.
5. To create a dissolve, change the transition type from Splice to Dissolve.
A dissolve is created using the default values in the EditReel Setup menu.
 - To modify a dissolve, gesturally trim the head and tail from the outgoing or incoming clips, adjust the dissolve length in the Dissolve Length field, or modify any of the other dissolve parameters.
 - Click Trx Curves to open the Dissolve Editor to modify a complex dissolve.

[illegible]

22

Trimming Shots

Take a little off the sides

When you bring two clips together to create a transition, there are usually a number of frames left over from the incoming and outgoing clips. You can modify the transition by trimming, slipping, and sliding these head and tail frames.

Summary

In this chapter, you learn about:

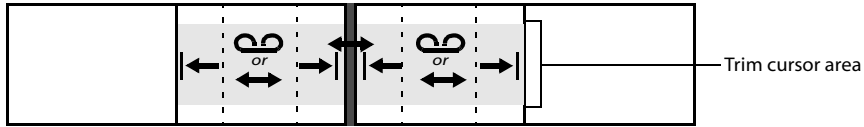
- “Trim Cursors” on page 436
- “Trimming Shots with Audio” on page 438
- “Head, Tail, and Offset Information” on page 437
- “Sliding Transitions” on page 439
- “Using the Slide Lock” on page 437

About Trimming

Trimming is the process of adding or removing frames from the head or tail of a shot in order to change its duration. You trim shots in the EditReel. You can also slip and slide shots. When you slip or slide a shot, you do not change its duration but only modify the head or tail frames or the in and out points to change the frames that appear in the final edit.

Trim Cursors

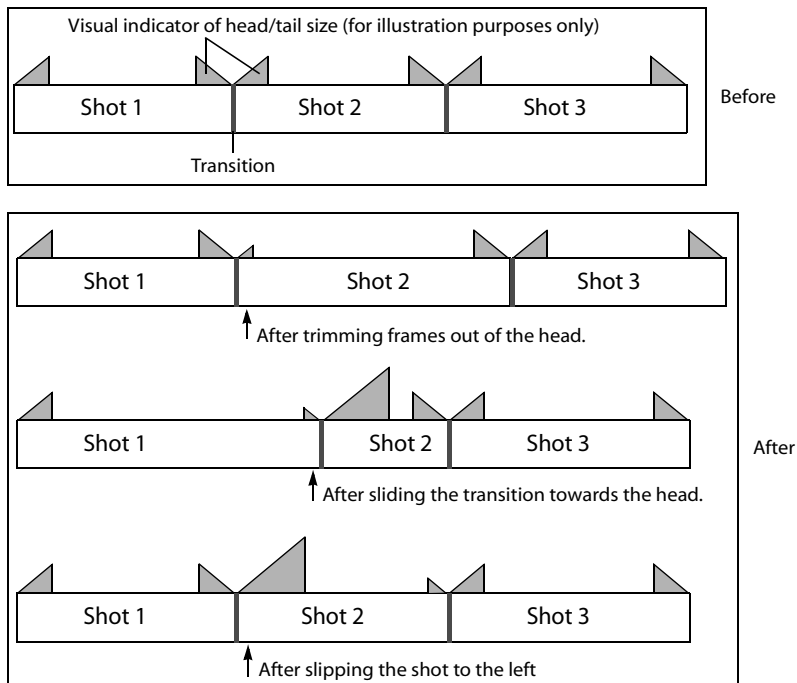
The trim cursors appear in the trim cursor area on the two frames on either side of the transition in EditReel. The following illustration shows trim cursors, and locations on EditReel where they appear.



Use:	To:
← Trim Head	Trim frames into or out of the head of the shot.
→ Trim Tail	Trim frames into or out of the tail of the shot.
↔ Slide	Simultaneously trim frames in and out of head and tail of shots on either side of a transition. The effect is that the location of transition slides to left or right.
⌛ Slip	Simultaneously trim frames in and out of the head and tail of a shot. The effect is that transitions remain where they are, but the shot is slipped forward or backward. Hold CTRL when the Slide cursor is visible to bring up Slip cursor.

Effects of Trim Operations

The following illustration shows the effect of various trim operations. These are conceptual representations of trimming operations and are not meant to be visual reproductions of the EditReel.



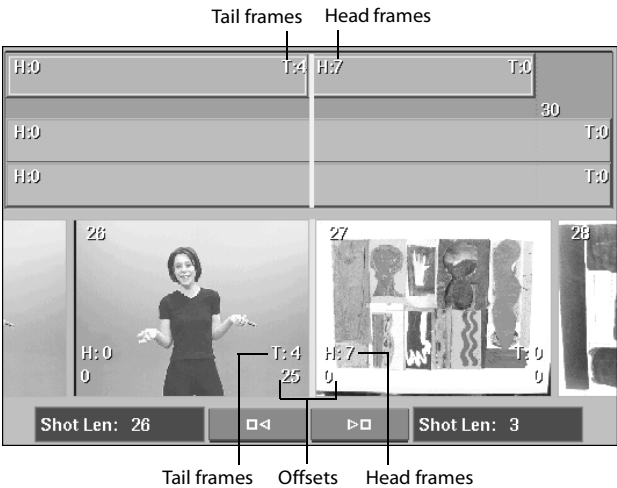
Head, Tail, and Offset Information

Offsets are reference numbers that indicate the change, in frames, that takes place when you trim. Head and tail offsets are displayed for the current clip on the frame that is centred on the EditReel. Offsets for the head and tail of the clips on either side of the transitions are displayed.

Offsets are created as follows.

	Action	Result Offset
Head Offsets	Trimming frames out of a clip's head.	Positive
	Trimming frames into a clip's head.	Negative
Tail Offsets	Trimming frames out of a clip's tail.	Negative
	Trimming frames into a clip's tail.	Positive

NOTE: Offset information is reset to zero when you exit the Player. Or you can reset the offset by clicking the offset value and clicking either Reset Offset or Reset All.



Using the Slide Lock

When a shot is slid, the heads and tails of the transitions at either end of the shot are trimmed. You can prevent the shot you are sliding from having its head and tail trimmed by enabling the Slide Lock button below the EditReel.

When Slide Lock is enabled, the head and tail of the slid shot are unaffected. The sliding action occurs by having the heads and tails of the surrounding shots trimmed.

Trimming Shots with Audio

If the clip contains audio, you can trim video and audio tracks simultaneously. Enable the Link Audio button to trim audio as you trim video. When you enable the Link Audio button, all elements that will be affected are highlighted in yellow.

NOTE: You cannot trim audio independently of video.

If you have waveforms enabled, **flame** needs to rebuild the waveforms after every trim operation. To rebuild the waveforms manually, click the Render Au. button that appears after you trim. To rebuild the waveforms automatically, enable the Auto Load button.

Trimming, Sliding, and Slipping Shots

You can trim, slide, and slip a shot in the Player's EditReel.

To trim, slide, or slip a shot:

1. Center the shot on the desktop reel.

2. Click the Play button.

The Player appears.

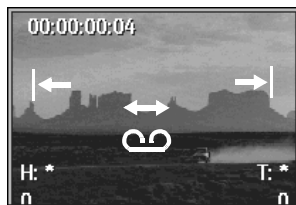
3. Select EditReel.

The EditReel appears.

Select EditReel to trim your clip.



4. Move cursor over the clip until the trim cursor appears for the operation you want to perform.



Move the cursor over the frame until you get the trim cursor you want. Hold **CTRL** to get the Slip cursor.

5. Drag left or right to add or remove frames from the head or tail. You can also click on the head or tail value and enter the number to trim.

NOTE: You can no longer trim, slip or slide once the head or tail value reaches 0.

Sliding Transitions

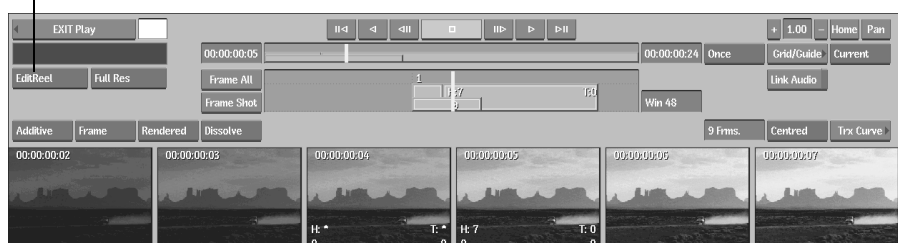
You can slide a transition in the Player's EditReel.

To slide a transition:

1. Centre the transition on the desktop reel.
2. Click the Play button.
The Player appears.
3. Select EditReel.

The EditReel appears.

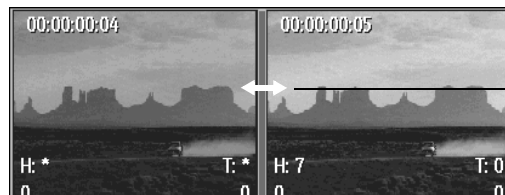
Select EditReel to trim your clip.



The timeline shows the elements that make up the clip, including the head and tail frames.

NOTE: The EditReel direction is left-to-right.

4. Position the cursor over the transition icon between the two shots.



Move the cursor over the transition icon until the slide cursor appears.

5. Drag the slide cursor left or right to trim frames into or out of the transition.
6. Click Exit Play to return to the desktop.

NOTE: You can use the Undo and Reset buttons to undo modifications made to the soft edit. Click the Undo button to undo the last modification. Click the Reset button to undo all modifications made to the soft edit since you entered the Player. Click the Reset Au. button to reset the audio tracks but not the video track.

This image shows a full page of white paper with horizontal dashed lines. The lines are evenly spaced and run across the width of the page, providing a guide for handwriting practice. There are no margins, text, or other markings on the page.

Use timewarps to apply motion effects to your clip such as slowing down or speeding up the clip, or adding trail to the image.

Summary

In this chapter, you learn about:

- “Creating Basic Timewarps” on page 442
- “Modifying Timewarps in the EditReel” on page 447
- “Modifying Timewarps in the Timewarp Editor” on page 444

Soft timewarps are created with the TimeWarp command. For more information, see “Soft Edits” on page 389.

About Timewarps

A timewarp is a clip in which the speed of the action taking place appears to go faster or slower than when it was originally recorded. This effect occurs when the number of frames in the clip is expanded or contracted, and the action is interpolated over the new number of frames.

Choosing the Right Method for Your Timewarp

Before you create a timewarp, you need to decide if you will create it with the TimeWarp command in the Editing menu (basic timewarp), or with Motion Estimation in Batch (advanced timewarp).

Timewarping Audio

You can timewarp audio by using the Audio Timestretch command in the Editing menu. See “Audio Timewarps” on page 481.

Creating Basic Timewarps

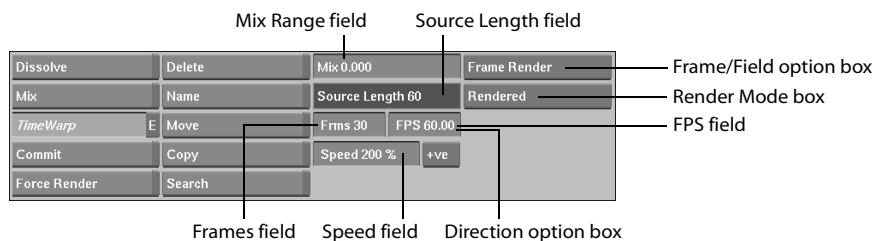
The TimeWarp command in the Editing menu lets you speed up or slow down the action in a clip by decreasing or increasing the number of frames. You can also use a timing curve to produce more complex timing effects. For example, you can reverse the action in a clip or vary the rate at which the action occurs.

To create a timewarp:

1. Click the TimeWarp button in the Editing menu.

2. Select the source clip.

The TimeWarp menu appears.



3. Set the range for the mix in the Mix Range field.

When the timewarped clip appears, each frame is mixed with its preceding and succeeding frames and produces a trail effect. The Mix Range field specifies the range of frames used in the mix. Use a higher value to produce more of a trail effect.



This triangle moves across the screen from left to right. The following examples illustrate the difference between mix values of 5 and 25.



This is the same frame of the animation with a mix value of 5.



This is the same frame of the animation with a mix value of 25.

4. Set the length of the timewarped clip:

- To adjust the source clip to the specified number of frames, enter a number of frames in the Frames field.
- To play back the source clip at the specified frame rate, enter a frame rate in the FPS field.
- To measure the timewarp as a percentage of the source clip, enter a percentage in the Speed field. A speed of 100% has no effect on the source, while a speed of 50% results in a timewarped clip that is twice as long as the source (and plays back half as fast).

NOTE: Changing a value in one field causes the other fields to update automatically.

5. Select a timewarp direction.

Select:	To:
+ve	Play the timewarp in a forward direction.
-ve	Play the timewarp in reverse.

6. Click the Frame/Field option box to switch between rendering in frames or fields.

7. The Render Mode box affects when timewarps are rerendered.

Select:	To:
Rendered	Always render timewarps.
Unrendered	Never render timewarps. Timewarps will remain hidden until Force Render is selected. Select Unrendered to edit transitions in the Soft Clip Editor without having to wait for timewarps to be rendered.

NOTE: For more information, see “Render Mode Options” on page 423.

8. If you are not using the Same Clips option, select the source clip.

9. Select the destination.

If the “E” button is enabled, the Timewarp Editor appears (see “Modifying Timewarps in the Timewarp Editor” on page 444). Otherwise, a new timewarp is added to the destination.

Aborting Timewarps and Unrendered Frames

You can abort a timewarp as it is being generated by clicking anywhere on the screen. The frames that were not generated are replaced by frames with the message “Unrendered Frame”.

For example, if you abort a 10-frame timewarp after the first 5 frames, the first 5 frames of the timewarp are followed by 5 black frames with the message “Unrendered Frame”.

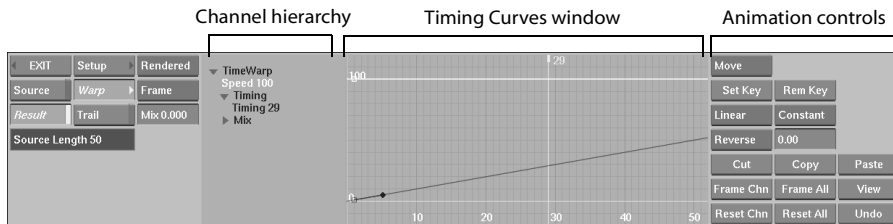
Frames with the “Unrendered Frame” message also occur when you lack framestore space. For example, when reprocessing a soft edit containing a timewarp, rather than changing the timing of the entire soft edit, “Unrendered” frames are substituted for the timewarped frames.

Modifying Timewarps in the Timewarp Editor

Use the Timewarp Editor to add more complex timing effects to a clip. For example, you can cause the action in a clip to accelerate, decelerate, or move in reverse.

To access the Timewarp Editor:

1. Click the Timewarp button.
2. Select the source clip.
An “E” (for Editor) appears on the TimeWarp button.
3. Click the TimeWarp button a second time.
4. Select the destination.
The Timewarp Editor appears.



Constant and Variable Timewarps

Constant timewarps have a continuous speed. In the Timewarp Editor, the timing channel for a constant timewarp is a straight line, containing exactly one keyframe. Once you grab a hermite handle and create a curve, the timewarp converts to a variable timewarp.

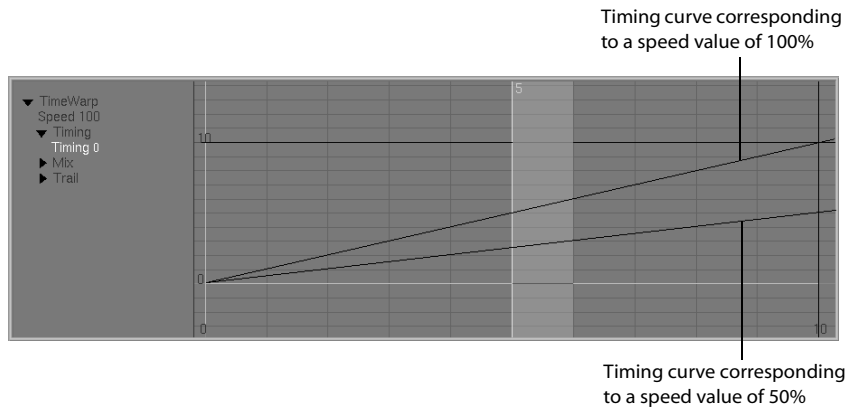
Constant timewarps can be edited from the EditReel or through the Speed field in the Timewarp menu.

Variable timewarps contain any number of keyframes. To edit the speed and curve of a variable timewarp, manipulate the timewarp channel. After you create a variable timewarp, it can be identified in the EditReel by a “*” in place of the element’s head and tail information.

Speed Curves Versus Timing Curves

Timewarps are created in one of two ways: by modifying the timing of the source clip, or by changing the playback speed of the source clip. When you make changes to a clip’s timing, you are essentially changing the time that it takes for a clip to reach a given frame. Take, for example, a 100-frame clip. You can adjust the timing so that only the 50th source frame is reached by the end of the clip’s 100 frames. This effectively halves the clip’s playback rate. A negative playback rate can be achieved by reversing a clip’s timing curve (by changing the value at the last keyframe to the value of the first keyframe and the other way around).

A steeper timing curve results in a faster playback rate.



Adjusting the clip's speed has different results. Setting the speed to 200% doubles the playback rate. A speed value of 50% plays the clip at half speed (or in slow motion). A negative speed value corresponds to playing the clip in reverse (where a -100% speed value creates a clip that plays back at the normal rate, but backward).

In the Timing Curves window, you can see that the speed curve and timing curve are, by definition, dependent on each other. You cannot change the speed without affecting the timing. Any action performed on one curve results in a corresponding action on the other. If you add a keyframe to the speed curve, then a keyframe is automatically added to the timing curve at the same frame.

To use the Timewarp Editor:

1. Specify the number of frames in the new clip in the Total Frames field (the field on the right beneath the image window controls).
2. Click the Frame /Field option box to switch between rendering in frames or fields.
3. Set the range for the mix in the Mix Range field.
4. Use the animation controls to modify the speed or timing curves by adding and moving keyframes. You can grab and drag the timebar to move to different positions in the timewarp.
5. Click Result to display the result clip in the image window. Click the ► button to preview the result of applying the timing curve to the source clip.
6. When you are satisfied with the results, click Process to process the clip. The generated clip appears on the destination reel.

Adding a Trail to the Image

You can add a trail to the image that precedes and/or follows the object.

- A pretrail mixes the frames that follow the current frame with the current frame.
- A posttrail mixes the frames that precede the current frame with the current frame.

The number of frames used in the mix is set with the pretrail and posttrail timing curves. Since you can change the shape of the timing curves, the trail values can be varied in the clip. When you use a pretrail or a posttrail, the Mix value is not used.

Setting the Fall-off for the Trail

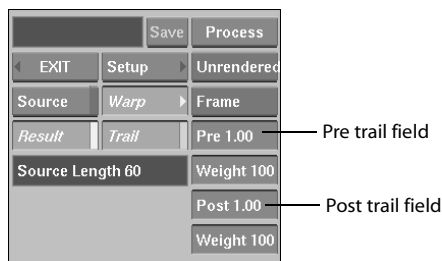
The Weight field specifies the fall-off for the image trail. There is one Weight field for each Pre trail and Post trail option.

- When the Weight is set to 100%, there is no fall-off. The first image in the trail has the same intensity as the image in the current frame.
- At 0%, the image in the current frame is at 100% intensity and the first image in the trail is at 0% intensity.

To add a pretrail or a posttrail to the image:

1. Enable the Trail button.

The Pre trail and Post trail fields appear.



2. Click the Pre trail or the Post trail.

The timing curve for the selected trail appears in the Timing Curves window.

3. Use the animation controls to set the timing curve for the trail by adding and moving points on the curve.
4. Set the fall-off value for the trail in its respective Weight field.
5. Click Result to display the result clip in the image window. Click the ► button to preview the result of applying the timing curve to the source clip.
6. When you are satisfied with the results, click Process to process the clip. The generated clip appears on the destination reel.

Display Options in the Setup Menu

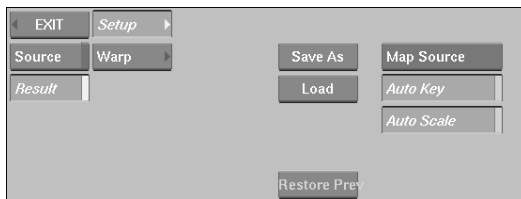
In the Timewarp Setup menu, two options control which frame is displayed in the image window while in Source mode. These options do not affect the processed clip.

- With the Map Source option selected the current frame number is interpreted as a position within the result clip. This value is scaled to display the corresponding frame in the source clip.
- With the Don't Map Source option selected, the current frame number is interpreted directly as the frame number within the source clip.

The Setup menu also provides an Auto Key option. Keyframes can be set automatically by the system or explicitly by the user. Enable the Auto Key button to set keyframes automatically. To manually set the keyframes, disable the Auto Key button and set the keyframes with the Set Key button in the animation controls. For more information, see “Using the Channel Editor” on page 129.

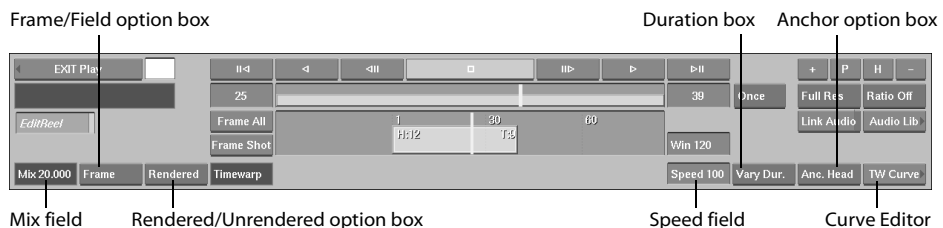
The Auto Scale function is used to automatically scale the timing curve in order to reflect changes to the timewarp duration.

Click the Setup button to display the Timewarp Setup menu. Click the Warp button to return to the Timewarp Editor.



Modifying Timewarps in the EditReel

After creating a timewarp, you can use the EditReel to change its speed as a constant value, timing, and length. Centre the timewarp over the play controls in the EditReel (an element that is lighter grey than other elements) to make the timewarp controls appear in the EditReel.



To modify a timewarp in the EditReel:

1. Load the clip containing the timewarp into the Player.
2. Select EditReel from the EditReel view box.
The EditReel appears.
3. Position any part of the timewarp (represented by the light grey element) at the center point of the EditReel.
The timewarp controls appear.
4. From the timewarp controls, you can modify Rendering options, Speed, and Anchor options.
5. If you want to create a variable speed timewarp, click the TW Curve button to open the Timewarp Editor.

Timewarp Controls

The following timewarp controls are found in the EditReel.

TW Curve button — Click to create complex timing effects. The Timewarp Editor appears. Use the Timewarp Editor as described in “Modifying Timewarps in the Timewarp Editor” on page 444.

Note that when you enter the Timewarp Editor from the EditReel, it differs slightly from when you enter the Timewarp Editor from the desktop:

- The duration and anchor settings from the EditReel timewarp controls are carried into the Timewarp Editor. Controls appear in the Timewarp Editor to modify these settings.
- There is no Process button. To process the result of your Timewarp Editor session, click the Exit button. The timewarp is processed and you are returned to the EditReel. To exit the Timewarp Editor without processing, *press and hold* the Exit button and select the Cancel option that appears. You are returned to the EditReel.
- The Mix value can only be modified after the TW Curve button has been clicked in the EditReel. Each frame is mixed with its preceding and succeeding frames, producing a trail effect. The number of frames used to mix the current frame with its preceding and succeeding frames is specified in the Mix field.

Speed Field — Indicates the playback speed as a percentage value. A speed value of 200% represents a clip that plays back twice as fast, while a speed of 50% represents a clip that plays half as fast.

Anchor Option Box — Maintains a match frame or locks the position of a frame whenever you alter the timewarp speed.

Select:	To:
Anchor Head	Lock the first frame of the head as the first frame of the timewarp regardless of any heads or tails that are created when you adjust the timewarp speed or length in the EditReel.
Anchor First	Lock the first frame of the clip as the first frame of the timewarp regardless of any heads or tails that are created when you adjust the timewarp speed or length in the EditReel.
Anchor Tail	Lock the last frame of the tail as the last frame of the timewarp regardless of any heads or tails that are created when you adjust the timewarp speed or length in the EditReel.
Anchor Last	Lock the last frame of the clip as the last frame of the timewarp regardless of any heads or tails that are created when you adjust the timewarp speed or length in the EditReel.

Duration Box — Locks or varies the duration of the timewarp in relation to speed changes.

This image shows a full page of white paper with horizontal dashed lines. The lines are evenly spaced and run across the width of the page, providing a guide for handwriting practice. There are no margins, text, or other markings on the page.

flame uses SGI audio or the Discreet Audio subsystem to work with digital audio.

Summary

In this chapter, you learn about:

- “Starting flame with Audio” on page 452
- “Using SGI Audio” on page 452
- “Sync Issues” on page 461
- “Using Discreet Audio” on page 464
- “Audio Timewarps” on page 481
- “Audio Sparks” on page 483

About Audio

Using **flame**’s digital audio capabilities, you can record, load, and play a two-track audio file simultaneously with a clip. Tools are available for slipping, trimming, and scrubbing. A visual waveform and timeline help you to ensure the audio and video are in sync. If you are using Discreet Audio, you can edit the EQ of the audio. You can also add audio dissolves at the same time you add video dissolves.

There are two types of digital audio available: SGI and Discreet Audio (Sonic Solutions).

SGI audio is a standard audio card that requires no additional hardware for operation within the software.

Each system has different capabilities, so consult the section in this chapter for the digital audio system you are using.

flame supports import and export of the following audio file formats:

- Extended Audio Interchange File Format (AIFF-C) (.aifc)
- Audio Interchange File Format (AIFF) (.aiff)

- NeXT .snd and Sun .au
- Waveform Audio File Format (RIFF) (.wav)
- Berkeley/IRCAM/CARL Sound File format (.bsf)
- MPEG-1 Audio Bitstream encoded data (.mp1)
- Sound Designer File Format II (.sd2)
- Audio Visual Research File Format (.avr)
- Amiga IFF/8SVX File Format (.ami)
- Creative Labs VOC File Format (.voc)
- Sample Vision File Format (.svf)
- Creative Labs SoundFont2 File Format (.sf2)

Starting flame with Audio

If you have an audio system installed, you need to start **flame** with an additional command to initialize the audio subsystem when you start the software.

NOTE: The audio keywords must be specified in the config file. For more information, refer to the *flame Installation Guide*.

To start flame using audio:

1. Log in to your system as usual.
2. In a UNIX shell, type `flame_da` or `flames`.
flame starts, with the Audio subsystem running.

NOTE: If you need to kill **flame** when Discreet Audio is running, use the `kfs` command, not `kf`. For more information, see “Using the “Kill Process” Command” on page 20.

Using SGI Audio

The following sections describe how to use **flame** with SGI Audio.

Linking Audio to a Clip

You can use the AudioDesk to load or record an audio track and link the audio track to a clip. Linking audio to a clip creates a reference, or pointer, from the clip to the audio track. Any audio file can be linked to many video clips simultaneously.

The audio track is stored in the framestore partition. Since pointers are used to link the audio and clip, linking several clips to the same audio track does not take additional space in the partition.

You can also use the Discreet Network to transfer an audio-only clip from an Editing product.

NOTE: If the audio file contains more than two tracks, only the first two audio tracks are imported into the Effects product.

An audio-only clip appears as black proxies on the desktop reel. You can link an audio-only clip to a video clip by using the Copy command to copy the audio to the video clip.

Importing an Audio File

You can use the AudioDesk to import an audio file format and link it to a clip. Audio cannot be loaded independently from video, so you must always associate imported audio with a video clip. You import audio files from the AudioDesk.

To import an audio file from the AudioDesk, you access the audio file browser and then select the audio file to load. When you return to the AudioDesk, the clip you selected is linked to the audio file.

To import an audio file:

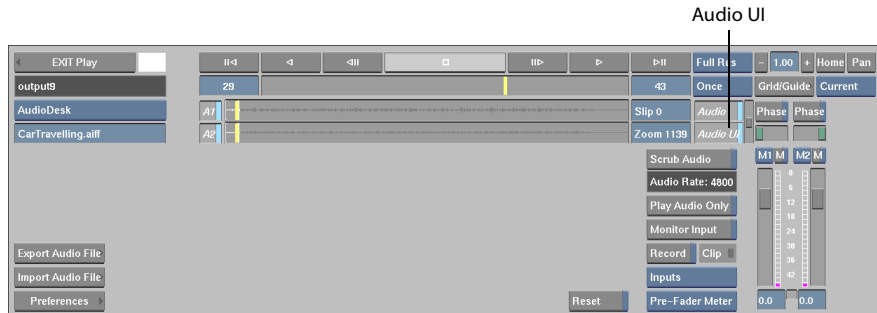
1. Center the clip on the reel and click Play.
The clip appears in the Player.
2. Select AudioDesk in the EditReel view box.

EditReel view box



3. Click Import Audio File.
The audio file browser appears.
4. Select the file format from the Format box.
NOTE: For a CD, type /CDROM in the Directory Name field.
5. Select the audio file from the file browser.
6. Click Load.
The audio track is loaded and appears in the AudioDesk.

7. To view the waveforms, enable Audio UI.



Importing Tracks from a CD

You can import tracks from a music CD if your system has a CDROM drive. CD tracks are imported as AIFF files. When you import an audio file into the system, the audio files are automatically re-sampled to the sampling rate specified in the Audio Preferences menu.

To import music from a CD:

1. From a UNIX shell, turn on the mediad utility by typing the following command at the prompt:
mediad -a
2. At the prompt type:
ls /CDROM
The tracks on the CD are listed as track1.aiff, track2.aiff, etc.
3. Follow the instructions in “Importing an Audio File” on page 453, selecting AIFF as the file format.
4. When you have finished importing the files, open a UNIX shell and turn off the mediad utility by typing:

mediad -k

NOTE: It is important that you turn mediad off when you are finished importing from a CD. If mediad remains on, you may experience dropped frames during clip I/O.

Removing Audio Files

You can use the audio file browser to remove audio files from the file system. You only need to access the audio file browser and select the audio file(s) to remove.

To remove an audio file:

1. Center any clip on the reel and click Play.
The clip appears in the Player.

2. Select AudioDesk in the EditReel view box.

3. Click Import Audio File.

The audio file browser appears.

4. Select the file format from the Format box.

5. Select the audio file(s) from the file browser.

6. Click Remove.

The audio track(s) are removed from the file system.

NOTE: You can remove multiple files at the same time as long as they are of the same file type. If you want to remove WAV files and AIFF files, you must remove them separately.

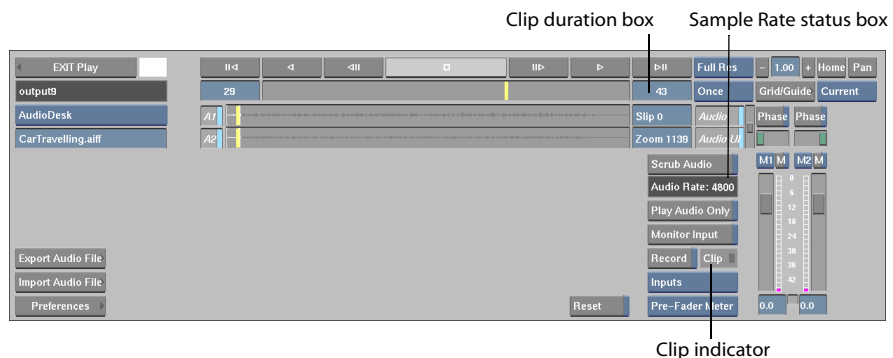
Recording Audio

You can use the AudioDesk to record audio and link the audio to a clip. Use the following procedure to record audio from any of four input sources. For more information on the record options, see “Record and Playback Options” on page 456.

To record audio:

1. Select AudioDesk in the Player.

The AudioDesk menu appears.



2. Specify the audio input source, sampling, rate and other audio preferences in the Preferences menu. See “SGI Audio Preferences” on page 459.
3. If you want to listen to the audio from the input source while recording, enable the Monitor Input button.
4. Adjust the output level at the source so that the clip indicator does not flash while recording.

NOTE: The clip indicator only tests the input level while recording. For more information, see “Clip Indicator” on page 457.

5. Click Record to begin recording the audio from the input source.
6. Click anywhere to end recording.

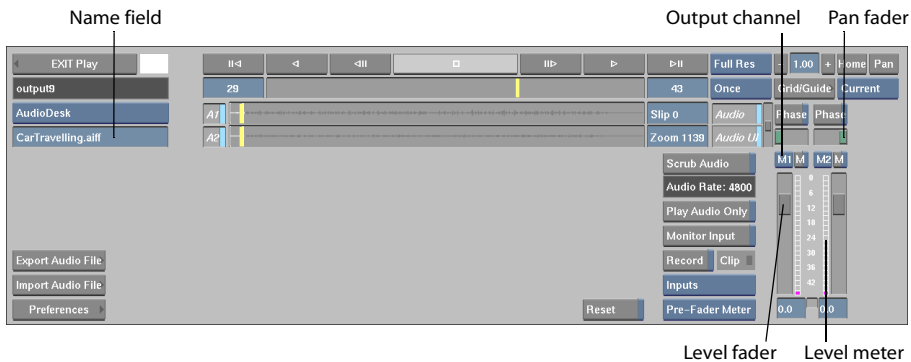
If you have waveforms enabled in the Audio Preferences menu, the waveform for the recorded audio is shown in the Audio Track window. Depending on the size of the audio file, it may take some time to build the waveform.

7. The name of the input source is shown in the Name field. If you want to change the name of the audio track, click the Name field and use the keyboard to enter a new name.
8. Return to the desktop or select a different view mode in the Player such as EditReel or Setup, then click Confirm to link the audio to the selected clip and return to the desktop reels. Click anywhere other than the Confirm button to abort linking the audio file.

You can play the clip with its audio track using the Player. For more information, see “Playing Clips with Audio” on page 463.

Record and Playback Options

Use the controls in the AudioDesk to select the record and playback options.

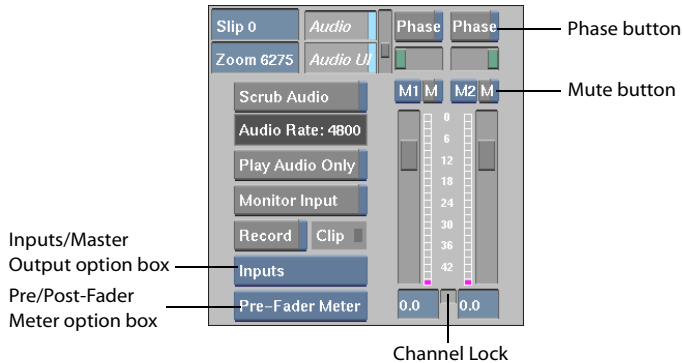


Name field — When you record or load an audio track, the name of the file or input source is used by default. To rename the audio track, click the Name field and use the keyboard to enter a new name.

Sampling Rate Status Box — Indicates the number of samples taken per second to convert the audio signal into a digital signal. You will get better sound quality if you use a higher sample rate, but more memory will be required to record or play the audio track. For more information on memory requirements for recording audio, see the *flame Installation Guide*.

Typically, a sample rate of 44.1 kHz is used to record from CD format, and a sample rate of 48 kHz is used to record from a DAT or digital VTR.

Clip Indicator — Flashes whenever the input level is too high during recording. Audio tracks recorded with the clip indicator light flashing may contain distortion. To prevent the clip indicator from flashing, lower the output level of the audio source.



Input Level Meters — Indicate the input level during recording. When playing back a clip, the level meters indicate the playback level.

Output Level Meters — Indicate playback volume. Adjust the percentage in this display to control the volume when playing the audio track and when monitoring audio input before recording.

Level Faders — Control the input or output gain.

Pre/Post Fader Meters — View signal before or after the input fader adjustment. For example, if you boosted the input gain, you may want to hear the signal at its original level, or after your input gain adjustment. Monitor the Pre-Fader Meter to hear the original signal, or the Post Fader Meter to hear the signal after it has been affected by adjustments to the Input Level Fader.

Inputs/Master Output — Monitors either the signal coming into the system, or the signal being output.

Output Channel — Reassigns the output channel.

Pan Fader — Pans the output signal between left and right channels.

Mute Button — Temporarily disables playback of a track.

Phase Button — Inverts the phase of the track. For example, if phase errors are introduced, invert the phase for one track only.

Monitor Input — Enable to monitor the input while recording.

NOTE: Disable the Monitor Input button after you finish recording the audio track. Feedback may occur if the button remains enabled when the audio is played.

Scrubbing Audio

You can scrub the audio to accurately locate features of the waveform.

To scrub audio:

1. Enable the Scrub Audio button.
2. Grab the positioner in the play bar.
3. Drag the positioner left or right to scrub the audio.

The audio scrubs at the speed you drag. You cannot scrub faster than normal playback speed.

NOTE: If the Scrub Audio button is disabled, you can use the **CTRL** key and drag the positioner to temporarily scrub the audio.

Scrubbing in Modules

If you load a clip with audio into a module, such as Text or Paint, you can scrub the audio by holding **CTRL** while you drag the positioner. You can also view the waveform for one audio channel in many modules.

To scrub audio in a module:

1. Load a clip with audio into the module.
2. Make sure Playlock is disabled.
3. While holding the **CTRL** key, drag the positioner below the image window. The audio scrubs as you drag.

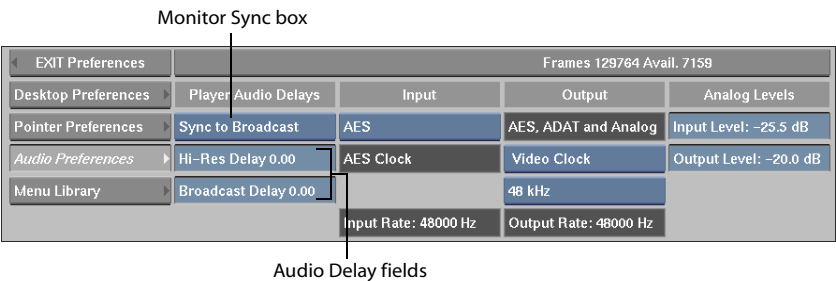
To view a waveform in a module:

1. Go to the Setup menu.
 2. In the Wave box, select Audio 1, or Audio 2.
- The waveform appears below the player controls.

SGI Audio Preferences

Click the Audio Preferences button to set the input and output levels of the analog device, to set the input and output sync sources, to set the input source and the Player delay, and to set the input and output sampling rate.

You can access the Audio Preferences menu from the AudioDesk, the Engineering menu, or the System menu.



Monitor Sync box

Use the Monitor Sync box to compensate for audio sync differentials between the broadcast monitor and the high-resolution monitor. (Not all systems introduce audio sync differentials between monitors.)

The Sirius video board, for example, introduces a two-frame difference in video track display between the broadcast monitor and the high-resolution monitor. Use the Monitor Sync box to determine which monitor will have audio sync when you play back a clip. If necessary, specify the audio delay in the corresponding broadcast or high-resolution monitor.

When the Sync to Broadcast option is enabled, audio is synchronized to the broadcast monitor during playback. When the Sync to Hi-Res option is enabled, audio is synchronized to the high-resolution monitor during playback.

Setting Audio Delay

You can specify the number of frames to delay the audio signal when syncing to either the broadcast or high-resolution monitor. Use this delay to compensate for timing delays introduced by adding circuit switching or by audio or video devices attached to your **flame** system. Enter the new value in the field for the monitor you want to change.

Use:	To:
Broadcast Delay box	Set the number of frames (-30.00 to 30.00) to delay the audio signal when syncing to the broadcast monitor.
Hi-Res Delay box	Set the number of frames (-30.00 to 30.00) to delay the audio signal when syncing to the high-resolution monitor.

Input Source

Specify the type of input you are using.

Select:	To:
AES	Record audio using the AES input.
ADAT	Record audio using the optical ADAT input.
Line	Record audio using the analog input.
Microphone	Record audio using the analog microphone input.

Once you select the audio source, specify the sampling rate you want to use.

Input Sync Source

This option box indicates the input clocks available depending on the input source for which you will receive the clocking signal for audio input. If you select ADAT as your I/O connection, **flame** automatically uses the clocking signal derived from the ADAT device during audio input. If you are using AES as your I/O connection, the software automatically uses the clocking signal derived from the AES source during audio input. If you are using either Line or Microphone, you need to select one of the following sync sources.

Select:	To:
Video Clock	Use the 48 kHz signal from the VTR.
AES Clock	Lock to the signal coming from the AES source.
ADAT Clock	Lock to the signal coming from the ADAT source.
Internal Clock	Lock to the internally generated signal.

Once you specify the input sync source, specify the sampling rate.

Input Sampling Rate

Depending on the Input Sync Source you selected, you may be able to define the sampling rate. This value specifies the sampling rate that is used to create the digital audio signal during recording.

Analog Levels

You can adjust the gain of the analog signal for both input and output. For example, if the signal being input is not loud enough, you can boost the input signal gain in the Input Level box. Or, if the signal being sent to your external device is too loud, you can reduce the output signal gain in the Output Level box.

Outputs

Audio is sent to all available outputs. For example, if your system configuration includes AES, ADAT and analog outputs, the audio signal is output to all of those outputs. This status box indicates all available outputs.

Output Sync

When outputting audio, specify the source for output sync.

Select:	To:
Video Clock	Use the VTR as a sync source.
AES Clock	Use the signal derived from the AES input. (A device must be connected.)
ADAT Clock	Use the signal derived from the optical ADAT input.
Internal Clock	Use the Internal Clock as a sync source.

Sync Issues

There are some sync issues to consider if you are using SGI audio.

Audio Sampling Rates

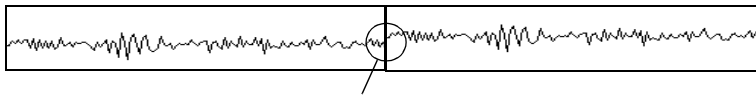


WARNING: When you import audio into the system, it is automatically resampled to the sampling rate defined in the Audio Preferences menu. However, if you work in several sessions at different sampling rates, it is possible to have many clips on your desktop at different sampling rates. You should avoid splicing together audio clips that have different sampling rates.

NOTE: This only applies to SGI Audio. Discreet Audio only runs at 48,000 Hz or 44,100 Hz.

Avoiding Audio Clicks and Pops (SGI Audio)

If you splice many audio clips together, you may experience clicks or pops in the audio at the splice points after you have committed the clip. This is due to the fact that the waveforms are not always at the same location on either side of the cut, so there is a slight “jump” in the waveform that results in a click or pop as in the following illustration:



The waveforms are not in the same location at either side of the splice point. This results in a click or pop. Add an audio dissolve to smooth the transition between the two waveforms.

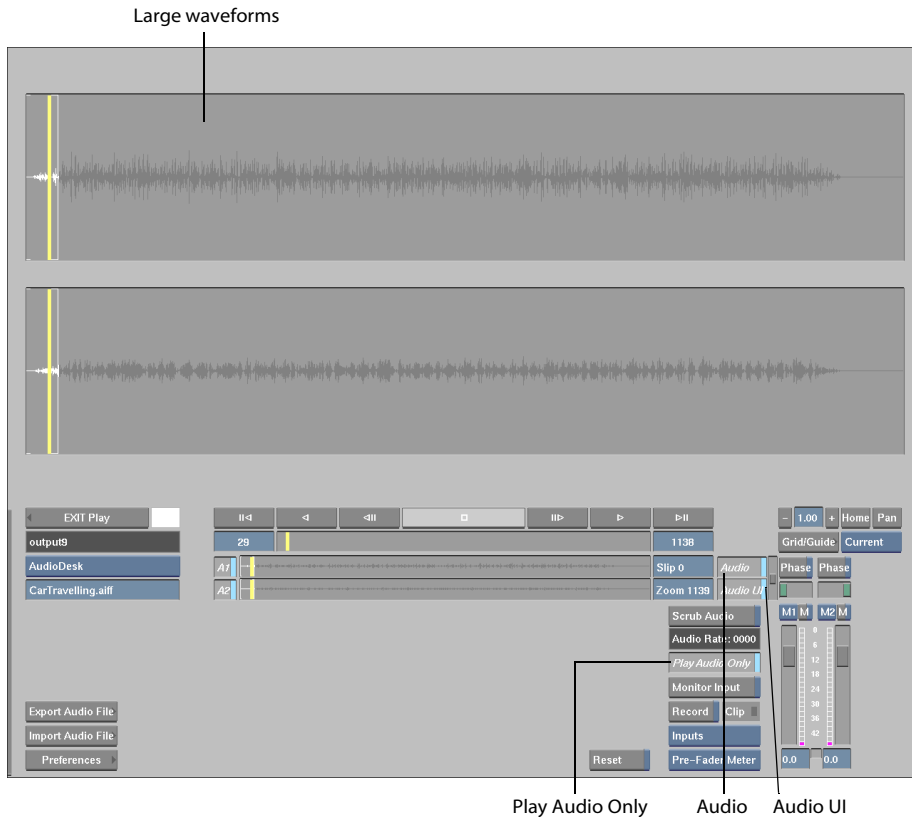
To work around this problem, you should add a dissolve on the audio tracks at the splice points. A one-frame dissolve should be sufficient to eliminate any unwanted clicks and pops.

You may also hear clicks and pops if you have mixed audio files with different sampling rates. Make sure you are always working with the same sampling rate.

NOTE: This problem does not apply to Discreet Audio because the Discreet Audio system automatically creates crossfades at the splice points during playback and commit.

Working with Waveforms

The tools in the AudioDesk can be used to view the waveform and slip the audio. For example, you can view the video in and out points on the waveform and slip the audio for more precise timing with the audio.



Play the audio track using the controls beneath the Audio Track window.

Audio

Enable to hear audio during playback.

Audio UI

Enable to view the waveforms beneath the image window controls.

Play Audio Only

Enable to play back only the audio. This also displays the large audio waveforms in place of the clip in the image window.

The Audio Waveform

The audio waveform appears below the image window controls when the Audio UI button is enabled. The left channel appears on the top, and the right channel appears on the bottom. You can view large waveforms by enabling the Play Audio Only button.

The space between two dotted lines represents one frame. The first frame in the clip has an open bracket and the last frame has a closed bracket. These brackets are used to indicate how the audio track is slip synced in relation to the beginning and end of the clip.

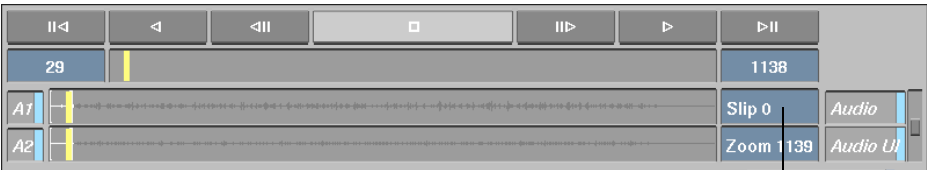
Zoom Box

Use the value in the Zoom box to change the zoom factor of the waveform. Increase the value to increase the zoom factor of the waveform. Decrease the value to decrease the zoom factor of the waveform.

Slipping Audio

The Slip box specifies the offset between the beginning of the audio track and the clip to which it is linked. For example, an offset of -6 frames begins playing the audio track six frames after the beginning of the clip. An offset of 6 frames begins the audio track six frames before the beginning of the clip. This means that the audio track is slip synced six frames.

Slipping the audio track is useful if, for example, your clip fades in and you want the audio track to begin a number of frames into the clip.



Change the value in the Slip box to offset the audio tracks.

Playing Clips with Audio

You can play the audio track linked to a clip using the AudioDesk in the Player. When Audio is enabled, the audio track and the clip play simultaneously. To hear only the audio, enable Play Audio Only. To view the video without the audio playback, disable the Audio button.

To play the audio track linked to a clip:

1. Centre the clip with the audio track on its desktop reel.
2. Click the Play button.
The Player appears.
3. Select AudioDesk.
4. Enable the Audio button to enable audio playback.

- Click the Audio UI button to display the audio options.



- To scale the waveform display, change the value in the Zoom box.
- To slip the audio, change the value in the Slip box.
- Enable the Meters button to view the output meters.
- Click the ► button to play the audio with the clip.

To synchronize audio when playing the clip at any frame rate, enable the A & V Synchronized option in the Setup menu. For more information, see “Maintaining Audio or Video Playback Fidelity (A&V Synch Box)” on page 80.

Using Discreet Audio

The following sections describe how to use **flame** with Discreet Audio.

Linking Audio to a Clip

You can use the AudioDesk to load or record an audio track and link the audio track to a clip. Linking audio to a clip creates a reference, or pointer, from the clip to the audio track. Any audio file can be linked to many video clips simultaneously.

The audio track is stored in the audiostream and is mirrored to the framestore partition. Since pointers are used to link the audio and clip, linking several clips to the same audio track does not take additional space in the partition.

You can also use the Discreet Network to transfer an audio-only clip from an Editing product.

NOTE: If the audio file contains more than two tracks, only the first two audio tracks are imported into the Effects product.

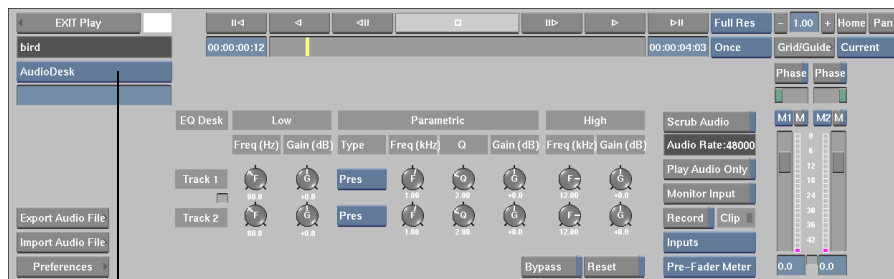
Importing an Audio File

You can use the AudioDesk to import an audio file format and link it to a clip. Audio cannot be loaded independently from video, so you must always associate imported audio with a video clip. You import audio files from the AudioDesk.

To import an audio file from the AudioDesk, you access the audio file browser and then select the audio file to load. When you return to the AudioDesk, the clip you selected is linked to the audio file.

To import an audio file:

1. Center the clip on the reel and click Play.
The clip appears in the Player.
2. Select AudioDesk in the EditReel view box.

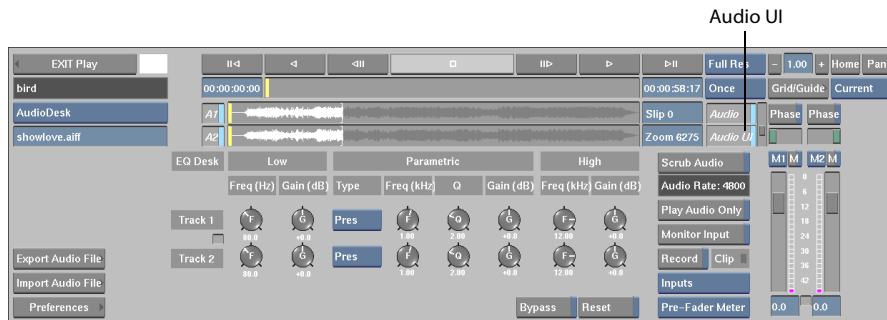


EditReel view box

3. Click Import Audio File.
The audio file browser appears.
4. Select the file format from the Format box.

NOTE: For a CD, type /CDROM in the Directory Name field.

5. Select the audio file from the file browser.
6. Click Load.
The audio track is loaded and appears in the AudioDesk.
7. To view the waveforms, enable Audio UI.



Audio UI

Importing Tracks from a CD

You can import tracks from a music CD if your system has a CDROM drive. CD tracks are imported as AIFF files. When you import an audio file into the system, the audio files are automatically re-sampled to the sampling rate specified in the Audio Preferences menu.

To import music from a CD:

1. From a UNIX shell, turn on the mediad utility by typing the following command at the prompt:
mediad -a
2. At the prompt type:
ls /CDROM
The tracks on the CD are listed as track1.aiff, track2.aiff, etc.
3. Follow the instructions in “Importing an Audio File” on page 453, selecting AIFF as the file format.
4. When you have finished importing the files, open a UNIX shell and turn off the mediad utility by typing:
mediad -k

NOTE: It is important that you turn mediad off when you are finished importing from a CD. If mediad remains on, you may experience dropped frames during clip I/O.

Removing Audio Files

You can use the audio file browser to remove audio files from the file system. You can remove audio files from the AudioDesk.

To remove an audio file, you only need to access the audio file browser and then select the audio file(s) to remove.

To remove an audio file:

1. Center any clip on the reel and click Play.
The clip appears in the Player.
2. Select AudioDesk in the EditReel view box.
3. Click Import Audio File.
The audio file browser appears.
4. Select the file format from the Format box.
5. Select the audio file(s) from the file browser.
6. Click Remove.
The audio track(s) are removed from the file system.

NOTE: You can remove multiple files at the same time as long as they are of the same file type. For example, if you want to remove WAV files and AIFF files, you must remove them separately.

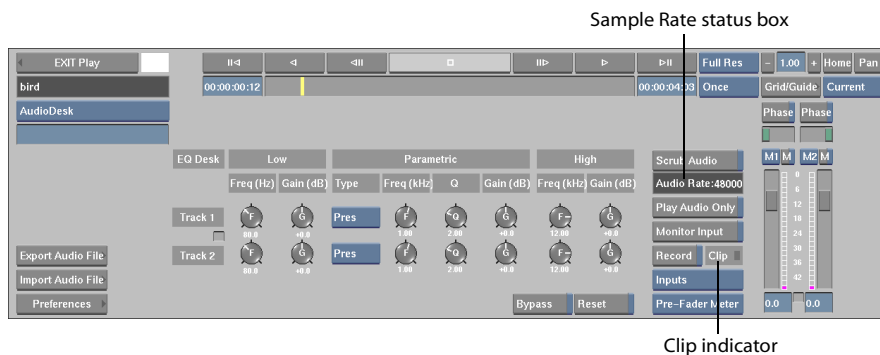
Recording Audio

You can use the AudioDesk to record audio and link the audio to a clip. Use the following procedure to record audio from any of four input sources. For more information on the record options, see “Record and Playback Options” on page 468.

To record audio:

1. Select AudioDesk in the EditReel view mode box.

The AudioDesk menu appears.



2. Specify the audio input source, sampling rate, and other audio preferences in the Preferences menu. See “Discreet Audio Preferences” on page 471.
3. If you want to listen to the audio from the input source while recording, enable the Monitor Input button.
4. Adjust the output level at the source so that the clip indicator does not flash while recording.

NOTE: The clip indicator only tests the input level while recording. For more information, see “Clip Indicator” on page 457.

5. Click Record to begin recording the audio from the input source.
6. Click anywhere to end recording.

If you have waveforms enabled in the Audio Preferences menu, the waveform for the recorded audio is shown in the Audio Track window. Depending on the size of the audio file, it may take some time to build the waveform.

7. The name of the input source is shown in the Name field. If you want to change the name of the audio track, click the Name field and use the keyboard to enter a new name.

8. Return to the desktop or select a different view mode in the Player such as EditReel or Setup, then click Confirm to link the audio to the selected clip and return to the desktop reels. Click anywhere other than the Confirm button to abort linking the audio file.

You can play the clip with its audio track using the Player. For more information, see “Playing Clips with Audio” on page 479.

Record and Playback Options

Use the controls in the AudioDesk to select the record and playback options.

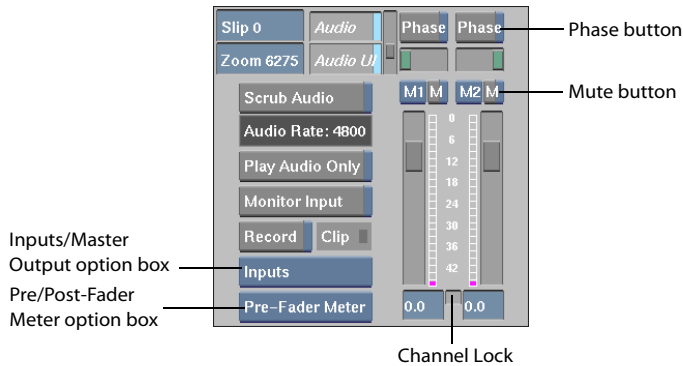


Name field — When you record or load an audio track, the name of the file or input source is used by default. To rename the audio track, click the Name field and use the keyboard to enter a new name.

Sampling Rate Status Box — Indicates the number of samples taken per second to convert the audio signal into a digital signal. You will get better sound quality if you use a higher sample rate, but more memory will be required to record or play the audio track. For more information on memory requirements for recording audio, see the *flame Installation Guide*.

Typically, a sample rate of 44.1 kHz is used to record from CD format, and a sample rate of 48 kHz is used to record from a DAT or digital VTR.

Clip Indicator — Flashes whenever the input level is too high during recording. Audio tracks recorded with the clip indicator light flashing may contain distortion. To prevent the clip indicator from flashing, lower the output level of the audio source.



Input Level Meters — Indicate the input level during recording. When playing back a clip, the level meters indicate the playback level.

Output Level Meters — Indicate playback volume. Adjust the percentage in this display to control the volume when playing the audio track and when monitoring audio input before recording.

Level Faders — Control the input or output gain.

Pre/Post Fader Meters — View signal before or after the input fader adjustment. For example, if you boosted the input gain, you may want to hear the signal at its original level, or after your input gain adjustment. Monitor the Pre-Fader Meter to hear the original signal, or the Post Fader Meter to hear the signal after it has been affected by adjustments to the Input Level Fader.

Inputs/Master Output — Monitors either the signal coming into the system, or the signal being output.

Output Channel — Reassigns the output channel.

Pan Fader — Pans the output signal between left and right channels.

Mute Button — Temporarily disables playback of a track.

Phase Button — Inverts the phase of the track. For example, if phase errors are introduced, invert the phase for one track only.

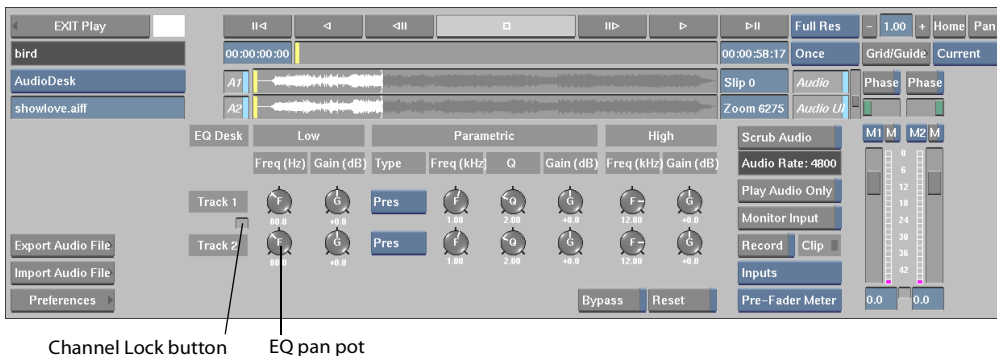
Monitor Input — Enable to monitor the input while recording.

NOTE: Disable the Monitor Input button after you finish recording the audio track. Feedback may occur if the button remains enabled when the audio is played.

EQ Controls — The EQ section is a 3-band EQ with adjustments for low, mid and high frequencies. The mid EQ is a parametric EQ. You can adjust the EQ for each clip that has audio.

Adjusting the EQ

Use the EQ controls to adjust the EQ for specific frequencies.



You can adjust the cut-off frequency and Gain for the High and Low Shelf portions of the audio signal. With the parametric filter you can select the centre frequency and adjust the gain and the Q using either a Presence or Notch filter.

To adjust the EQ:

1. Click on the pan pot for the Frequency, Gain, or Q value you want to adjust.
Two white arrows appear indicating the direction you can move the mouse or stylus.

2. Drag the pan pot to the new value.

The pan pot is highlighted green when it is a value other than the default setting.

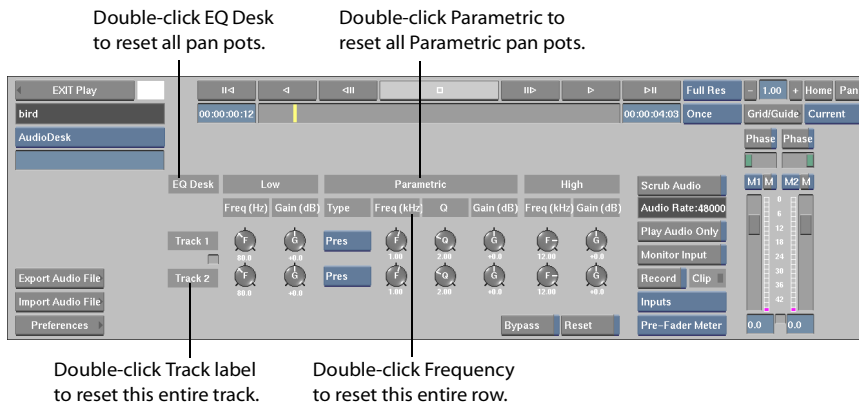
The new EQ settings are now applied to the track. To bypass an EQ setting, enable the Bypass button and click the pan pot for the setting you want to bypass. To bypass all EQ settings, enable the Bypass button and click EQ Desk. You can also bypass an entire row, or section of the EQ controls.

Resetting EQ Values

You can use the Reset option to reset the default EQ values for a single pan pot, a range of pan pots or the entire EQ desk. You reset a value by double-clicking it.

To reset EQ values:

1. Reset the EQ value you want.



To reset:

- An individual pan pot
- A row of EQ values
- All EQ values in a range
- All EQ values
- All EQ values for a single channel only

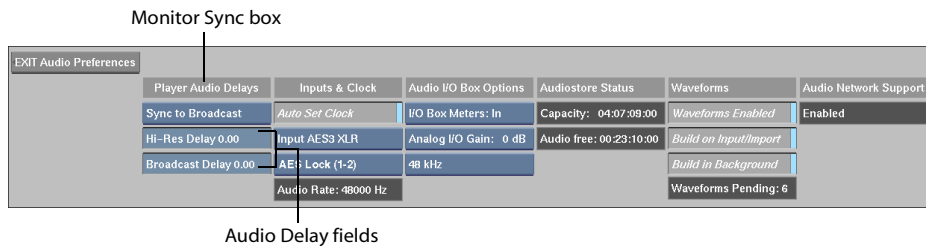
Double-click:

- The pan pot.
- Freq, Gain or Q.
- High, Parametric, or Shelf.
- EQ Desk.
- The channel number. If a channel is locked with another, both channels are reset.

Discreet Audio Preferences

Click the Audio Preferences button to set the input and output levels of the analog device, to set the input and output sync sources, to set the input source and the Player delays, and to set the input and output sampling rate. You can also view the audiostore and audio network status.

You can access the Audio Preferences menu from the AudioDesk, the Engineering menu, or the System menu.



Player Audio Delays

Use the Player Audio Delay options to control how audio is synced with video during playback.

Monitor Sync box

Use the Monitor Sync box to compensate for audio sync differentials between the broadcast monitor and the high-resolution monitor. (Not all systems introduce audio sync differentials between monitors.)

The Sirius video board, for example, introduces a two-frame difference in video track display between the broadcast monitor and the high-resolution monitor. Use the Monitor Sync box to determine which monitor will have audio sync when you play back a clip. If necessary, specify the audio delay in the corresponding broadcast or high-resolution monitor.

When the Sync to Broadcast option is enabled, audio is synchronized to the broadcast monitor during playback. When the Sync to Hi-Res option is enabled, audio is synchronized to the high-resolution monitor during playback.

Setting Audio Delay

You can specify the number of frames to delay the audio signal when syncing to either the broadcast or high-resolution monitor. Use this delay to compensate for timing delays introduced by adding circuit switching or by audio or video devices attached to your **flame** system. Enter the new value in the field for the monitor you want to change.

Use:	To:
Broadcast Delay box	Set the number of frames (-30.00 to 30.00) to delay the audio signal when syncing to the broadcast monitor.
Hi-Res Delay box	Set the number of frames (-30.00 to 30.00) to delay the audio signal when syncing to the high-resolution monitor.

Input Sync Source

Specify the input for the sync source. Refer to the table specific to your configuration. If you have 8 I/O-track support:

Select:	To:
Input AES3 XLR	Connect to a digital audio source from the XLR connectors.
Input SPDIF RCA	Connect to a digital audio source from the SPDIF RCA connector.
Input Analog	Connect to an analog audio source from the analog XLR connectors.

If you have 4 I/O-track support:

Select:	To:
Input AES3 XLR	Connect to a digital audio source from the XLR connectors.
Input SPDIF RCA	Connect to a digital audio source from the SPDIF RCA connector.
Input TOS Optical	Connect to a digital audio source through the Toslink interface. Use the AD/DA Converters for analog I/O.
Input Off	Disconnect the input (no audio source available).

For more information on input settings, refer to the *Discreet Audio Hardware Configuration Guide*.

Inputs and Clock

Use the Clock options to specify the type of sync and the sync source for Discreet Audio.

Enable the Auto Set Clock button to have **flame** attempt to reestablish sync if it is temporarily lost. For example, if Auto Set Clock is enabled, and the AES cable is momentarily removed from the VTR, **flame** will attempt to reestablish sync until the AES cable is returned. After a few attempts, **flame** will automatically revert to Internal Sync source if the AES source is still not present.

Specify the type of sync you are using.

Select:	To:
AES Lock (Channels 1-2)	Use the signal from AES channels 1-2 as the sync source. Use AES Lock for digital audio.
AES Lock (Channels 3-4)	Use the signal from AES channels 3-4 as the sync source. Use AES Lock for digital audio.
AES Lock (Channels 5-6)	Use the signal from AES channels 5-6 as the sync source. Use AES Lock for digital audio (for 8-track support only).
AES Lock (Channels 7-8)	Use the signal from AES channels 7-8 as the sync source. Use AES Lock for digital audio (for 8-track support only).

Select:	To:
Word Sync	Use Word Sync as a sync source. It is recommended to use Word Sync only when doing analog audio I/O.
Word Sync Inverted	Use the inverted Word Sync signal as a sync source. It is recommended to use Word Sync only when doing analog audio I/O.
Internal Clock	Use the internal clock on the Discreet Audio subsystem as a sync source. You cannot do I/O using the Internal Clock as a sync source.

Audio Rate Box

The Audio Rate box indicates the current sampling rate based on the clock and input settings. If the system is not in sync, a red LED display lights up on the Audio Rate box. You can select either 48,000 Hz or 44,100 Hz.

Audio I/O Box Options

Use the Audio I/O Box options to control the settings on the external Discreet Audio hardware.

I/O Box Meters

Specify which signal appears in the meters on the external Discreet Audio unit.

Select:	To:
I/O Box Meters: In	View the incoming audio signal on external Discreet Audio unit.
I/O Box Meters: Out	View the outgoing audio signal on the external Discreet Audio unit.

Analog I/O Gain

Change this value depending on the type of VTR you are using to input and output audio.

Select:	If:
0 dB	You are importing or exporting audio with a digital VTR.
-14 dB	You are importing or exporting audio with an analog VTR such as BetaSP.

Sampling Rate

If you are using Internal Clock for your sync source, you can specify the sampling rate for the external Discreet Audio unit. You can select 48 kHz or 44.1 kHz.

Audiostore Status

These status boxes inform you of the total capacity of your audiostore and the space remaining. You cannot modify the values in these boxes.

Waveforms

Use the waveforms control to specify when waveforms are built.

Audio Waveforms

You can specify when **flame** builds audio waveforms. Enable Waveforms Enabled to view waveforms in the AudioDesk.

Build on Import/Export

Build the waveforms during file import. If this option is disabled, waveforms are not built until you want to view them.

Build in Background

Enable to work in **flame** while the waveforms are being built. When you use this option, the Waveforms Pending status box appears to inform you how many waveforms have not yet been built in the background.

Audio Network Support

If you have the Discreet Network enabled, the Audio Network Support field displays “Enabled”. If you do not have the Discreet Network enabled, or if for some reason it is not configured properly, this field displays “Disabled”. The Network Support field only appears if you are using Discreet Audio.

When the Discreet Network is enabled, audio files are mirrored automatically from the audiostore to the framestore. Only audio that is mirrored on the framestore can be transferred with the Discreet Network. If the Network Support field displays “Disabled”, you cannot transfer audio using the Discreet Network. For information on configuring the Discreet Network, refer to the **flame** *Installation Guide*.

Discreet Audio and Sync

NOTE: The following section applies only if you are using Discreet Audio.

To ensure audio is properly synced to video, Discreet Audio must receive the clocking signal from House Sync. House Sync can be routed to Discreet Audio either through an AES audio source or a Video Ref-to-Word Sync converter. For information on wiring your Discreet Audio for House Sync, refer to the *Discreet Audio Hardware Configuration Guide*.

By default, Discreet Audio locks to the AES signal it receives from the AES audio source connected to the system. To lock to House Sync, the AES signal must originate from an audio device that supplies an AES signal derived from House Sync. It is recommended that you use a professional audio device with genlock support such as a digital VTR, genlock DAT, or genlock CD-player. If your Discreet Audio system is wired for House Sync through an AES audio source, refer to “Configuring Discreet Audio for House Sync” on page 476.

To lock to Word Sync, you must convert the House Sync to Word Sync by connecting the signal to a Video Ref-to-Word Sync converter (not included with your Discreet Audio system). Once

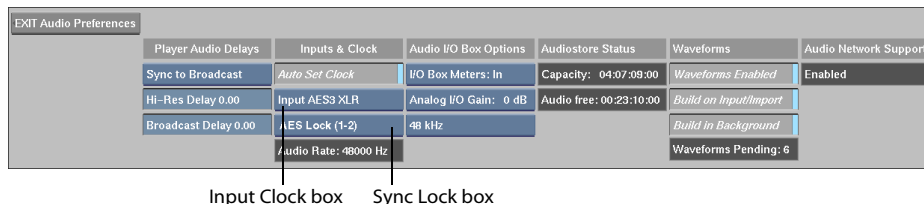
the clocking has been converted to Word Sync, route it to Discreet Audio through the digital input/output adapter and analog/digital converters. If your Discreet Audio system is wired for Word Sync, refer to “Configuring Discreet Audio for Word Sync” on page 476.

NOTE: You can only use analog audio when you connect **flame** to Word Sync. To use digital audio, you must connect Discreet Audio to House Sync.

Configuring Discreet Audio for House Sync

If your Discreet Audio system is wired for House Sync through an AES audio source, use the following procedure to ensure **flame** is properly syncing to the correct signal. If **flame** is not properly syncing to the correct signal, you may encounter problems when you input and output clips.

NOTE: If you have not wired your Discreet Audio system for House Sync, or if you are unsure about the proper sync signal, refer to the *Discreet Audio Hardware Configuration Guide*.



To configure Discreet Audio for House Sync:

NOTE: Before configuring **flame** for House Sync, make sure the yellow sample clock knobs on the front panel of the analog/digital converters are set to Digital Pass. For more information, refer to the *Discreet Audio Hardware Configuration Guide*.

1. Click Audio Preferences.
2. Ensure the Auto Set Clock button is enabled.
3. Select Input AES3 XLR in the Input Clock box.

The current sync rate appears below the Sampling Rate status box. If the rate shown is not 48000 Hz or 44,100 Hz, then your Discreet Audio system is not properly receiving House Sync.

Configuring Discreet Audio for Word Sync

If your Discreet Audio system is wired for Word Sync through a Video-Ref-to-Word Sync converter, use the following procedure to ensure **flame** is properly syncing to the correct signal. If **flame** is not properly syncing to the correct signal, you may encounter problems when you input and output clips.

NOTE: If you have not wired your Discreet Audio system for Word Sync, or if you are unsure about the proper sync signal, refer to the *Discreet Audio Hardware Configuration Guide*.

To configure Discreet Audio for Word Sync:

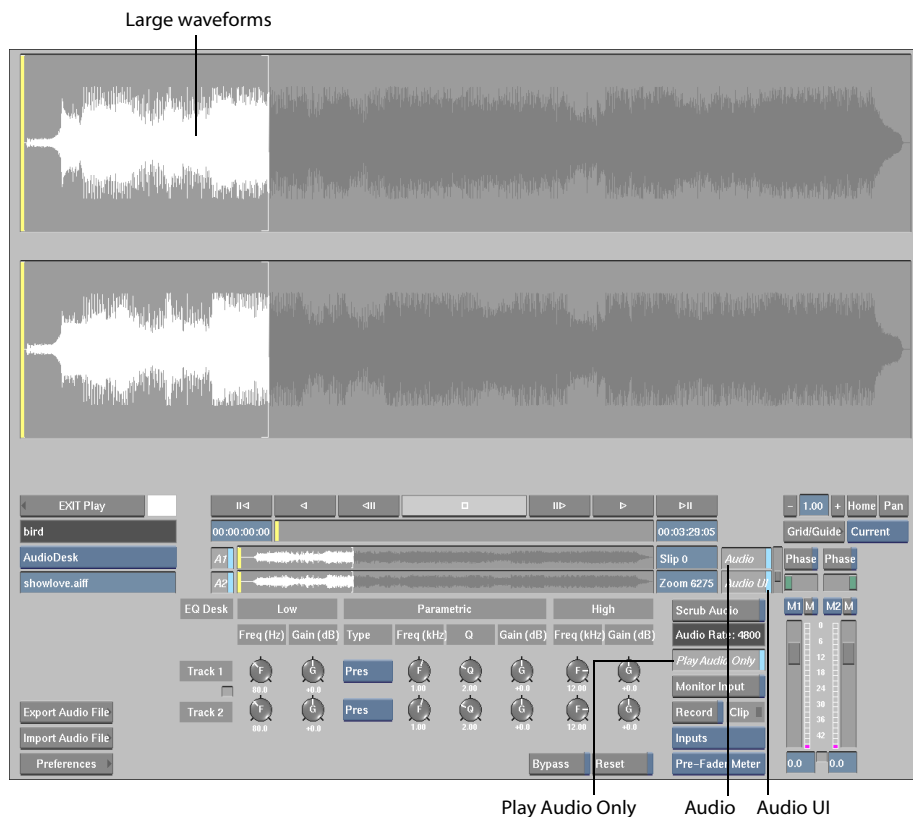
NOTE: Before configuring **flame** for Word Sync, make sure the yellow sample clock knobs on the front panel of the analog/digital converters are set to WS (Word Sync). For more information, refer to the *Discreet Audio Hardware Configuration Guide*.

1. Click the Audio Preferences button in the Preferences menu.
2. Select Word Sync in the Input Clock box.
3. Select Input TOS Optical in the Sync Lock box.

The current sync rate appears below the Sampling Rate status box. If rate shown is not 48000 Hz, or 44,100 Hz then your Discreet Audio system is not properly receiving House Sync.

Working with Waveforms

Use the tools in AudioDesk to view waveform and slip audio. For example, you can view the video in and out points on the waveform and slip the audio for more precise timing with the audio.



Play the audio track using the Player controls beneath the Audio Track window.

AudioDesk Controls

The following controls are available in the AudioDesk to work with your audio files.

Audio

Enable to hear audio during playback.

Audio UI

Enable to view the waveforms beneath the image window controls.

Play Audio Only

Enable to play back only the audio. This also displays the large audio waveforms in place of the clip in the image window.

The Audio Waveform

The audio waveform appears below the image window controls when the Audio UI button is enabled. The left channel appears on the top, and the right channel appears on the bottom. You can view large waveforms by enabling the Play Audio Only button.

The first frame in the clip has a white open bracket and the last frame has a white closed bracket. These brackets are used to indicate how the audio track is synced in relation to the beginning and end of the clip.

When you zoom in on the large waveforms, the small waveforms beneath the Player controls do not zoom so that you can always see your position in the entire audio track.

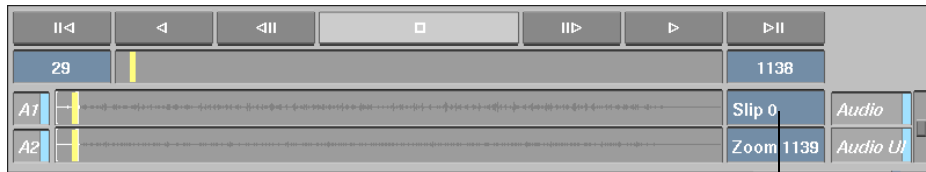
Zoom Box

Use the value in the Zoom box to change the zoom factor of the waveform. Increase the value to increase the zoom factor of the waveform. Decrease the value to decrease the zoom factor of the waveform.

Slipping Audio

The Slip box specifies the offset between the beginning of the audio track and the clip to which it is linked. For example, an offset of -6 frames begins playing the audio track six frames after the beginning of the clip. An offset of 6 frames begins the audio track six frames before the beginning of the clip. This means that the audio track is slip synced six frames.

Slipping the audio track is useful if, for example, your clip fades in and you want the audio track to begin a number of frames into the clip.



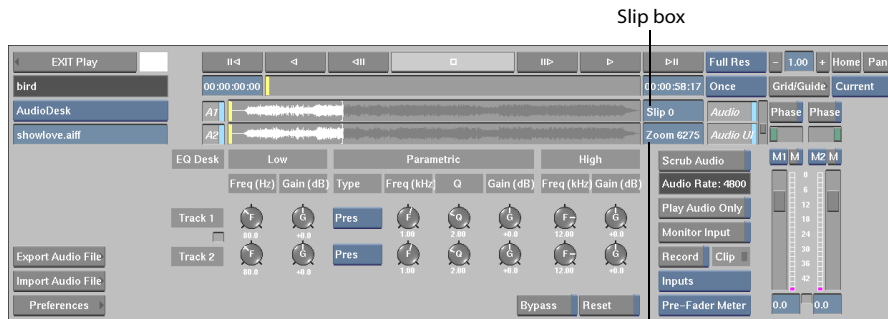
Change the value in the Slip box to offset the audio tracks.

Playing Clips with Audio

You can play the audio track linked to a clip using the AudioDesk in the Player. When Audio is enabled, the audio track and the clip play simultaneously. To hear only the audio, enable Play Audio Only. To view the video without the audio playback, disable the Audio button.

To play the audio track linked to a clip:

1. Centre the clip with the audio track on its desktop reel.
2. Click the Play button.
The Player appears.
3. Select AudioDesk.
4. Enable the Audio button to enable audio playback.
5. Click the Audio UI button to display the audio options.



Zoom box

6. To scale the waveform display, change the value in the Zoom box.
7. To slip the audio, change the value in the Slip box.
8. Click the ► button to play the audio with the clip.

Scrubbing Audio

You can scrub the audio to accurately locate features of the waveform.

To scrub audio:

1. Enable the Scrub Audio button.
2. Grab the positioner in the play bar.
3. Drag the positioner left or right to scrub the audio.

The audio scrubs at the speed you drag. You cannot scrub faster than the regular playback speed.

NOTE: If the Scrub Audio button is disabled, you can use the **CTRL** key and drag the positioner to temporarily scrub the audio.

Scrubbing in Modules

If you load a clip with audio into a module, such as Text or Retouch, you can scrub the audio by holding **CTRL** while you drag the positioner. You can also view the waveform for one track in many modules.

To scrub audio in a module:

1. Load a clip with audio into the module.
2. View the Front.
3. While holding the **CTRL** key, drag the positioner below the image window. The audio scrubs as you drag.

To view a waveform in a module:

1. Go to the Setup menu.
 2. In the Wave box, select Audio 1, or Audio 2.
- The waveform appears below the player controls.

Playing Clips with Audio

You can play the audio track linked to a clip using the AudioDesk in the Player. The Player plays the audio track and the clip simultaneously. To hear the audio without playing back the video, enable Play Audio Only.

To play the audio track linked to a clip:

1. Centre the clip with the audio track on its desktop reel.
 2. Click the Play button.
- The Player appears.
3. Enable the Audio button to enable audio playback.

- 4. Click the Audio UI button to display the audio options.



- 5. To scale the waveform display, change the value in the Zoom box.
- 6. To slip the audio, change the value in the Slip box.

NOTE: Click the ► button to play the audio with the clip.

Audio Timewarps

You can timewarp your audio with the Audio Timestretch option in the Editing menu. Audio Timestrech is supported with any audio subsystem. You can also adjust the pitch of the audio to compensate for its change of speed. For example, when you convert your video from 30fps to 24fps, you will want to make the same speed modification to your audio. However, the pitch of the audio will also be lower, so you need to apply a pitch shift so that the timestretched audio sounds the same as the original source.

If you are moving video material between NTSC, PAL, HD and film partitions, you can apply a slight timewarp to audio and also pitch shift it so that the audio sounds the same in any standard.

To apply an audio timestretch:

- 1. Click Audio Timestretch in the Editing menu.

The Audio Timestretch menu appears.



- 2. Select the clip with audio you want to timestretch.

3. Define the parameters for the timestretch.

Use:	To:
Speed	Change speed of audio clip based on a speed percentage. The value in the Time Ratio field automatically changes as you modify the speed percent.
Time Ratio	Change the speed of the audio clip. This is the ratio of the number of samples in the result divided by the original number of samples. The value in the Speed field automatically changes as you modify the Time Ratio.
Pitch Shift	Change the pitch of the audio clip. Use this option to counteract the pitch change that occurs when speeding up or slowing down the playback of the clip. This is the ratio of the shifted frequency values divided by the original frequency values. (A Pitch shift value of 2 shifts one octave up, a Pitch shift value of 0.5 shifts one octave down.)
Quality Factor	Change the quality of the resulting audio. Use a low quality factor to experiment with different settings, and then use a high quality factor to process the final result. Use one of the following modes: 0: Quick mode 1: Standard mode 2: High Quality
Rhythm	This parameter adjusts rhythm accuracy versus sound quality (from -5 to +5). A value of +5 brings high rhythm accuracy - rhythmical events occur at the exact position. A value of -5 brings high sound quality. The sound quality is very high, while rhythmic event positions may vary +/- 1000 samples from their exact position.

4. Select the destination.

The audio timestretch is processed.

Converting Audio between Different Standards

When you convert your material between different standards, your audio may become out of sync due to the change in the video playback rate. Use the Preset option to convert the audio based on the difference in frame rates.



Preset option

If you are not using the Preset option, you can also use the following table to enter the correct Time Ratio based on the standards between which you are transferring material.

When making this conversion:	Use this Time Ratio value:
PAL to NTSC	.83
PAL to 24p	1.04
NTSC to PAL	1.2
NTSC to 24p	1.25
24p to NTSC	.8
24p to PAL	.96

Audio Sparks

In addition to the audio tools in **flame**, there is an audio Spark included with **flame** that you can use to apply additional effects to your audio. There are four plug-ins that you can use with the audio Spark.

Use one of the following VST™ (Virtual Studio Technology) Sparks to make advanced adjustments to your audio:

- Compression
- EditEQ
- Modulation
- Reverb

The Spark VST can be animated on a clip basis.

To use an audio spark:

1. Click L on a Spark button to go to the Spark browser.
2. Select VSTAudioEffect from the Sparks menu.

The VST Spark is loaded and you are returned to the desktop.

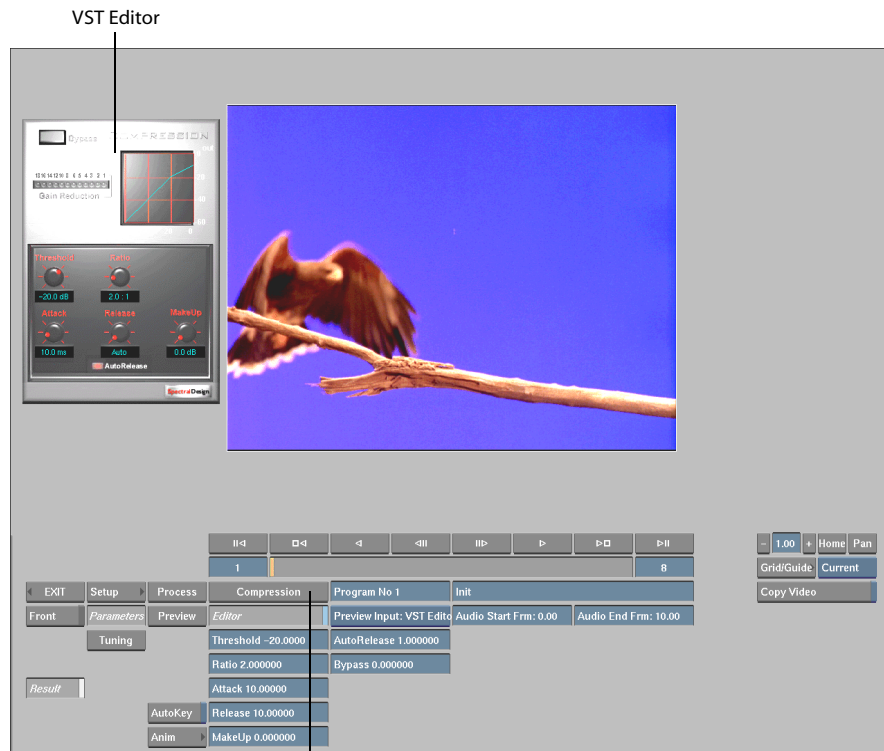
3. Click VSTAudioEffect.

You need to load the VSTAudioEffect into a Spark before you can use it.

EDITING	▶ Paint	Keyer	VSTAudioEffect	Delete
PROCESSING	▶ Warper	Compositor	Sparks	L Name
EFFECTS	▶ Text	Quick Composite	Sparks	L Move
FORMAT	▶ Stabilizer	Optics	Sparks	L Copy
SYSTEM	▶ Action	Modular Keyer	Sparks	L Search

4. Select the clip with audio you want to edit.

The Spark VST menu appears.



Click the Effect button to load a different VST Effect.

5. To load a plug-in, click the Effect button and select one of the following from the file browser.

Select:	To:
Compression	Use compression to modify the overall dynamic range by adjusting parameters such as threshold, attack and release.
EditEQ	Adjust the EQ with a 4-band EQ interface.
Modulation	Apply modulation effects such as delay, chorus, flanger, and phaser.
Reverb	Apply reverb effects.

6. Modify the values directly in the VST Editor or by changing the values in the numeric boxes to get the result you want. You can make the changes while playback is stopped, or you can see your changes dynamically update while the clip plays back. See “Changing VST Values during Playback” on page 485.

NOTE: Separate documentation has been provided for the audio sparks. Consult that documentation for a description of how to use each effect.

7. Click Preview to hear the result.
8. Click Process to process the final effect. If you want to process only the audio, disable Copy Video.

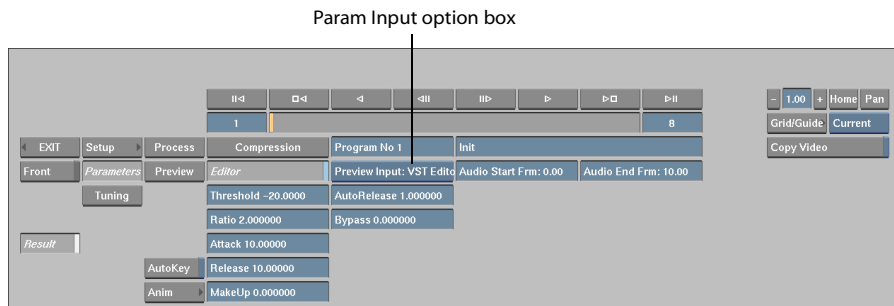
Changing VST Values during Playback

You can choose one of two parameter modes while the audio is playing back after you click either Preview or Process. In VST Editor mode, you can dynamically update any of the values in real time. If you have AutoKey enabled in this preview mode, you create an animation of the VST Spark values in real time as the audio is playing back. In Animation mode, the controls and values automatically update if you created an animation, however, you cannot modify any of the values during audio playback.

You can move the position of the VST Editor by clicking **ALT-F7** and dragging the editor to a new location.

To change VST values during playback:

1. Select Param Mode: VST Editor in the Param Input option box.



2. Click Preview or Process.
The audio begins to play back.
3. Change the values by dragging in the numerical boxes or in the VST Editor.

Creating VST Programs

You can create many different VST programs to have many different versions of the effect. You can quickly select any program to determine which one provides the best result before you process the final clip. You can save up to 10 programs with each setup.

To create a new program:

1. Create the VST Effect as described in “Changing VST Values during Playback” on page 485.
2. Enter any value between 1 and 10 in the Program Number box.

Tuning Options

There are two tuning options for working with audio in the VST Spark.

Use:	To:
Preview Buffer	Specify the number of samples simultaneously loaded into the preview buffer. It is recommended to keep the default value; however, you may need to modify the value if you hear clicks and pops.
Input Gain	Change the gain of the audio signal that is sent to the VST Spark.



Section 4: Transforming Clips

Create complex

effects with

workflow solutions

found on the

Processing and

Format menus.

The Processing Menu

Into the mill

The Processing menu contains several functions to add effects to your clips such as the ability to apply filters, generate mattes, and use the Colour Corrector. Use the Processing menu to access the Batch module.

Summary

In this chapter, you learn about:

- “Using the Average Command” on page 491
- “Creating Compound Images” on page 493
- “Flipping Images” on page 494
- “Using Logical Operations” on page 495
- “Using the Difference Command” on page 498
- “Using the Auto Matte Command” on page 500
- “Creating a Monochrome Clip” on page 501
- “Creating a Negative Clip” on page 502
- “Modifying Luma/Chroma of a Clip” on page 502
- “Creating Coloured Frames” on page 503
- “Degraining and Regraining Clips” on page 504
- “Matching Grain” on page 508
- “Applying Grain with a Uniform Spatial Distribution” on page 513

About Processing

Click the Processing button in the Main menu to display the Processing menu. A brief description of each button in the Processing menu is provided.

LIBRARY				
EDITING	Filter	Difference	Monochrome	Delete
PROCESSING	Average	Auto Matte	Negative	Name
EFFECTS	Compound	Batch	Colour Correct	Move
FORMAT	Flip	DeGrain	Posterise	Copy
SYSTEM	Logic Ops	ReGrain	Coloured Frame	Search

Filter — Applies different effects to a clip such as textures, blurring, edge detection, embossing, sharpening, or a combination of effects. You can use the filters supplied with **flame** or you can create a custom filter. See Chapter 51, “Paint: Using Filters and Special Effects Media.”

Average — Uses an image averaging function to add motion blur to a clip. See “Using the Average Command” on page 491.

Compound — Removes noise from a static image by compressing several frames into a single frame. See “Creating Compound Images” on page 493.

Flip — Flips the frames in a clip horizontally, vertically, or both horizontally and vertically. See “Flipping Images” on page 494.

Logic Ops — Accesses the Add, Subtract, Multiply, Max/Lighten, Min/Darken, Screen, Overlay, Hard Light, Soft Light, Difference, and Exclusion modes, as well as various other Boolean operations. Use these modes to combine two source clips by applying the selected operation to their colour components. See “Using Logical Operations” on page 495.

Difference — Generates a matte clip from two clips that contain the same background but different foreground elements. See “Using the Difference Command” on page 498.

Auto Matte — Generates a high-contrast matte from a clip. See “Using the Auto Matte Command” on page 500.

Batch — The Batch module lets you build a process tree of connected tasks. The result of one task is used as the input of the next task. You can preview the result at any point in the process tree, and modify or delete any task without affecting the other tasks in the tree. See Chapter 29, “Batch Processing.”

DeGrain — Is a general noise reducer that removes noise, softens the image and selectively restores the sharp details of an image. See “The DeGrain Command” on page 504.

ReGrain — Synthesizes noise patterns and applies them to clips. It modulates the intensity of the noise according to the luma or density of the image pixels. See “The ReGrain Command” on page 506.

Monochrome — Generates a monochrome copy of a clip. See “Creating a Monochrome Clip” on page 501.

Negative — Generates a negative copy of a clip. See “Creating a Negative Clip” on page 502.

Colour Correct — Adjusts the colours in a clip. Options include:

- Adjusting the gamma, luma (brightness), contrast, hue, and saturation
- Colour balancing

- Rewiring the RGB channels
- Creating colour curves to accurately remap the colour values for different colour models of an image.

For more information, see Chapter 26, “Colour Corrector.”

Posterise — Produces a posterised copy of a clip by reducing the number of luma and/or chroma levels in the clip. See “Modifying Luma/Chroma of a Clip” on page 502.

Coloured Frame — Generates clips of identical frames of a solid colour, noise, or colour bars. See “Creating Coloured Frames” on page 503.

Processing Commands

This section describes basic image-processing commands found in the Processing menu. These commands are used to create visual effects, generate mattes, and enhance image quality.

Using the Average Command

You can use the Average command to simulate the motion blur of a moving object in a clip.

flame creates the motion blur by applying an image averaging function to each frame in the source clip. For each frame, **flame** superimposes partially transparent copies of the images in the surrounding frames on top of the image in the current frame. Since the moving object is in a different position in each successive frame, the superimposed images in one frame will also show the object in different positions, creating the blur effect.

The number of images superimposed in one frame determines the length of the blur. You can control the length of the blur effect as well as the transparency of the superimposed images with the controls in the Average menu.

The Average Over Value — The Average Over value controls the number of frames used to calculate the average (the number of images superimposed on one frame). You enter this value in the Average Over field. The Average Over value affects both the length and the transparency of the blur. For example, averaging over a large number of frames increases the length of the blur and the transparency of the superimposed images.

Weighted and Uniform Averages — You can also control the transparency of the blur with a weighted or a uniform average. For each frame processed in a weighted average, the frame closest to the current frame carries the most weight in the average calculation and is the most opaque. The frame furthest from the current frame carries the least weight and is the most transparent. In a uniform average, all frames carry equal weight and are equally transparent.

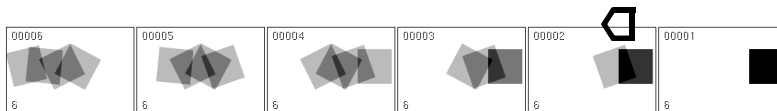
Past and Future Frames — You can use Past frames or both Past and Future frames to calculate the average:

- With Past Frames, **flame** considers only the frames that precede the current frame. The motion blur follows the object.
- With Past and Future frames, **flame** considers the frames on both sides of the current frame.

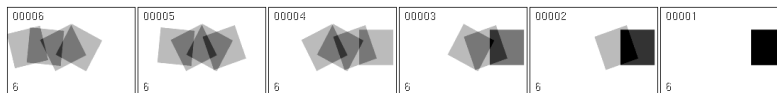
Examples of Averaging Source Clips

The following examples demonstrate using the Average command to process a source clip that is six frames long with the Average Over value set to 4. The first result clip shows a uniform average using past frames; the second result clip shows a weighted average also using past frames. The frames in the result clips are generated as follows:

- Frame 1 is the same as source frame 1. Since there are no clips preceding the first frame, the first frame in the generated clip is always the same as the first frame in the source clip.



Action: Select the source clip



Result: Uniform average over 4 frames



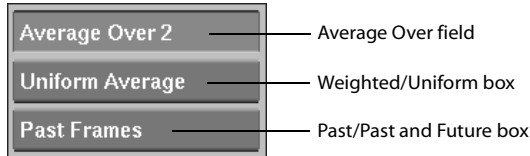
Result: Weighted average over 4 frames

- Frame 2 is the result of averaging source frames 1 and 2.
- Frame 3 is the result of averaging source frames 1, 2, and 3.
- Frame 4 is the result of averaging source frames 1, 2, 3, and 4.
- Frame 5 is the result of averaging source frames 2, 3, 4, and 5.
- Frame 6 is the result of averaging source frames 3, 4, 5, and 6.

To use the Average command:

1. Click the Average button in the Processing menu.

The Average menu appears.



2. Enter the number of frames for the average in the Average Over field.
3. Select either Weighted or Uniform Average and either Past or Past and Future frames to use in the calculation.
4. Select the source clip.
5. Select the destination reel.

The result clip appears on the destination reel.

Creating Compound Images

The Compound command combines several frames in the source clip into a single frame. For example, this is useful in removing noise from a clip of a static image. You can use the Compound command to combine all frames in the clip into a single frame for a clean image.

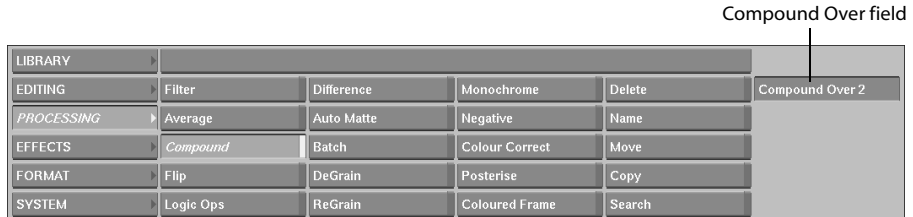
The Compound Over Field — The Compound Over value determines the number of frames to combine into one frame. This value also determines the percentage of each source frame used to produce the new frame.

For example, if the Compound Over value is set to 2, then 50% of each of the first two frames of the source clip is combined to generate the first frame of the destination clip. The third and fourth frames of the source clip are combined to generate the second frame of the destination clip, and so on. Compounding a clip over two frames generates a destination clip half the length of the source clip. To compress 20 frames into one, set the Compound Over value to 20.

To use the Compound command:

1. Click the Compound button in the Processing menu.

The Compound Over field appears.



2. Set the value in the Compound Over field.

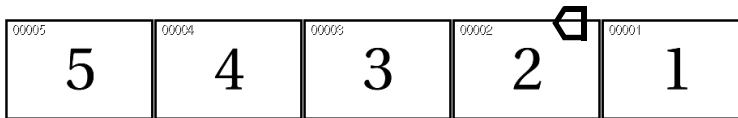
3. Select the source clip.

4. Select the destination reel.

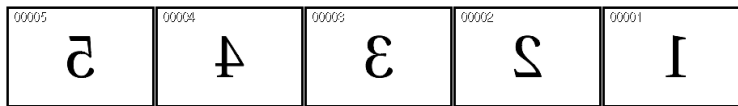
The result clip appears on the destination reel.

Flipping Images

You can generate a mirror image of a clip with the Flip command. The clip can be flipped horizontally, vertically, or both, as shown:



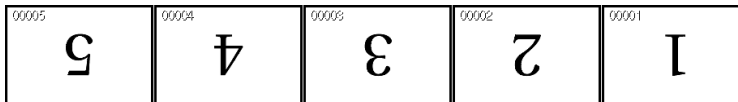
Action: Select the source clip



Result: Flipping the clip horizontally



Result: Flipping the clip vertically

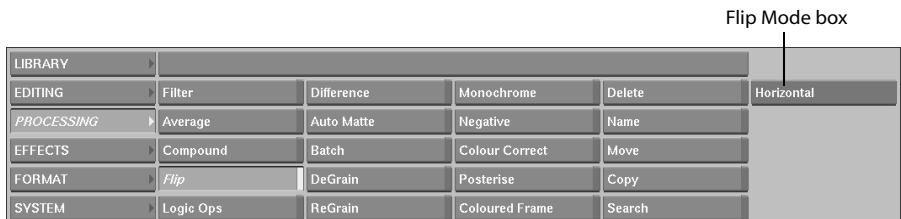


Result: Flipping the clip horizontally and vertically

To flip clips:

- 1. Click the Flip button in the Processing menu.

The Flip Mode box appears.



- 2. Select an option from the Flip Mode box.



Select:	To:
Horizontal	Flip each frame in the clip about the horizontal axis.
Vertical	Flip each frame in the clip about the vertical axis.
Horizontal & Vertical	Flip each frame in the clip about both axes.

- 3. Select the source clip.
- 4. Select the destination reel.

The result clip appears on the destination reel.

Using Logical Operations

Logical operations are applied separately to each of the Red, Green, and Blue components of images. You can apply Logic Ops commands to combine the RGB channels of corresponding pixels from two source images as described in the following table.

Select:	To:
Add	Add the luma values of the corresponding pixels from two source clips and assign the resulting value to the corresponding pixel in the generated clip. If the result is greater than 255 (in 8-bit mode), the pixel in the destination clip is clamped at a value of 255 (white). In 12-bit mode, the maximum colour value is 4095. The order in which the two source clips are selected does not affect the result of the Add operation. The generated clip is brighter than either of the two selected source clips.

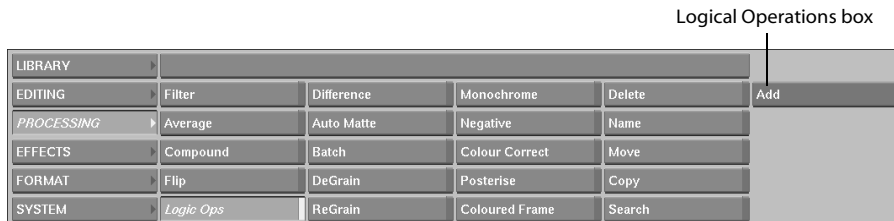
Select:	To:
And	Apply a bitwise logical AND operation between the RGB channels of corresponding pixels in the two source clips. The order in which the two source clips are selected does not affect the result of the AND operation.
AndInv	Apply a bitwise logical AND operation between the negative of the RGB channels of the pixels of the front clip and the RGB channels of the pixels of the back clip.
AndRev	Apply a bitwise logical AND operation between the negative of the RGB channels of the pixels of the back clip and the RGB channels of the pixels of the front clip. AndRev and AndInv yield the same result if you switch the order of the two source clips.
Difference	Calculate the difference between the RGB channels of the corresponding pixels of the two source clips. As opposed to Subtract, the order of the two source clips does not matter, as the smaller channel value is always subtracted from the larger channel value. Blending with white inverts the colour channel values; blending with black produces no change.
Exclusion	Add the RGB channel values of the corresponding pixels from the two source clips, then subtract twice the product of these channels. The order of the source clips does not matter.
HardLight	Multiply or screen the selected colour of the front clip onto the back clip, depending on the blend colour. The effect is similar to shining a harsh spotlight on the image, and greatly reduces the contrast levels in the image. If the blend colour (light source) is lighter than 50% grey, the image is lightened as if it were screened—it is useful for adding highlights to an image. If the blend colour is darker than 50% grey, the image is darkened, as if it were multiplied—it is useful for adding shadows.
Max/Lighten	Compare the RGB channels of each pixel of the source clips individually and return the larger of the two values. The order in which you select the source clips does not affect the result.
Min/Darken	Compare the RGB channels of each pixel of the source clips individually and return the smaller of the two values in the resulting clip. The order in which you select the source clips does not affect the result.
Multiply	Multiply the RGB channel values of corresponding pixels of the two source clips and normalize the result by dividing by 255 in 8-bit mode, or 4095 in 12-bit mode. The resulting RGB channel values are assigned to the corresponding pixels in the generated clip. The order in which you select the source clips does not affect the result.
Nand	Apply a bitwise logical AND operation between the RGB channel values of the corresponding pixels of the two source clips, negate the result, and assign the result to the channels of the corresponding pixels in the generated clip.

Select:	To:
Nor	<p>Apply a bitwise logical OR operation between the RGB channel values of the corresponding pixels of the two source clips, negate the result, and assign the result to the channels of the corresponding pixels in the generated clip.</p> <p>The order in which you select the source clips does not affect the result.</p>
Or	<p>Apply a bitwise logical OR operation between the RGB channel values of the corresponding pixels of the two source clips, and assign the result to the channels of the corresponding pixel in the generated clip.</p> <p>The order in which you select the source clips does not affect the result.</p>
OrInv	<p>Apply a bitwise logical OR operation between the negative of the RGB channel values of the pixels in the front clip and the RGB channel values of the corresponding pixels in the back clip, and assign the result to the channels of the corresponding pixels in the generated clip.</p>
OrRev	<p>Apply a bitwise logical OR operation between the RGB channel values of the pixels in the front clip and the negative of the RGB channel values of the corresponding pixel in the back clip, and assign the result to the channels of the corresponding pixels in the generated clip.</p> <p>OrRev and OrInv yield the same result if you switch the order of the two source clips.</p>
Overlay	<p>Multiply or screen the colours, depending on the RGB channel values of the first clip you select. Patterns or colours overlay the existing RGB channel values while preserving the highlights and shadows of the first clip's colour. The first clip's colour is not replaced but is mixed with the second clip's colour to reflect the lightness or darkness of the original colour.</p>
Screen	<p>Multiply the inverse of the second clip's colours with the colours of the first clip. The resulting colour is always a lighter colour. The colour remains unchanged when you screen with black. Screening with white produces white. The effect is similar to projecting multiple photographic slides on top of each other.</p>
SoftLight	<p>Shine a soft, diffuse light on the image. If the blend colour (light source) is lighter than 50% grey, the image is lightened. If the blend colour is darker than 50% grey, the image is darkened.</p> <p>Using this mode with a black frame results in a very dark effect, with white a very bright one.</p>
Subtract	<p>Subtract the RGB channel values of the pixels from the back clip from the RGB channel values of the pixels from the front clip and assign the result to the RGB channel values of the pixel in the generated clip. If an RGB channel value from the back clip is larger than the corresponding channel value in the front clip, yielding a negative result, that result is clamped at 0 (black).</p> <p>The order of the source clips is important since the back clip is subtracted from the front clip. The resulting clip is always darker than either of two source clips.</p>

Select:	To:
Xnor	<p>Apply a bitwise logical exclusive OR operation between the RGB channel values of the corresponding pixels of the two source clips, negate the result, and assign the result to the channels of the corresponding pixels in the generated clip.</p> <p>The order in which you select the source clips does not affect the result.</p>
Xor	<p>Apply a bitwise logical exclusive OR operation between the RGB channel values of the corresponding pixels of the two source clips, and assign the result to the channels of the corresponding pixels in the generated clip.</p> <p>The order in which you select the source clips does not affect the result.</p>

To use the Logic Ops commands:

1. In the Processing menu, click Logic Ops.
The Logical Operations box appears.



2. From the Logical Operations box, select the mode you want to use.
3. Select the two source clips—a front and a back—to be used for the operation.
The order in which clips are selected affects the outcome of some of the operations. See the description of each mode in the preceding table.
4. Select the destination reel.
The generated clip appears on the destination reel.

Using the Difference Command

The Difference command generates a matte clip from two source clips with the same background but different foreground elements. This allows you to remove an image from one context to add it to another. You can even generate a composite clip while creating the difference matte. When you select the Front/Back/Comp input mode and load a composite clip into the Difference menu, a composite clip is created using the front clip, composite clip, and difference matte.

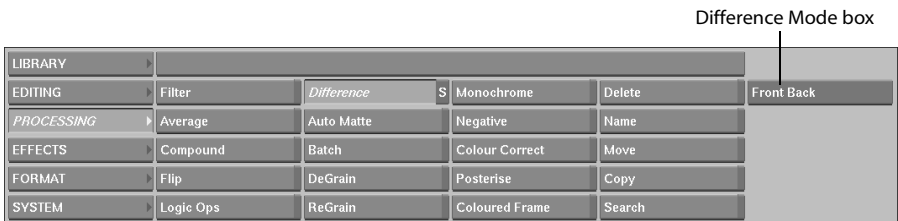
The Difference command calculates the difference between corresponding pixels of the two source images. The value of the pixel in the back image is subtracted from the value of the corresponding pixel in the front image, and the resulting absolute value is used in the difference matte.

The matte is created using the Tolerance and Softness values that you specify. The Tolerance value specifies the difference level that is considered black. A higher Tolerance value includes more black in the matte. The Softness value is used to soften the transition between the light areas and the dark areas of the matte by adjusting the amount of grey at its edges. Grey information is not included in the matte when the Softness is zero. Use a higher Softness value to increase the grey.

To open the Difference menu:

- 1. Click the Difference button in the Processing menu.

The Difference Mode box appears.

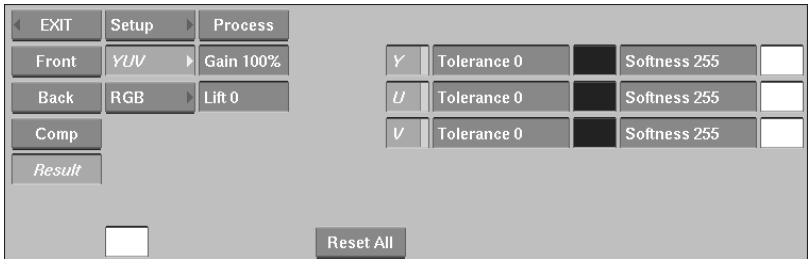


- 2. Select an option from the Difference Mode box.

- | | |
|-----------------|---|
| Select: | To: |
| Front/Back | Load a front clip and a back clip. |
| Front/Back/Comp | Load a front clip, a back clip, or a composite clip (new background). |

- 3. Select the source clips. They must be selected in order: front, back, composite.
- 4. Select the destination reel.

The Difference menu appears.



To create the difference matte:

1. Click Result to display the resulting matte clip.
2. Select the colour model to use for the difference matte: RGB or YUV.
3. Choose the colour channel(s) to use for the difference matte. You can use any of the Y, U, or V channels in the YUV colour model, or any of the R, G, or B channels in the RGB colour model.
4. Set the tolerance for the matte by pressing on the colour box beside the Tolerance field in the same row as the colour channel you are using. Drag the colour picker around the area of the image to matte out.

NOTE: You can also set the tolerance for the matte directly in the Tolerance field.

5. Set the softness for the matte using either the Softness field or the colour box beside the field.
6. Use the controls in the menu to manipulate the resulting matte.

Use:	To:
Gain	Boost the matte. Difference multiplies the values of the pixels in the result matte by the Gain.
Lift	Add the Lift value to all pixels in the difference matte.

7. Click Process.

Using the Auto Matte Command

The Auto Matte command generates a high-contrast matte from a clip. The parameters for the command are set in the Auto Matte menu:

- The Min field specifies the minimum colour value for the matte. Any pixel having a luma value below the minimum value is set to black. The minimum can be set to any value between 0 and 255 in 8-bit mode, or between 0 and 4095 in 12-bit mode.
- The Max field specifies the maximum colour value for the matte. Any pixel having a luma value above the maximum is set to white. The maximum can be set to any value between 0 and 255 in 8-bit mode, or between 0 and 4095 in 12-bit mode.
- The Gain value is used to boost the matte. Any pixel with a luma value between the minimum and maximum values is multiplied by the Gain and the new value is assigned to the resulting pixel. Using a low value for the Gain produces a soft matte with more grey levels. Using a high value produces a matte with fewer grey levels. Gain is expressed as a percentage value. The default value of 100% has no effect on the image since luma values are multiplied by 1.

To use the Auto Matte command:

1. Click the Auto Matte button in the Processing menu.

The Auto Matte menu appears.

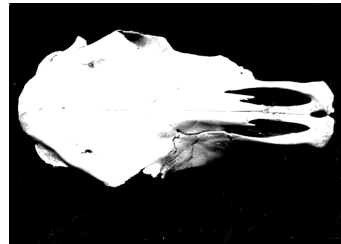


2. Set the values in the Min, Max, and Gain fields.
3. Select the source clip.
4. Select the destination reel.

A matte clip appears on the destination reel.



Original image



Matte generated with Min = 82, Max = 157

Creating a Monochrome Clip

The Monochrome command quickly generates a monochrome clip.

To use the Monochrome command:

1. Click the Monochrome button in the Processing menu.
2. Select the source clip.
3. Select the destination reel.

The monochrome clip appears on the destination reel.

Creating a Negative Clip

The Negative command quickly generates the negative of a source clip. The colour values of each pixel in the source image are inverted to produce the negative image.

To use the Negative command:

1. Click the Negative button in the Processing menu.
2. Select the source clip.
3. Select the destination reel.

A negative copy of the clip appears on the destination reel.

Modifying Luma/Chroma of a Clip

You can specify the number of luma and/or chroma levels in a source clip with the Posterise command. This command divides each frame in the clip into an equal number of luma and/or chroma levels, and maps each pixel of the image to the closest matching level.

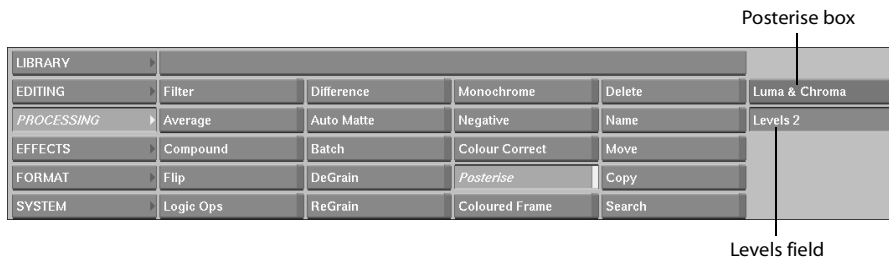
To reduce the number of luma levels in a clip, select Luma in the Posterise box. To reduce the number of chroma levels, select Chroma. Chroma levels work on each of the red, green, and blue channels. For example, a chroma level of 2 produces two levels of red, two of green, and two of blue.

To reduce both the luma levels and the chroma levels, select Luma & Chroma in the Posterise box.

To use the Posterise command:

1. Click the Posterise button in the Processing menu.

The Posterise menu appears.



2. Select the channel(s) to use in the Posterise box.
3. Specify the number of levels you require in the Levels field.
4. Select the source clip.
5. Select the destination reel.

The posterised clip appears on the destination reel.



Original image



Posterised image, Levels = 5

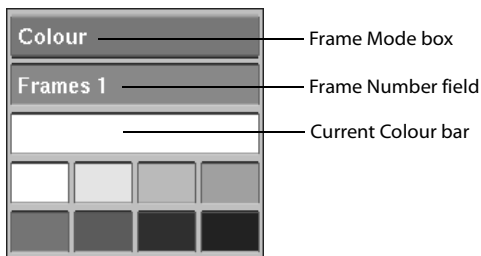
Creating Coloured Frames

The Coloured Frame command generates a clip with one or more identical frames of a solid colour, noise, or SMPTE or PAL colour bars at 75% or 100% luma.

To create coloured frames:

1. Click the Coloured Frame button in the Processing menu.

The Coloured Frame menu appears.



2. Select the type of frame to generate from the Frame Mode box.

HINT: Best results are obtained when a value from 1 to 20 is used.

Select:	To:
Colour	Generate frames of a solid colour.
Noise	Generate frames of video static noise.
Colour Noise	Generate frames of colour video static noise.
SMPTE Bars 75%	Generate frames of SMPTE standard colour bars at 75% luma.
SMPTE Bars 100%	Generate frames of SMPTE standard colour bars at 100% luma.
PAL Bars 75%	Generate frames of PAL standard colour bars at 75% luma.
PAL Bars 100%	Generate frames of PAL standard colour bars at 100% luma.

3. Specify the number of frames that you want to generate in the Frame number field.

4. If the Frame Mode is set to Colour, then set the colour to be used for the frames in the Current Colour bar. Perform one of the following:
 - Several colour pots appear beneath the Current Colour bar. Click one of these colour pots to transfer that colour into the Current Colour bar.
 - Click on the Current Colour bar to call up the colour picker. Use the colour picker to create the current colour as described in “The Colour Picker” on page 57.
5. Select the destination reel.
The generated clip appears on the destination reel.

Degraining and Regraining Clips

It is possible to add or remove grain from the RGB channels of a selected colour in any image using the DeGrain and ReGrain commands. Grain in an image can be either beneficial or detrimental to your clips. In the background of an image, graininess can make it difficult to “pull” a clean and effective key. As an added effect, graininess can be effective in synthesizing a film quality in a video image.

The following section introduces both the DeGrain and ReGrain commands in the Processing menu.

The DeGrain Command

The DeGrain command is a general noise reducer that consists of two filters:

- An RGB-averaging filter that removes noise and softens the image.
- An RGB-sharpening filter that selectively restores the sharp details of an image.

These filters can be adjusted to achieve the desired compromise between a blurred de-grained image and a sharp grainy image.

With the DeGrain command you can:

- Remove grain from a film-originating image to transfer or match it to video quality.
- Remove noise on video clips.
- Reduce the size of grain on images printed on film that was push-processed to compensate for overexposure or underexposure.
- De-grain images before creating a matte in order to obtain a more precise result.
- Make an image less grainy. In some cases, it may be best to remove the grain completely, and then manually restore the desired amount of grain using the ReGrain command. In other cases, you get material with completely different grain signatures that need to be composited together. It is a good idea to remove the grain from one, and give it the grain of the other.

NOTE: The amount of grain existing in each channel of a colour is not generally uniform. Usually the blue channel contains more grain than the red or green channels. Therefore, you will obtain

the best results using the DeGrain command if you de-grain each channel of a colour separately. Also, because grain depends on film stock, lighting conditions, colour, resolution, etc., it is advisable to create a new DeGrain setup for each shot you need to process.

To reduce grain in a clip:

1. Click the DeGrain button in the Processing Menu.
2. Select a clip to de-grain and a destination reel.

The DeGrain menu appears.



NOTE: Since the clip appears in Result view by default, a delay may occur while the system applies the current DeGrain filter (if any).

3. Click Setup, Reset All, and then Confirm.
4. If you are working with an interlaced clip, select Field from the Frame/Field option box.

NOTE: Since de-graining applies filtering to an image, it is best to apply filtering to fields when the image material is from a video source clip.

5. Enable Crop and draw a crop box in an area that contains a great deal of grain.

NOTE: When the Crop button is enabled, DeGrain specifications are restricted to the area within the crop box. When the Crop button is disabled, DeGrain specifications are applied to the entire image.

6. Click the R, G, or B button to de-grain the appropriate channel from the colour in the selected area.
7. Enable the Exclusive button to show the selected channel as greyscale. This is useful when viewing the blue channel, as the eye is less sensitive to blue than to other colours.
8. Disable the P (Proportional) button to the left of the sliders to adjust the channels independently.

You can apply DeGrain specifications to all channels simultaneously by enabling the P button, but adjusting each one individually is far more accurate.

9. Drag the appropriate RGB Average slider upward until the desired amount of grain is reduced in the selected channel.
10. Drag the appropriate RGB Sharpen slider upward until you are satisfied with the image's sharpness.

NOTE: It is at this point that you will recognize the need for compromise between removing grain and retaining sharpness in an image.
11. Move the crop box to different areas of the image to ensure that areas with different luma values also have the desired amount of grain.
12. Repeat steps 4 to 10 for the other channels in each selected area of the image.
13. Click Process to de-grain the clip.
flame saves the de-grained clip on the destination reel.

The ReGrain Command

The ReGrain Command synthesizes noise patterns and applies them to clips. It modulates the intensity of the noise according to the luma (or density) of the image pixels using a granularity curve that specifies the gain for the luma values of each channel.

There are two modes within the ReGrain command:

- In Type mode, the granularity curves of some Kodak emulsions are built in, and the RGB Gain parameters that you set are translated as scaling factors for the curves.
- In Custom mode, you can use the ReGrain Channel Editor and the RGB Radius sliders to build granularity curves from grain that you wish to match.

Use the Type mode to apply grain using the built-in curves for Kodak emulsions. Use the Custom mode to exactly match the grain of a background clip.

Keep the following points in mind:

- Create a matte before re-graining specific areas of a clip.
- For elaborate compositing, use the Keyer to produce both the composite and the matte, and then re-grain the result using the matte.
- If you want to add colour-corrected grain to a clip, colour-correct it before using it to build a grain profile.
- If the original grain has to be reapplied to a de-grained clip, load the original clip as the Back clip, and the de-grained version as the Front clip. Then match the original clip to the de-grained clip across every necessary area of the image. You can also use this method to build a very accurate setup from a grainy clip: de-grain it, match the grain, and save the setup to be used on other clips.

Applying Grain to a Clip

Select the Type mode from the ReGrain menu to apply grain using built-in curves for Kodak emulsions.

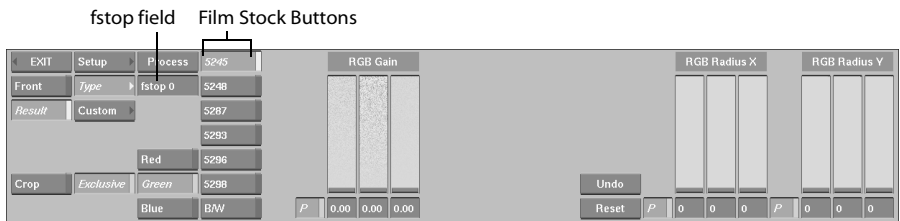
To apply grain in Type mode:

- 1. Click the ReGrain button in the Processing menu.

The Input Mode box appears.

Select:	To:
Front	Apply grain to different areas of one image.
Front Back Matte	Apply grain through the matte.

- 2. Select an input mode, the necessary clips, and a destination reel.
- The ReGrain Menu appears. By default, the Type button is enabled.



NOTE: Since the clip appears in Result view by default, a delay may occur while the system applies the current ReGrain filter (if any).

- 3. Click one of the Film Stock buttons.
- 4. To compensate for under or overexposure, set the relative exposure offset using the fstop field.
The range of permissible f-stop values is from -3 to 3; values entered must be integers.
- 5. If you want to sample grain from an area on the back clip, click the Back button.
The back clip appears.
- 6. Hold down the **CTRL** key and draw a selection box on the back clip. Be careful to select areas of uniform colour with no image detail, and especially, no edges.
The values contained within the selected area of the image are displayed in the ReGrain Channel Editor. The RGB Gain and Radius sliders are also automatically adjusted to display these values.
- 7. Disable the P (Proportional) button to adjust the channels independently.
You can apply DeGrain specifications to all channels simultaneously by enabling the P button, but adjusting each one individually is far more accurate.

8. Drag the appropriate RGB Gain slider to increase or decrease the colour value for the selected channel.
The RGB Gain sliders are translated as scaling factors for the built-in granularity curves.
9. Click the R, G, or B button to adjust the grain size for the appropriate channel.
10. Enable the Exclusive button to show the selected channel as greyscale. This is useful when viewing the blue channel, as the eye is less sensitive to blue than to other colours.
11. Drag the X-Radius and Y-Radius sliders for the selected channel to change the horizontal or vertical size of the grain.
12. Select the matte option:

Select:	To:
Matte On	Re-grain the part of the image corresponding to white part of the matte.
Matte Invert	Re-grain the part of the image corresponding to the black part of the matte.
Matte Off	Re-grain the entire front clip.

13. Click Process to re-grain the clip.
flame saves the re-grained clip on the destination reel.

Matching Grain

Select the Custom mode from the ReGrain menu to match grain. The process pipeline for re-graining a clip in Custom mode is as follows:

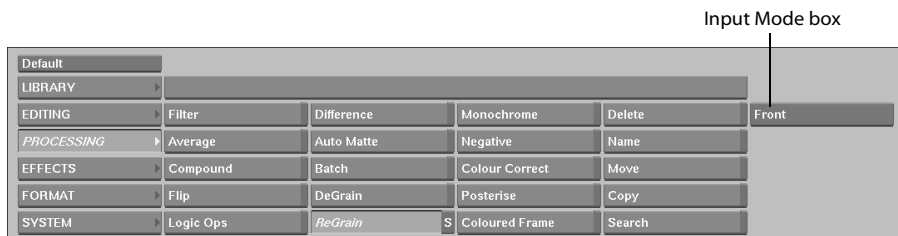
- Begin the ReGrain process.
- Analyze the grain of the clip you want to match.
- Compare the grain that you want to match.
- Adjust the grain size if necessary.
- Adjust the grain colour if necessary.
- Process the clip.

Beginning the ReGrain Function

The ReGrain function lets you work with a single clip or a front, a back, and a matte. If you are matching the grain of a background, select the clip to re-grain as the front; the clip you want to match as the back; and an appropriate matte. If you are matching the grain of two shots, select any clip as the matte and disable the matte in the ReGrain Menu.

To regrain a clip:

1. Click the ReGrain button in the Processing Menu.
The Input Mode box appears.



2. Select an input mode.

Select:

To:

Front

Match the grain of two areas of the same image.

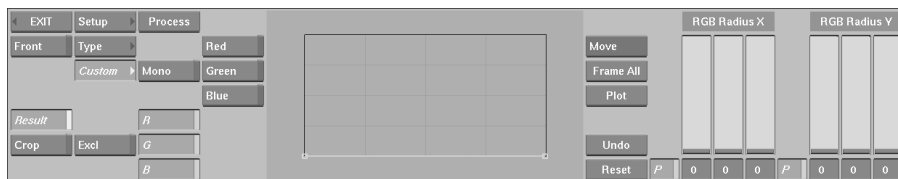
Front Back Matte

Match the grain and composite or match the grain of two shots.

3. Select the clips that you want to match, and click a destination reel.
The ReGrain menu appears. By default, the Type button is enabled.

4. Click the Custom button.

The Custom menu appears.



Analyzing the Grain

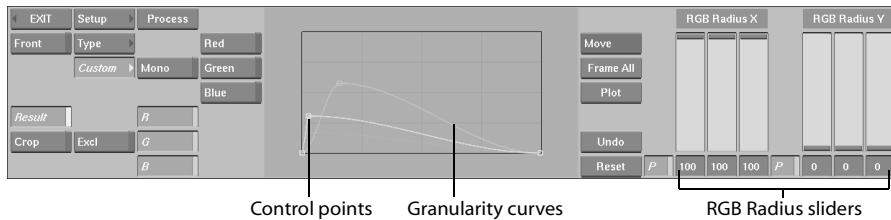
The ReGrain Channel Editor extracts the intensity of the grain in each channel as well as information needed to produce the desired average grain size.

NOTE: When selecting areas for analysis, select areas of uniform colour with no image detail. Otherwise, you will obtain bad control points in the granularity curves and wrong grain strength for some colours.

To analyze grain:

1. If the grain you want to match is in an area on the back clip, click Back. If it is in an area of the front clip, click Front.
The appropriate clip appears.
2. Hold down the **CTRL** key and draw a selection box on one of the major colours in the back clip. Be careful to select areas of uniform colour with no image detail.

The ReGrain Channel Editor sets control points in the appropriate channels that reflect the extracted intensity for the average luma of the selected grain field.



3. Draw a crop box for each major colour in the clip.

flame adds control points to the curve for each colour that you select.

This can be done for all the channels simultaneously, or separately by enabling the appropriate channel button.

NOTE: Repeat these steps as necessary and try to sample areas of the image of all intensities (shadows, midtones and highlights). In most cases, these are the only steps you will have to take to get a perfect match.

Comparing the Grain

There are two relevant procedures for comparing the grain of two images; using a crop box to see a section of the re-grained clip against the background, and using the Result button to see the result of re-graining and compositing. Both procedures are useful when comparing grains from a front and back clip

To compare grain:

1. Click Front to display the front clip.
2. Enable the Crop button and draw a crop box on the front clip.

NOTE: When the Crop button is enabled, DeGrain specifications are restricted to the area within the crop box. When the Crop button is disabled, DeGrain specifications are applied to the entire image.

3. Select Off as the Matte option.
4. Click Back.

flame displays the area of the front clip that you selected in the crop box against the back clip. You can now compare the grain of the two clips, and decide on the size or colour adjustments to apply.

To use the Result button:

The following procedure is useful when comparing grains from a front and back clip.

1. Disable the Crop button to turn off the crop box.
2. Select On as the Back option.
3. Select On as the Matte button.
4. Click Result.

flame displays the composite image of the front, back, and matte. The area of the front clip that shows through the matte is re-grained.

Adjusting the luma of the grain

Adjusting the luma of the grain changes the grain's intensity.

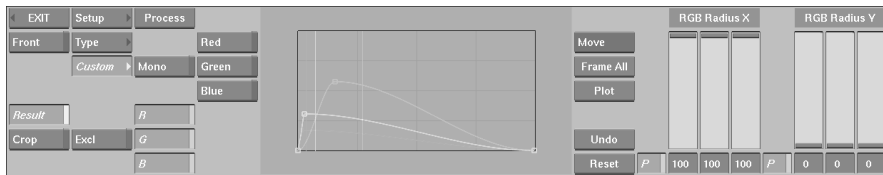
To adjust the grain luma:

1. Click the Plot button.

The colour picker appears.

2. Select an area from the front clip in which the grain strength is different than an area from the back clip.

Select a significant area of the image, such as the darkest area, the lightest area, or flesh tones. Vertical lines appear on the graph.



3. From the Edit Mode box, select Move.
4. Click the curve that you want to modify.

Select:**If:**

- | | |
|-------------|---|
| Red curve | The grain is too red or not red enough. |
| Green curve | The grain is too green or not green enough. |
| Blue curve | The grain is too blue or not blue enough. |

HINT: You can also click the Red, Green or Blue button to the select a curve.

5. Look for the intersection of each vertical line and the corresponding curve; move the control point upward or downward to increase or decrease the grain intensity.

NOTE: If there is no control point at or near the intersection of the granularity curve and the vertical line, add one using the Add option in the Edit Mode box. Keep in mind that typical granularity curves are usually smooth, and excessive control points could make the curve discontinuous. This being said, in most cases, the three first steps of this procedure should yield satisfactory results.

6. Repeat steps 1 to 5 for different areas of the front clip, until you are satisfied with the results.

Adjust the grain size

When you adjust the grain size, especially when you are trying to match grain, look at an area of intermediate brightness.

The grain in dark areas sometimes appears larger than in bright areas because of bottom clamping. Since the ReGrain command produces grain of uniform size for all luma values, it is better to first match the size with an intermediate luma value and then to boost the low luma control points by increasing their gain. This produces extreme clamping and results in a visually larger grain size for dark areas. The low luma control points are the left-most control points in the granularity curve.

To adjust the grain size:

1. Click the R, G, or B button to adjust the grain size for the red, green, or blue channel.
2. To show the selected channel as greyscale, enable Exclusive. This is useful when viewing the blue channel, as the eye is less sensitive to blue than to other colours.
3. Disable the P (Proportional) button to adjust the channels independently.
You can apply DeGrain specifications to all channels simultaneously by enabling the P button, but adjusting each one individually is far more accurate.
4. To change the horizontal size of the grain, drag the X-Radius slider for the selected channel.
5. To change the vertical size of the grain, drag the Y-Radius slider for the selected channel.

Process the Clip

Select the Matte option before processing. If you are matching two shots, make sure the matte is off before clicking the Process button.

Select:	To:
Matte On	Re-grain the part of the image corresponding to the white part of the matte.
Matte Invert	Re-grain the part of the image corresponding to the black part of the matte.
Matte Off	Re-grain the entire front clip.

Click the Process button to re-grain the clip. **flame** saves the re-grained clip on the destination reel.

Applying Grain with a Uniform Spatial Distribution

Normally, the spatial distribution of grain is different for each channel of a colour. However, if you click either the B/W button in the Type menu or the Mono button in the Custom menu, the grain will have the same spatial distribution for all three channels. The RGB gain and the RGB curves in Custom are still enabled and can be used to give a tint to the grain (useful when re-graining sepia images). Similarly, the RGB radius settings are also enabled, allowing different size and orientation for the three channels.

The procedure to apply grain with a uniform spatial distribution is the same as the one to apply or match grain except that you must enable the B/W or Mono button. For more information, see “Applying Grain to a Clip” on page 507.

[illegible]

26

Colour Corrector

Do adjust your set

Use the Colour Corrector to adjust the colour balance, colour values, and rewire the colour channels of an image.

Summary

In this chapter, you learn about:

- “Accessing the Colour Corrector” on page 515
- “Using the Histogram” on page 528
- “Previewing and Processing the Changes” on page 517
- “Remapping the Colour Values” on page 530
- “Colour Sampling” on page 520
- “Animating the Setup Values” on page 532
- “Adjusting the Colour Balance” on page 526

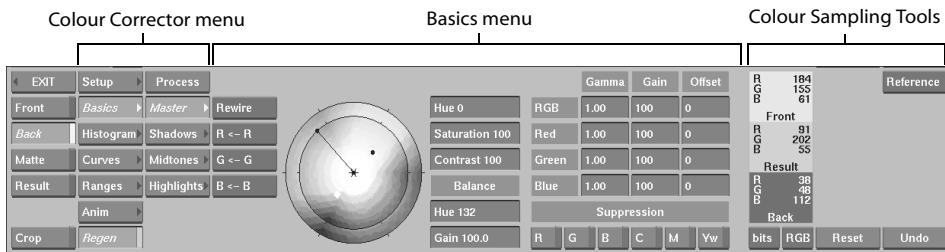
Accessing the Colour Corrector

To colour correct a single clip, load a front clip only. If you want to match colours of two clips, load a front and back clip. To create and colour correct a composite clip, load a front, back, and matte clip.

To access the Colour Corrector:

1. Click the Colour Corrector button in the Processing Menu.
2. Select an option from the Input Mode box.
3. Select the source clip.
4. Select the destination for the processed clip.

The Colour Corrector menu appears.



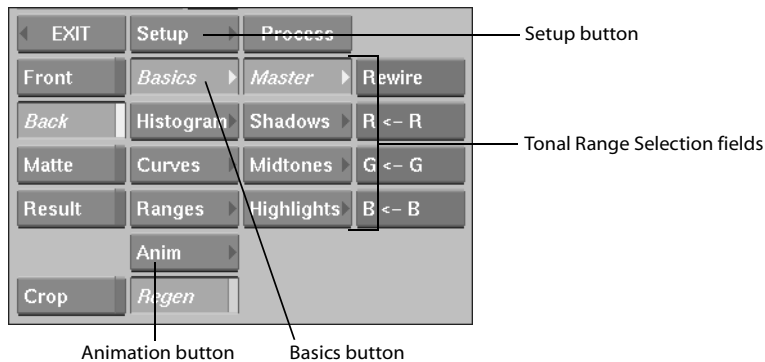
Loading a Clip from Another Menu

You can apply colour correction to any of the clips loaded into Paint, 3D Effects, or the Compositor. To enter the Colour Corrector from one of these other menus, click the C button for the clip that you want to load. The selected clip is loaded into the Colour Corrector. When you exit the Colour Corrector, you return to the previous menu.

You can also use the Colour Corrector to remove colour spill or to colour correct the front clip in the Keyer. See Chapter 33, “The Keyer.”

Colour Corrector Menu

Use the Colour Corrector menu to display the clips in the image window, access the Setup, Basics, Histo, Curves, Ranges, Animation menus, and select the luma range of the clip (shadows, midtones, or highlights) that you want to correct.



Hiding the Colour Corrector Menu

You can hide the Colour Corrector menu while you are adjusting your image if you find the menu distracting while you are trying to perform colour correction. Hide the menu when you modify numeric fields or use the colour wheel.

To hide the Colour Corrector menu and adjust numeric fields or the colour wheel:

1. Click and hold any numeric field or the colour wheel. For example, click and hold the Hue field.
2. Press the **BACK-QUOTE** (') key while continuing to hold the cursor.
3. Drag the cursor to modify the field value or adjust the colour wheel (which is hidden).
The changes you apply will be visible in the image if Regen is enabled.
4. Press the **BACK-QUOTE** (') key to reopen the menu.

Displaying the Clips

You can display each clip individually in the image window. Click the Front button to display the front clip. Click the Back button to display the back clip. Click the matte button to display the matte clip.

Click the Result button to display the result clip. The result clip shows the composite of the front, matte, and back clips.

Hot Keys

Select:	To Display:
F1	The front clip.
F2	The back clip.
F3	The matte clip.
F4	The result clip.

Use the controls located beneath the image window to change the frame displayed in the image window. Use the widescreen viewing tools in the upper right corner of the panel to enlarge and pan the image.

Previewing and Processing the Changes

You can preview the result clip before you apply the colour changes to the front clip. Click the Result button to display the result clip in the image window. You can continue to adjust the colours until you are satisfied with the results.

To apply the colour changes to the front clip, click the Process button. The processed clip is saved on the desktop. Click the Exit button in the Colour Corrector to return to the desktop. If you enter the Colour Corrector from another module, you will return to that module when you exit.

Dynamic Updating

Use the Regen button to enable dynamic updating of changes you make in the Colour Corrector. When Regen is enabled, you see the changes interactively as you adjust the values in the various displays. When Regen is disabled, you do not see changes until values are entered by lifting the stylus or mouse, or by pressing the **ENTER** key from the calculator.

To enable dynamic updating, click the Regen button.

Changing the Background Brightness

Use the Background field in the Setup menu to adjust the brightness of your work area background. Entering a higher number will increase the background colour's brightness.

To change the background brightness:

1. Click the Setup button.
2. Click the Background field and enter a new value.

NOTE: Changing the background brightness is a global setting. Changing the brightness through the Colour Corrector menu has the same effect as changing it through the Desktop Preferences menu.

Creating a Crop Box

Use a crop box to view the colour changes to a limited region of the front clip. Colour changes only appear in the area within the crop box. This feature speeds up processing, and also lets you see the difference between the source clip and result clip.

For example, while you perform colour correction on a clip, you can use the crop box to split the image window. The split window allows you to view the front clip and the result or matte clip at the same time.

To draw a crop box:

1. Hold down the **CTRL** key or hold the stylus button and drag the box across the image.
The Crop button is automatically enabled.
2. To adjust the size of the crop box, drag on the corner points of the box. To move the crop box, click on a side of the box and drag to a new location or press **CTRL** again.
You can also use the four position displays (Left, Right, Bottom, Top) in the Setup Menu.

Changing the Colour of the Crop Box

You can change the colour of the crop box so that it is easier to distinguish from the rest of your image.

To change the colour of the crop box:

1. In the Setup menu, click on the colour pot beside the Border button.
The colour picker appears.
2. Select a colour with the colour picker.
3. Click in the Border colour pot to apply the new colour to the border.

Turning the Crop Box Border Off

You can turn off the crop box border to prevent it from affecting your colour correction.

To turn the crop box border off:

1. In the Setup menu, disable Border.

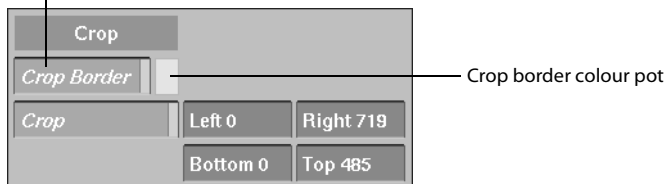
Crop Box Protocols

If you load:	Inside crop box:	Outside crop box:
A front and back clip	Shows front clip	Shows back clip
A front clip only	Shows colour corrected clip	Shows non-colour corrected clip

Adjusting Crop Box Parameters

Crop box parameters are set in the Setup menu. Click the Border button to draw a border around the crop box. You can use the colour pot to select a colour for the border.

Crop Border button



Saving Colour Correction Setups and Preferences

Once you create a colour correction setup, you can save the setup in the colour correction library. Any setup can be loaded from the library for use with another set of clips.

You can also save, load, or remove colour correction preferences using the Load or Save button:

- To save a setup or preferences, click Save.
- To load or remove a setup or preferences, click Load.

For more information on saving, loading, and removing library items, see Chapter 7, “Saving Setups and Preferences.”

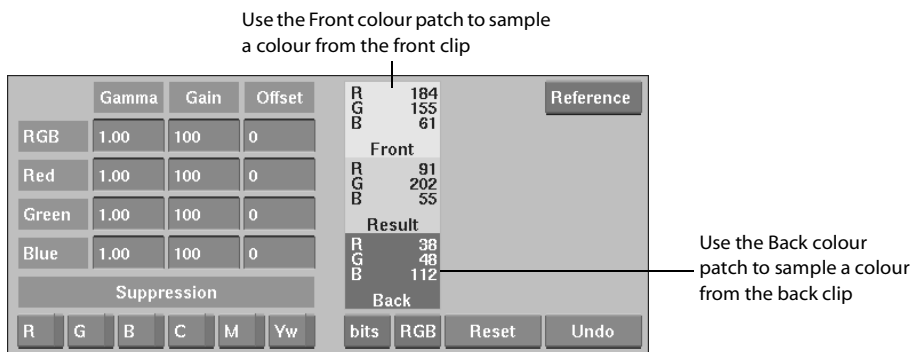
Inverting Your Matte

You can invert the matte clip by clicking the Invert Matte button in the Setup menu.

Colour Sampling

You can sample colours from the front, result, and back clips, and instantly see how the front colour is affected by the colour correction setup.

The colours are sampled using the three-colour panel in the Basics menu. The sampled colours appear in the panel with the colour values. You can display the RGB, HLS, or Y (luma) values for the sampled colours. You can also choose to display numerical sample data as either RGB bit values or percentages.



To sample a colour:

1. Select the sample option you want to use.

Select:	To:
Y	Display the NTSC luma values for the colours sampled.
HLS	Display the hue, lightness, and saturation values for the colours sampled.
RGB	Display the red, green, and blue values for the colours sampled.
Off	Turn off the colour information display.

2. Click in a colour panel.

HINT: Zoom in for more precise colour sampling.

3. Use the colour picker to pick the colour from the image and sample the colour of an individual pixel. To sample an average colour, use the colour picker and press **ALT** and drag or press **CTRL** and draw a selection box.

The sampled colour appears in the colour patch.

Colour Sampling from the Front and Back Clips

Use the colour patches to perform accurate colour matching between front and back clips.



To match colour samples from a front and back clip:

1. Click the front and the back colour patches.
2. Use the colour picker to select a colour from the image.

Two reference points appear on the colour wheel. The black point represents the colour value sampled from the front clip, and the white point represents the colour value sampled from the back clip.

Colour Sampling Using Match

If you load a front and back image, you can sample twice from the image and set key frames for the match.

For more information about colour matching using Match, see “Matching Colours Using Match” on page 531.

Colour Sampling and the Processing Order

You can use any combination of colour correction commands to modify the front clip. The commands are processed in the following order:

- RGB Rewiring, Monochrome, Negative
- Hue Shifting
- Saturation
- Colour Balancing
- Gamma, Gain, Offset, and Contrast
- Histogram
- Colour Curves
- Chroma Suppression

This processing order is important in determining the colour that appears in the middle (result) section of the colour patch. The original front colour always appears in the left section of the patch.

The middle section of the colour patch always displays the colour resulting from the application of all colour correction setups; in other words, the colour at the end of the processing order. This colour updates as you change any parameter value.

The Basics Menu

This menu contains the controls used to perform colour correction. You can:

- Adjust the hue, contrast or saturation of an image.
- Create a negative or monochrome image.
- Rewire the red, green, or blue channels of an image.
- Adjust the gamma of the individual red, green, blue, and RGB channels.
- Adjust the red, green, blue, or RGB values in an image by modifying the gain and the offset values for the channel.

To display the Basics menu, click the Basics button in the Colour Corrector menu.



Adjusting the Shadows, Midtones, and Highlights

For Saturation, Gamma, Gain, Offset, and Contrast, the shadows, midtones, and highlights ranges of the image can be adjusted together or separately.

- Click the Shadows button to adjust only the shadows, or dark areas, in the image.
- Click the Midtones button to adjust only the midlevel greys.
- Click the Highlights button to adjust the highlights, or light areas, in the image.
- Click the Master button to make the adjustment across the entire image.

Changes made in the Master range are always applied internally after the three individual ranges.

NOTE: Monochrome, Negative, Chroma Suppression, and RGB Rewiring always affect the Master tonal range.

Adjusting the Hue

You can adjust the hue of the image by changing the value in the Colour Corrector controls. Hue is the main attribute that distinguishes one colour from another. When you adjust the hue, you also change the colours in the image.

NOTE: Hue shift can be applied to individual ranges (highlights, midtones, and shadows), or the Master tonal range.

To adjust the hue of an image:

1. Click the Basics button in the Colour Corrector menu.
2. Select the range that you want to modify.
3. Click the Hue field and adjust the level of hue and saturation.
4. Click the process button to process your clip.

Adjusting the Saturation

The level of saturation determines the purity of colour by changing amount of grey in the image. Increasing the level of saturation decreases the amount of grey and produces purer colours. Decreasing the level of saturation increases the amount of grey and reduces the purity of colours.

Click the Master button to adjust the saturation over the entire image. Alternatively, click the Shadows, Midtones, or Highlights button to adjust the saturation in just the dark areas (shadows), midtones areas, or light areas (highlights) of the image.

To adjust the saturation of an image:

1. Click the Basics button in the Colour Corrector menu.
2. Select the range that you want to modify.
3. Click the Saturation field and decrease or increase the level of saturation.
4. Click the Process button to process your clip.

Adjusting the Contrast

You can control the gradations between the light and dark areas of an image by adjusting the contrast. To adjust the contrast, change the value in the Contrast field. Increase the value to increase the contrast in the image, and decrease the value to decrease the contrast in the image. The minimum value for contrast is 0, which produces a flat grey image.

To adjust the contrast in an image:

1. Click the Basics button in the Colour Corrector menu.
2. Select the range that you want to modify.
3. Click the Contrast field and decrease or increase the contrast value of the image.
4. Click the Process button.

Rewiring the RGB Channels

Use the Red, Green, and Blue Channel Rewiring fields to rewire the red, green, and blue channels of an image. When you rewire a channel, the values for the current colour channel are replaced with those of the new channel. For example, if you select the $R \leftarrow G$ option in the Red Channel Rewiring field, the colour values of the red channel of the image are replaced with the colour values of the green channel.

The RGB Rewiring commands can be applied to the Master range only.

To rewire the red channel:

1. Click the Basics button in the Colour Corrector menu.
2. Select an option in the Red Channel Rewiring field.

Select:	To:
$R \leftarrow R$	Use colour values for the red channel.
$R \leftarrow G$	Replace colour values of the red channel with those of the green channel.
$R \leftarrow B$	Replace colour values of the red channel with those of the blue channel.
$R \leftarrow Y$	Replace colour values of the red channel with the luma of all channels.
$R \leftarrow 1-R$	Replace colour values of the red channel with its inverse. (For instance, if $R = 100$, $R \leftarrow 1-R = 155$.)

3. Click Process.

NOTE: The green and blue channels are rewired in the same way as rewiring the red channel (described above).

Creating a Monochrome or Negative Image

Convert a colour image into a monochrome or negative image using the Rewire option box.

To convert a colour image into a monochrome or negative image:

1. Click the Basics button in the Colour Corrector menu.
2. Select Mono or Negative from the Rewire option box.
The image is automatically converted to a monochrome or a negative image.

3. Click Process to apply the changes to the clip. To cancel the changes, click Reset.

You can also convert the image manually by clicking on the appropriate Rewire selection box (R, G or B) and selecting the Y option for monochrome or $1-R$ for a negative image.

HINT: You can also create a monochrome or negative image using the Monochrome and Negative buttons on the Processing menu. For more information, see “Creating a Monochrome Clip” on page 501, and “Creating a Negative Clip” on page 502.

Adjusting the Gamma

You can adjust the grey values of an image by adjusting the gamma. This allows you to brighten or darken an image without greatly affecting the shadows or highlights.

To adjust the gamma, use the Gam fields in the RGB controls.

To adjust the gamma of the image:

1. Click the Basics button in the Colour Corrector menu.
2. Select the tonal range that you want to modify.
3. Click the Gam field and decrease or increase the gamma value.

Adjusting the Gain and Offset

You can boost the colours in the image by increasing the Gain and Offset values. The Colour Corrector multiplies the pixel colour values by the Gain, and then adds the Offset value. The resulting colour values are clipped at the maximum value of 255.

You can also reduce the colours in the image by decreasing the Gain and Offset values. The resulting colour values are clipped at the minimum value of 0.

Gain is expressed as a percentage value. The default value of 100% has no effect on the image since the colour values are multiplied by 1.

NOTE: You can adjust the Gain for individual ranges, or across all ranges simultaneously (by using the Gain fields in the Global controls). Increasing the Gain in the RGB channel has the same effect as increasing the brightness of the image.

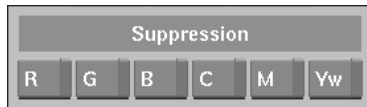
To boost or reduce the colours in the image:

1. Click the Basics button in the Colour Corrector menu.
2. Select the range that you want to modify.
3. Click the Gain field for the colour channel that you want to modify and enter a new value.
4. Click the Offset field for the channel that you want to modify and increase or decrease the Offset value.
5. Click the Process button to process your clip.

Using Chroma Suppression

Use the Chroma Suppression buttons to suppress the RGB or CMY_w colours in an image. To suppress one of these colours, disable the designated button for that colour. The status box on the button is light blue when the button is disabled.

Chroma suppression is useful for removing blue spill from a blue-screen composite.



To suppress a colour:

1. Click the Basics button in the Colour Corrector menu.
2. Click the Master button. Chroma suppression can only be applied to the Master tonal range.
3. Click the appropriate button for the colour to suppress.

Click:	To suppress:
R	Red
G	Green
B	Blue
C	Cyan
M	Magenta
Y _w	Yellow

NOTE: Suppression of RGB and CMY_w always reduces luma.

Adjusting the Colour Balance

You can adjust the colour balance in an image using the colour wheel menu. The two parameters used with colour balancing are Tint and Strength. The value of Tint determines the colour to add and the value of Strength specifies the amount of colour to use.

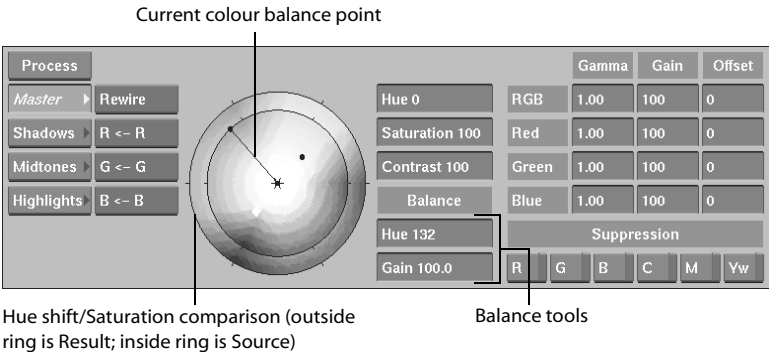
To change the colour balance, you can either set the values in the Tint and Strength displays or use the colour wheel.

Pure red is the 0 degree point for hue on the colour wheel. When you increase the Tint value, you move counterclockwise on the colour wheel. When you use a negative value for Tint, you move clockwise on the colour wheel.

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

NOTE: If Saturation is set to 0, then no colour balancing is performed, regardless of the Hue value.

Use the colour wheel and the Tint and Strength fields to adjust the colour balance of an image. The Tint value determines the colour to add and the Strength value specifies the amount of colour to use.



NOTE: In the Channel Editor, the Tint and Strength channels are labelled hue_balance and gain_balance, respectively.

Creating and Modifying a Colour Balance Point

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

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

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

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

The shadows, midtones, and highlights of the image can be adjusted together or separately.

To change the colour balance in the image:

1. Click the Basics button.
2. Select the range that you want to modify.

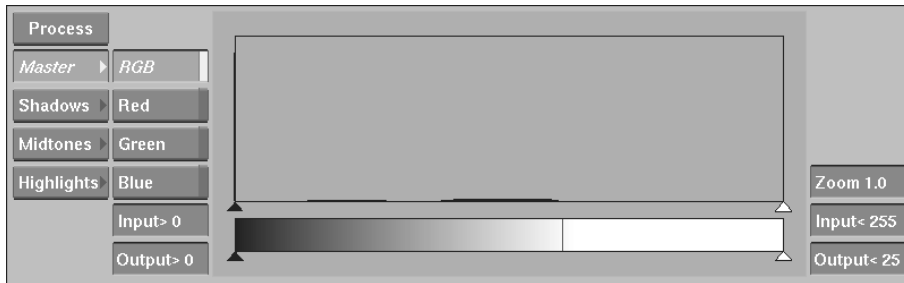
3. Position the cursor over the centre point on the colour wheel. Click and drag toward the colour that you want to use.

As you drag the cursor, a colour balance point follows the cursor movement. Release the cursor to view the effect on the image.

4. If you are not satisfied with the resulting image, click and drag the colour balance point again. You can continue to move the colour balance point until the desired effect is achieved.
5. Click the Process button to process your clip.

Using the Histogram

You can use the histogram to adjust the range of values used for the red, green, blue, or RGB channels of an image. Click the Histogram button in the Colour Corrector menu to display the histogram.



The histogram shows the colour distribution of pixels in the front image for the selected channel. The horizontal axis of the histogram represents the values of pixels, from 0 at the left to 255 (white) at the right. The vertical axis represents the number of pixels with these colour values.

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

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

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

If you set the maximum value for the range to 150, pixels with values greater than 150 are remapped to 255 (white). Pixels with colour values less than 150 are remapped to the corresponding values. This brightens the image and increases the contrast in the highlight areas.

You can also set the maximum and minimum limits for the colour range by entering the values directly in the Max and Min numeric displays on either side of the histogram.

To increase the contrast in an image:

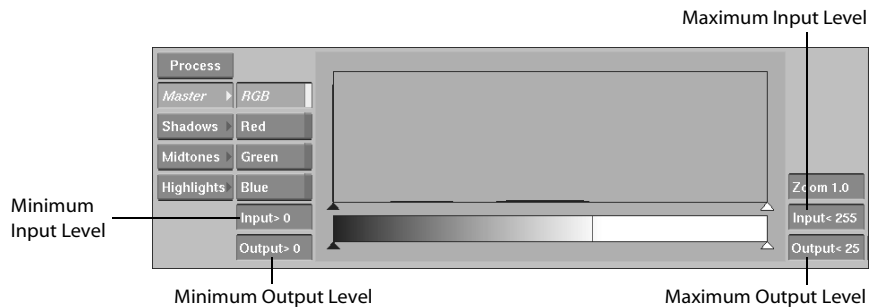
1. Click the Histogram button in the Colour Corrector menu to display the histogram.
2. Click the Master, Shadows, Midtones, or Highlights button to select the parts of the image that you want to modify.
3. Click the RGB, Red, Green, or Blue button to select the channel that you want to work with.
4. Position the cursor on the black triangle below the histogram. Adjust the level until the value that you want to use as the lower limit for the colour range appears in the Min display. This darkens the shadow areas.

Or:

Position the cursor on the white triangle below the histogram. Adjust the level until the value that you want to use as the upper limit for the colour range appears in the Max display. This increases the brightness in the highlight areas.

NOTE: If you are in RGB, you can invert the image by reversing the order of the black and white triangles.

5. Click Process to apply the changes to the clip. To cancel the changes, click Reset.



Setting the Output Levels

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

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

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

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

You can also set the maximum and minimum limits for the colour range by setting the values directly in the Max and Min numeric displays.

To reduce the contrast in an image:

1. Click the Histogram button in the Colour Corrector menu to display the histogram.
2. Click the Master, Shadows, Midtones, or Highlights button to select the parts of the image that you want to modify.
3. Click the RGB, Red, Green, or Blue button to select the channel that you want to work with.
4. Position the cursor on the black triangle below the Output Levels bar. Adjust the level until the value that you want to use as the lower limit appears in the Min display. This has the effect of lightening the shadow areas.

Or:

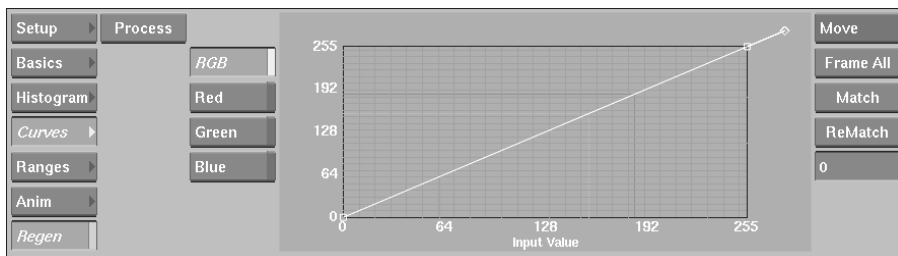
Position the cursor on the white triangle below the Output bar. Adjust the level until the value that you want to use as the upper limit appears in the Max display. This has the effect of decreasing the brightness in the highlight areas.

You can also set the upper and lower limits by entering the values directly in the Max and Min numeric displays.

Remapping the Colour Values

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

Click the Curves button in the Colour Corrector menu to display the Curves menu.



The Colour Curves

There is one colour curve for each of the red, green, blue, and RGB channels of the image. The colour curves are generated by plotting the input values for the source image versus the output values for the resulting image. The input values range from 0 to 255, and are plotted along the horizontal axis. The output values range from 0 to 255, and are plotted along the vertical axis.

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

The colour values of an image are remapped by changing the shape of the colour curve. The curves editor can be used to add, delete, or move control points on the curves. For example, you can move the end control points on the curves to change the maximum and minimum values for the colour range. You can add a control point anywhere on the curve to remap a particular colour value.

The curves editor has similar controls to the Channel Editor. For more information, see “Animation Controls” on page 132.

Matching Colours Using Match

The Match button allows you to quickly sample a colour from the back clip to the front clip, and adjust the result with the colour curves.

To use Match:

1. Load a front and back clip from the desktop.
2. Click the Front colour patch.
The colour picker appears.
3. Using the colour picker, select a colour in the front clip to be corrected.
The colour is added to the Front and Result colour patches.
4. Click the Back colour patch.
The colour picker appears.
5. Using the colour picker, select a colour in the back clip to match to.
6. Click Match.

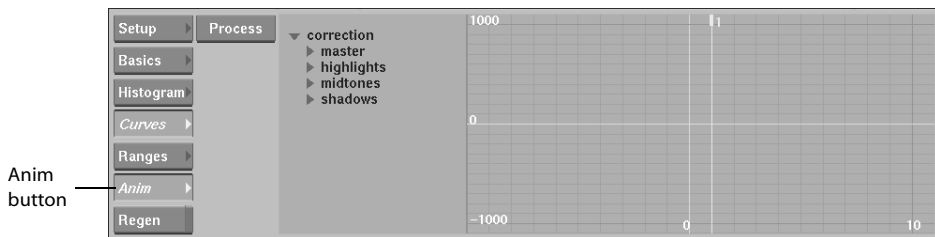
The colour is added to the Result colour patch and points are added to the colour curves.

When you use Match, a number is displayed in the numerical field below the Rematch button. The first match you perform is indicated by a 1 in the field, the second by a 2, and so on. To revert to a previous match, click the numerical field and type the number of the match you want to recall. The colour patches and colour curves reset to the values stored in the specified match.

To reset all the colour curves, click Reset. To reset the red, green, or blue curve only, select a curve and click Reset Curve.

Animating the Setup Values

Click the Anim button in the Colour Corrector menu to display the Animation menu. You can use the Animation controls to animate any of the numeric parameters in the setup. For more information on Animation, see Chapter 9, “Animation.”



Setting Keyframes Automatically

A keyframe consists of a channel value and the specific frame in which the value is set. A keyframe is saved automatically each time you change a channel value. You can turn off the automatic keyframe feature by disabling Auto Key in the Setup menu.

When Auto Key is disabled, you must explicitly set and remove keyframes using the Set Key and Rem Key buttons in the Anim menu. The procedures for using Set Key and Rem Key are described in “Setting and Removing Keyframes” on page 151. The Auto Key button is found in the Setup menu of all modules that use the Channel Editor.

27

Colour Warper

Another dimension of colour space

The Colour Warper allows you to manipulate colour space with intuitive controls that provide precise colour correction, and histograms that help you visualize your image's colours.

Summary

In this chapter, you learn about:

- “Accessing the Colour Warper” on page 534
- “Fundamentals” on page 534
- “Using the 2D Vectorscope and RGB Viewer” on page 537
- “Viewing Clips in the Reference Buffer” on page 541
- “Colour Matching” on page 541
- “Basics Menu” on page 542
- “Warp|Gam Menu” on page 545
- “Selective Menu” on page 547
- “Subsetups” on page 552
- “Hot Keys” on page 553

About the Colour Warper

The Colour Warper is a node that you drag and drop into the Batch process tree or the Modular Keyer's processing pipeline. You use the Colour Warper to colour correct clips, fine-tune mattes, adjust specific colours and colour ranges, refine keys, and match colours or overall saturations.

Accessing the Colour Warper

The Colour Warper is available in Batch and the Modular Keyer. You may want to use the Colour Warper from the Modular Keyer to refine a key, or access it from Batch to perform colour correction to your clips.

NOTE: It is recommended that you be familiar with using the Batch schematic and the Modular Keyer processing pipelines before using the Colour Warper. For more information, see Chapter 29, “Batch Processing,” and Chapter 34, “The Modular Keyer”.

To add a Colour Warper node in Batch:

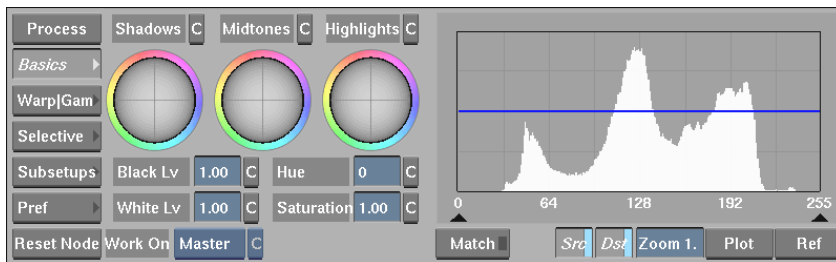
1. Click Batch from the Processing menu.
2. Select a destination reel.
3. Drag and drop a Colour Warper node to the process tree.
4. Click the Colour Warper node to display the Basics menu.

To add a Colour Warper node in the Modular Keyer:

1. Click Modular Keyer from the Effects menu.
2. Select a front, back and key-in clip.
3. Drag and drop a Colour Warper node to the front, back, or key-in clip's pipeline.
4. Click the Colour Warper node to display the Basics menu.

Fundamentals

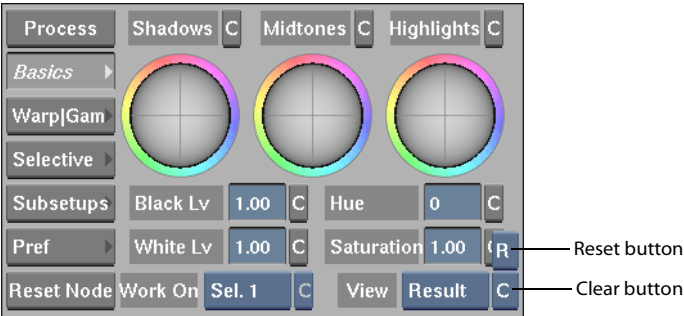
When you click the Colour Warper node in either Batch or the Modular Keyer, the Colour Warper menu appears:



Use:	To:
The Basics menu	Adjust shadows, midtones, and highlights, and to control black and white levels, hue, and saturation. See “Basics Menu” on page 542 for more information.
The Warp Gam menu	Perform colour warping and adjust gamma. See “Warp Gam Menu” on page 545 for more information.
The Selective menu	Define colour ranges using softness and tolerance, and apply colour correction to a selected region. See “Selective Menu” on page 547 for more information.
The Subsetups menu	Store, compare, and share Colour Warper subsetups. See “Subsetups” on page 552 for more information.
The Pref menu	Adjust vectorscope and hue cube settings. See “Setting Vectorscope Preferences” on page 540 for more information.

Using Reset and Clear

Each menu in the Colour Warper has an individual reset and clear function for each control. Use C to clear and reset a control for the current frame only. Use R to reset the control for all the keyframes in the animation curve.

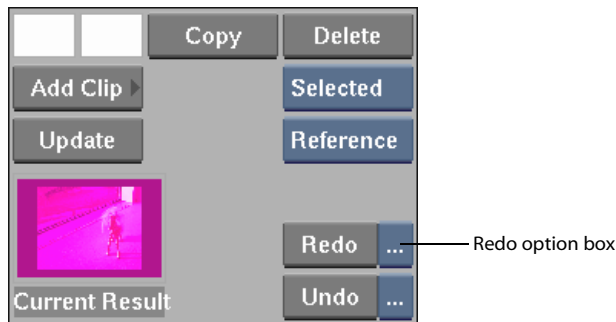


If there is no keyframe, use the C buttons to automatically set a keyframe at the current frame and to add a keyframe in the animation channel under Matrix.

NOTE: The trackball and Luma field in the Warp|Gam menu cannot be reset. Use the Undo box if you want to undo these controls (see the following section).

Using Undo and Redo

Use the Undo and Redo boxes to remove or redo colour corrections to your clips. Undo and Redo store the last 10 actions performed in the Colour Warper. To undo or redo a function, click either box and select an option from the list.



Processing

Processing in the Colour Warper is extremely efficient and preserves the original data. Functions are concentrated into one single processing engine: shadows control/level, highlights control/level, hue shift, saturation, matching, midtone, and gamma adjustment are all part of the same calculation.

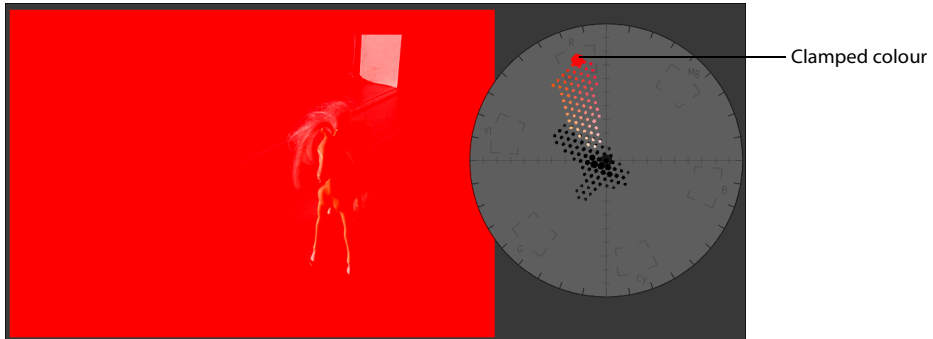
The processing engine's setup contains three shape channels: Midtones, Matrix, and Gamma (RGB). All operations can be animated, such as midtones, RGB curves, Master and Gamma curves. For more information about animation, see Chapter 9, "Animation."

When you use the Colour Warper, it is recommended that you process your clip in the following order: set whites, set blacks, and adjust midtones.

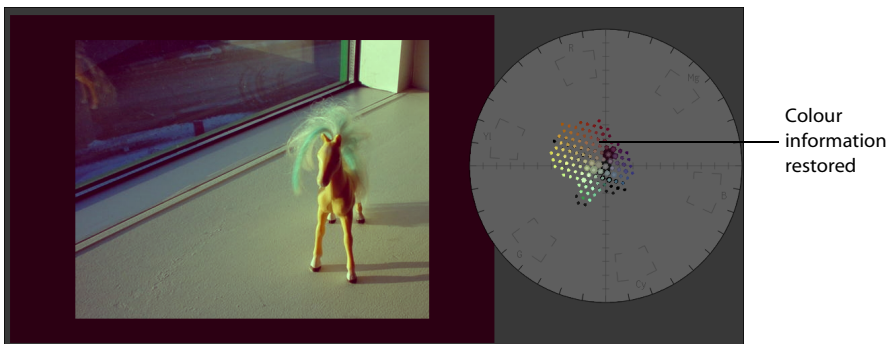
Colour Clamping

The use of a single matrix provides faster processing and preserves image quality— each step is independent of the others. For example, if you clip colours using white level, it is still possible to retrieve highlights that are clamped by using midtones or the controls in the Warp|Gam menu.

Clamping occurs when colours go beyond the RGB range and colour information is lost. If you were using the Colour Corrector, colours lost in clamping would be irretrievable and you would need to reset or clear your colour correction.



In the Colour Warper, however, lost colour information is retrieved simply by readjusting the colour correction using the trackballs.

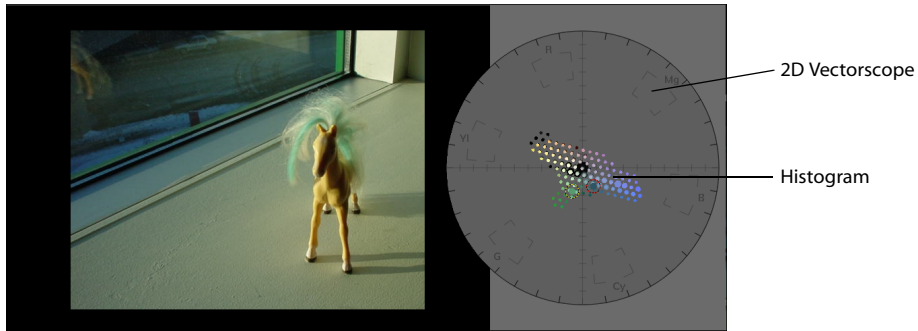


Using the 2D Vectorscope and RGB Viewer

Use the vectorscopes to help you match colours, adjust shadows and highlights, view plotted and reference colours, and colour correct clips. The colours in the image are represented by cubes in the vectorscope's histogram. As you colour correct a clip, the cubes move accordingly and the histogram updates simultaneously to reflect any changes. Colours you plot, reference colours, source and destination colours are also distinguished in the histogram. See “Matching Colours with the Histograms” on page 539 and “Viewing Source and Destination Colours” on page 539 for more information.

The 2D Vectorscope

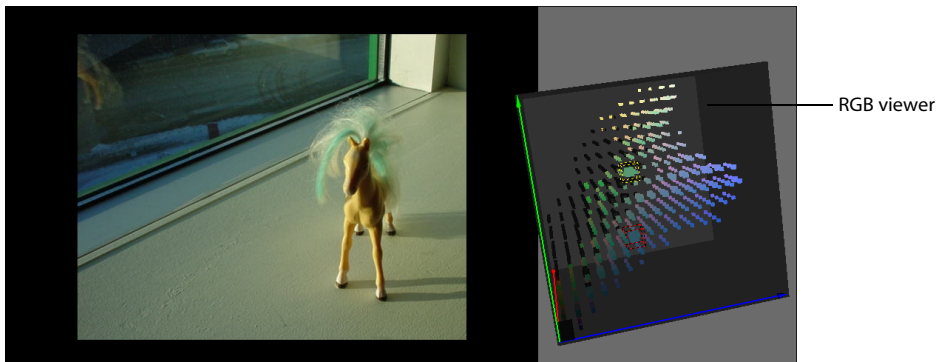
The 2D vectorscope provides a two-dimensional colour map of your clip. The cubes of colour are a colour model of your destination clip, and plotted and reference colours are represented by outlined colour cubes.



The RGB Viewer

The RGB viewer is a three-dimensional colour model of the clip. Its X, Y, and Z axes represent the red, green, and blue components of the clip's colour space. The cubes of colour show the distribution of all the colours of the clip in RGB colour space. This three-dimensional colour map is referred to as the 3D histogram. When you scroll through a clip, the 3D histogram cubes update to display the colour distribution of each frame.

Use the 3D histogram to analyze the colour mapping of the clip and to locate specific colours.



Switching from the 2D Vectorscope to the RGB Viewer

Rotate the 2D vectorscope (**CTRL** and drag) to switch to the RGB viewer. Alternatively, double click the vectorscope to toggle between the 2D vectorscope and the RGB viewer.

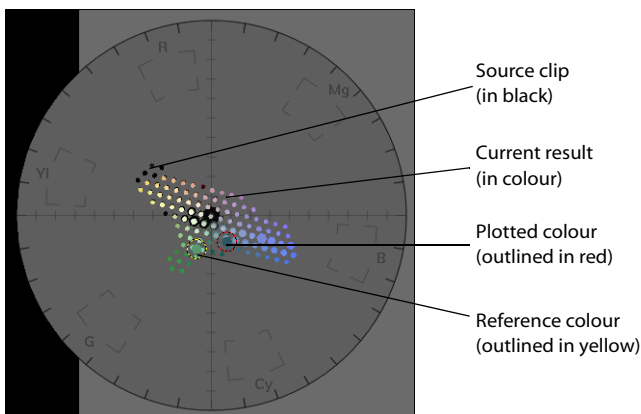
Viewing Source and Destination Colours

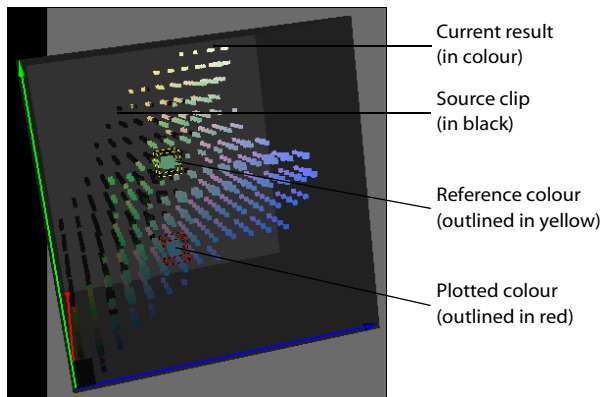
In the Basics and Warp|Gam menus, use the Src and Dst buttons to display the source and/or the destination colours in the histogram. Use Src to see a view of the image before colour correction has been applied and use Dst to provide real-time representation of the corrected image. When both Src and Dst are enabled, you can view both the original image and the colour-corrected result

Your source clip is displayed in black in the histogram. This allows you to view the original colour information as you make changes to the image, while distinguishing this information from your current result by keeping it in black. The clip's current result is displayed in colour, providing a visual representation of its colour information. As you use the controls in the Colour Warper, the current result display updates simultaneously and reflects any changes that you make.

Matching Colours with the Histograms

The current result display is useful when matching plotted and reference colours. For example, to make sure that a colour is truly black, pick a black in a reference image using the Plot button, and use the trackballs to drag the colour to the centre of the histogram. The plotted colour is outlined in red to distinguish it from the other colours in the image, and moves with the rest of the colours in the histogram as you use the trackballs. Reference colours are outlined in yellow, and represent a colour picked from a clip in the reference buffer. You can match plotted and reference colours by using the trackballs and aligning the colours in the histogram. For more information on using Plot and Ref, see “Using Plot and Ref” on page 544.



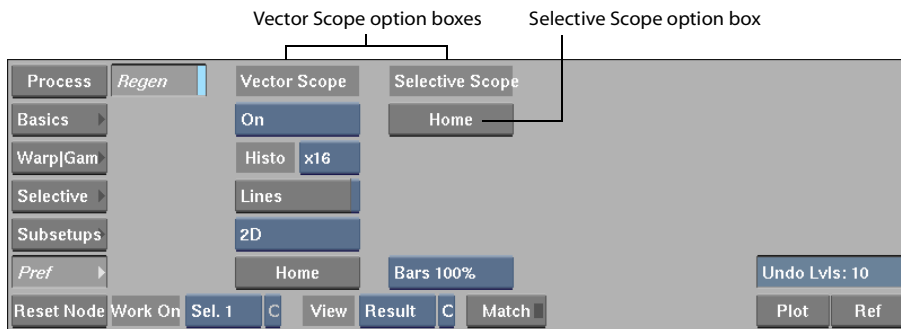


Setting Vectorscope Preferences

Change the display at any time to help you best visualize the contents of your image and work with the viewer. Click and hold **CTRL** to rotate the RGB viewer, or click and hold **SHIFT** to resize it. To move either vectorscope, click the 2D vectorscope or RGB viewer, click **ALT** and drag.

Use the Pref menu to:

- Adjust vectorscope settings
- Return the vectorscope and hue cube to their default size and location



Click:	To:
On or Off	Toggle the vectorscope on or off.
Histo options	Set the vectorscope's histogram to display x8, x16, or x32.
Lines	Enable lines to be displayed in the RGB viewer.
2D or 3D	Toggle between the 2D vectorscope and the RGB viewer.

Click:	To:
Home (Vector Scope)	Reset the vectorscope to its original size and location.
Home (Selective Scope)	Reset the Selectives menu hue cube to its original size and location.
Bars 100% or Bars 75%	Match 75% and 100% SMPTE bars.

Viewing Clips in the Reference Buffer

The split bar provides two different view modes simultaneously while you use the Colour Warper. This is useful when performing colour matching because you are able to see both your reference and destination clips when picking a range of colours to match.

You can toggle the split bar on or off in the Setup menu. Click Setup and select Split On or Split Off.

NOTE: You must click Grab in the Setup menu to view the clip in the reference buffer. Otherwise, the reference buffer will be black.

Grab

In the Setup menu, click Grab to view the front, key-in, back, matte, context 1, context 2, or result in the reference buffer with your sample clip.

Flush

In the Setup menu, click Flush to clear the reference buffer.

For more information on the reference buffer, see Chapter 4, “Fundamentals.”

Colour Matching

Use the Match feature to match a range of colours to a colour range in a reference clip. You can match whites, blacks, or the overall saturation of an image.

To use Match:

1. Make sure your view mode is set to CurResult.
2. In the Selective menu, click Match.
A “MATCHING: Select an area to be modified.” message is displayed.
3. Click and drag on the CurResult clip to draw a box around a range of colours. You can select an area of the image that is predominantly black, white, or contains a wide spectrum of colours.

The blue indicator on the Match button is enabled, indicating that you need to select a destination sample.

NOTE: To match the overall saturation of an image, select a larger area of the image.

4. Click Match.

A “MATCHING: Select an area to match to.” message is displayed.

5. Click and drag to draw a box around a range of colours to match to in the reference clip.

NOTE: It is recommended to match whites with whites, blacks with blacks, and to select a broad range of colours on both images to match saturation. Trying to match completely different colours will produce unpredictable results.

6. Repeat these steps as often as necessary to obtain the best possible match between colours.

You can use Match repeatedly without destroying the result. Instead, your colour match becomes more precise. You can use other controls in the Basics menu (such as Saturation) to refine the result, then reuse Match again.

Each match you perform can be cleared by using the Undo box. Click the Undo box and select a colour match to remove. Additionally, if you begin using Match and want to cancel the matching process, adjust any value using the trackballs or fields. The blue indicator on the Match button will be disabled, indicating that the matching process has been cancelled.

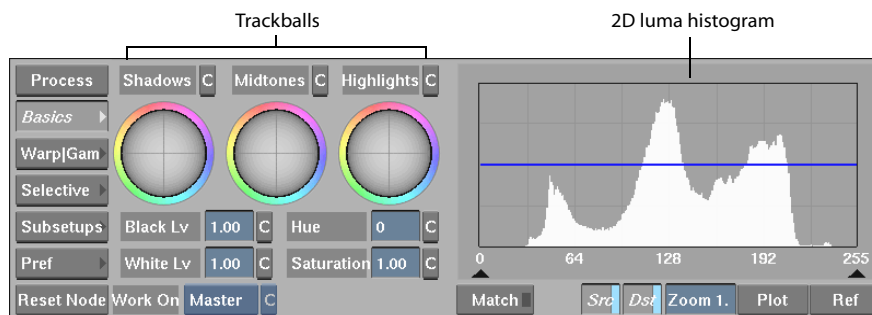
Use the Black Lv and White Lv fields in the Basics menu to match the luminance of a colour to the luminance of a source colour. See “Colour Correcting with the Numeric Fields” on page 543 for more information.

Basics Menu

Use the Basics controls to colour correct the result image and refine a matte you create using the Selective controls. For more information about creating mattes in the Selective menu, see “Selective Menu” on page 547.

Use the Basics menu to:

- Control shadows, midtones, and highlights balances with luma preservation.
- Define the midtone range to be affected.
- Control black & white level, hue, and saturation.



Colour Correcting with the Trackballs

Drag the trackballs to move the shadows, midtones, and highlights towards a specific colour range. The trackballs change the colour in your clip according to the direction the trackball is being dragged. For example, to add more yellow to the image's highlights, drag the Highlights trackball towards yellow.

Once the shadows and highlights are properly balanced, you can modify the midtones by using the Midtones trackball to give a specific tint (while still preserving luma) to the overall appearance of the image. Alternatively, use the Hue and Saturation fields.

Use the:	To:
Shadows Trackball	Modify the chroma in the low luminance area, but not affect the luma.
Midtones Trackball	Add horizontal RGB curves in the 2D luma histogram. These curves represent the amount of chroma tint you apply to the image. For example, if the red and green curves are above the middle line while the blue curve is below the middle line, you are adding a yellow tint to the midtones components of your image.
Highlights Trackball	Modify the chroma in the high luminance area, but not affect the luma.

Colour Correcting with the Numeric Fields

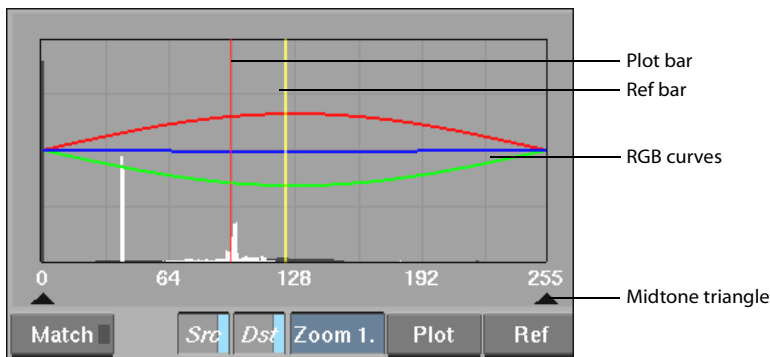
The BlackLv field affects the luma values in the image's shadows and keeps the chroma at the same position. The WhiteLv field affects the luma values in the image's highlights and keeps the chroma at a constant level. Hue “turns” the chroma (rotates it in the 2D luma histogram) and keeps its relative value intact. The Hue field does not affect the luma. Saturation amplifies or decreases the chroma in an image, and keeps the luma at a constant level.

Using the 2D Luma Histogram

The 2D luma histogram shows transformations of the main gamma control. You can define a region to be affected by the Midtones trackball by dragging the triangles.

By default, the Midtones trackball affects all ranges except white and black in varying degrees, as shown by the histogram's curves. To change the luminance range, use the two triangles below the histogram to restrict the regions affected by the Midtones trackball. Click on the histogram and drag the triangles closer together to limit the range of midtones by limiting the Midtones trackball's movements. The RGB curves only change within the area defined by the triangles. You can modify the range as many times as you like.

NOTE: You can animate the curve using the Shape channel under Midtone. For more information on animation, see Chapter 9, “Animation.”



Using Plot and Ref

Use Plot to sample a colour from your CurResult clip and move its position in the vectorscope, and to adjust its lightness value in the 2D luma histogram. For example, Plot is useful for selecting a colour to modify when performing a colour match using Ref.

You can also use Plot to get more information on colours you want to change. For example, when you are trying to adjust colours and levels and are not sure which controls to use, click Plot and select a colour in your image. If the plotted colour appears in the middle of the histogram, use Midtones. If the plotted colour appears on the left, use Shadows.

Use the Ref button to plot any pixel you see on the screen. For example, you can select a colour from images displayed in the reference buffer to match to a colour in another image.



You can use the Ref representation as a target and the Plot as the moving representation. In general, you move the Plot representation closer to the Ref representation in the vectorscope and 2D luma histogram. Plot is represented by a red bar and Ref is represented by a yellow bar in the 2D luma histogram. In the vectorscope, Plot is represented by a red circle around the plotted colour and Ref is represented by a yellow circle around the reference colour.

NOTE: The Ref representation does not move while you are colour correcting the image.

Plot can be used to sample shadows in your image and adjust the blacks in your clip. For example, you may want to sample a black colour in your clip and adjust its intensity.

To sample shadows in the image using Plot:

1. Click Plot and drag the cursor over the portion of the CurResult clip containing black.

NOTE: To get an average box for Plot or Ref, press Plot or Ref and hold **CTRL** while dragging on the image.

2. Using the Shadows trackball, move the colour to the center of the vectorscope to ensure it is truly black. Alternatively, you can move the trackball until the shadows are the desired colour.

A red plot line is displayed in the 2D luma histogram, indicating the intensity of the sampled blacks.

3. Adjust the black level using the trackballs or the BlackLv field.

HINT: Whites can be sampled using the same steps.

Ref can be used to match a specific colour in a reference clip to a colour in another clip. For example, you may want to match a red in one image to a different shade of red in another clip.

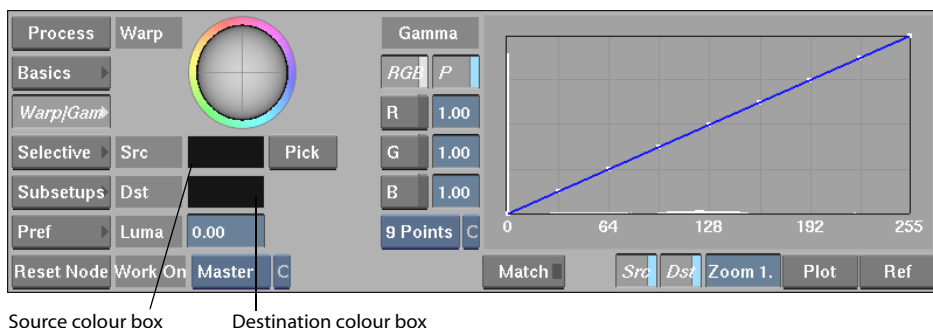
To match colours using Ref:

1. Click Plot and pick a colour in the clip you want to affect.
2. Click Ref and pick a colour to match to.
3. Use the trackballs to match the plotted and reference colour in the vectorscope. Move the plotted colour until it is over the reference colour in the vectorscope.

Warp|Gam Menu

Use Warp controls to pick a colour from your image and warp colour space in a localized area around that colour. You use Warp controls to perform fine adjustments after a colour correction has been completed. For example, you can adjust a blue-screen key if skin tones are discoloured at the edges, or readjust a colour that has become altered from other controls in the Colour Warper.

Use the Gamma controls to adjust the gamma of the R, G, and B levels in your image.



Warping Colours

When you colour correct clips, sometimes colours you do not want to alter are modified in the colour correction process. If you want to readjust a specific colour, select the colour again and adjust it with the Warp controls. This process can be repeated as many times as necessary.

Warping may need to be performed a few times before achieving the desired result.

To warp colours, you select a colour from the image and use the trackball and Luma field to adjust the colour. The selected colour is indicated by a white circle around the colour in the 2D vectorscope, a white cube around the colour in the RGB viewer, and a cyan bar in the 2D luma histogram. This helps you distinguish the colour you pick from the other colours in your clip.

To warp colours:

1. Click Pick and select a colour you want to adjust in the image.

2. Click the Src colour box to set the source colour.

The Dst (destination) colour box updates to match the source colour.

3. Adjust the destination colour using the trackball and Luma field.

The image is colour corrected locally.

4. Release the trackball or click the Luma field.

The Src colour box is set to the Dst box colour.

Suppressing a Hue Range

You can use the Warp controls to suppress a specific hue range. Select the colour that represents the range you want to suppress, and use the trackball to desaturate the colour by bringing it to the middle of the vectorscope.

To suppress a hue range:

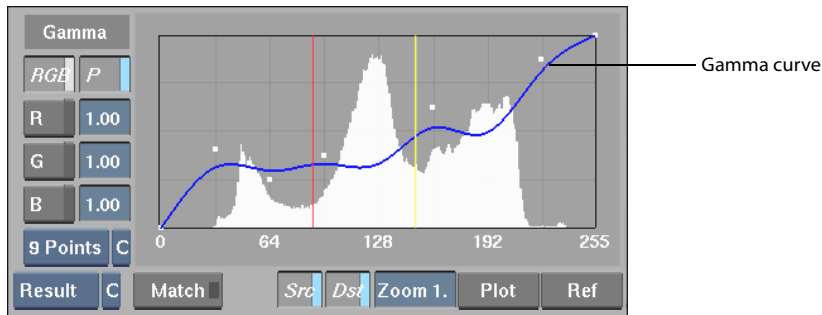
1. Click Pick and select a colour range to suppress.

2. Click Src to add the colour to the Source colour box.

3. Use the trackball to move the colour to the middle of the vectorscope.

Adjusting Gamma

Use the Gamma controls and curves to change gamma for R, G and/or B, and use three or nine control points to make fine adjustments to the curves. The 3 Points option allows global modification while the 9 Points option is recommended for fine-tuning adjustments. Enable the P button to adjust gamma proportionally, or disable P to adjust R,G, and B separately.



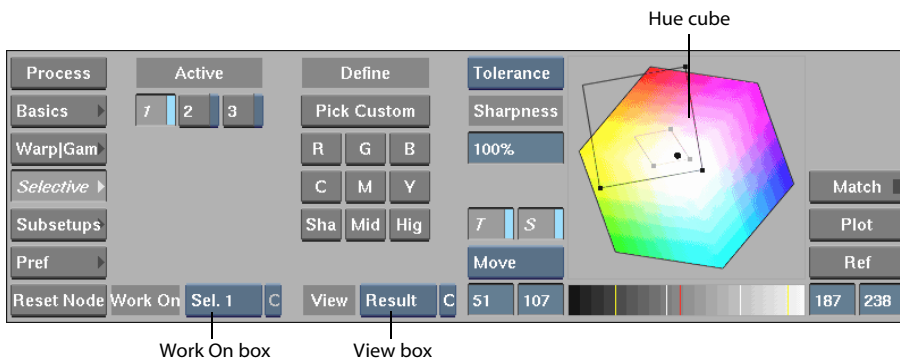
To adjust gamma, click on any point of gamma curve and move point up or down. Alternatively, you can click and drag in the R, G, and B fields or double click in any field and type a value.

HINT: To raise the contrast in an image, you can lower the gamma of the RGB curves. To adjust the black and white portions of the image, adjust the Black Lv and White Lv in the Basics menu. To smooth the contrast, increase the gamma and adjust the Black Lv and White Lv.

To clear or reset the Gamma control points, press C or R next to the 3 or 9 Points box. C resets the points on the curve for the current frame while R resets the points on the curve for the entire animation. To reset the Gamma numeric fields, **CTRL-CLICK** on a field.

Selective Menu

Use the Selective menu to colour correct specific colours, as opposed to a global correction to the master image. You can have up to three selectives cascaded in order after the master correction.



The Work On box indicates if you are working on the master or one of the selectives. To colour correct a specific range of colours in a selective, you begin by defining a matte that will characterize the range. Then, you use any of the tools in the Basics and Warp|Gam menu to achieve a correction associated to the selective you choose in the Work On box. This correction will only be applied where the matte is black. You output the matte by selecting Matte from the View box and by using the matte to perform a chroma key.

Once your selection is complete, all controls in the Basic and Warp|Gam menu will be applied to the region defined by the matte in the Work On selective.

Use the Selective menu to:

- Define up to three colour ranges using softness and tolerance.
- View the matte, a selected region, or the result.
- Apply color correction to a region using the Basics and Warp|Gam menus.

The Active buttons display the results from the Work On selectives when your view mode is set to CurResult.

Viewing the Defined Area

Use the View modes to help you adjust and refine your matte.

Select:	To display:
Matte	The matte through which the secondary correction will be done.
InvMatte	The inverse of the matte.
Selection	The colours of the image that you pick with the Define buttons through the greyscale display.
Result	The final colour correction.

When you choose one of the selectives from the Work On box, the View box is set to Selection if no selective range has been defined, and the C button next to the Work On box will be

disabled. When you determine a selective range using the Define controls, the range becomes visible through the greyscale display and the C and R buttons are enabled.



Original colour visible through the greyscale display.

NOTE: Switching to either the Basics or Warp|Gam menu will automatically switch the view to Result.

Selective Colour Correction

Selective colour correction is applied to a specific colour range. You can define up to three selective corrections.

To perform Selective colour correction:

1. Select Sel1 from the Work On option box. Only the region defined for Selective 1 will be affected.

The image turns to greyscale if you have not defined a selective range.

2. Define a matte using any of the Define controls.

If you select Pick Custom, drag the cursor over the image to define your selection.

NOTE: The Active 1 button is enabled automatically after you click a Define button. You can click the Active 1 button to disable it if you do not want to apply this condition to the global result.

3. Refine your matte by using Tolerance, +Softness, or -Softness from the Adjusting box. Select +Softness to add softness and -Softness to remove softness from the matte.
4. Drag the cursor over the image or drag and click **CTRL** to draw a box. Alternatively, move the handles of the tolerance or softness lozenge on the hue cube or make adjustments using the luma range.

For more information on using the luma range, see “Using the Luma Range” on page 551.

HINT: Use the Sharpness field to adjust softness and help with noise if the matte appears grainy.

You can create a matte to use in the Keyer.

To create a Keyer matte:

1. Set Work On to Sel 1.
2. Set View Matte.
3. Set your key using the Define controls.
4. Refine the matte as needed by adding or removing tolerance and softness. Use the boundaries in the hue cube or sample the image to fine-tune the matte.

The output will be a matte in the processing pipeline.

HINT: To get a good key, make sure that you have enough processing speed to see the result in real-time while interacting with the Tolerance and Softness boundary handles. If your resolution is too high and you are in the Modular Keyer, you can use the Crop feature to speed up processing time.

Clearing and Resetting the Matte

You can clear or reset the matte and the correction of any selective. Use the C/R button next to the Work On box to reset the correction but maintain the matte. Use the C/R button to the right of the View box to reset the matte but maintain the correction. Use C to clear and R to reset all related keyframes in the animation. If there is no matte or colour correction associated with the correction, the C/R button is disabled.

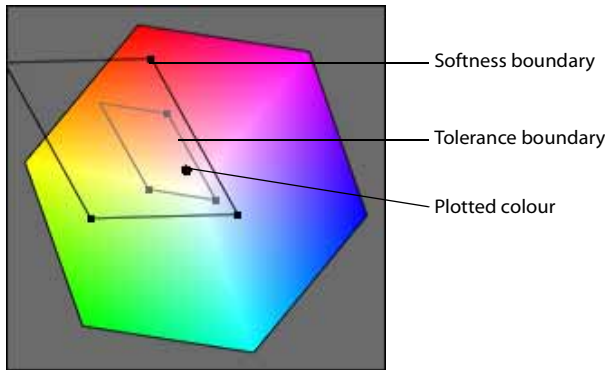
Using the Hue Cube

The hue cube shows the range using tolerance and softness indicators: the small boundary shows tolerance, and the larger boundary shows softness. These ranges can be adjusted to increase or decrease the range of colours affected. The T and S buttons display the tolerance and softness boundaries on the hue cube.

You can adjust softness and tolerance by moving the handles on the tolerance and softness boundary. Click and drag the points on either boundary area to move the handles.

HINT: To obtain a good level of softness, move the handle of the softness boundary all the way around (taking care not to move the handle too far) and look at the matte.

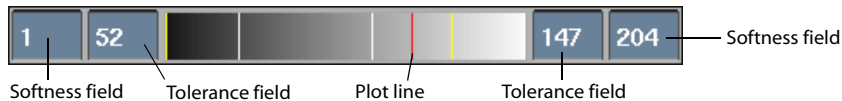
When you use Plot, a black dot appears on the hue cube representing the plotted colour. You can move the boundaries to include or exclude the plotted colour.



You can zoom the hue cube by changing the option from Move to Zoom and then dragging the cube. Press the **SPACEBAR** and drag the cube to pan the view. Set the option back to Move to move the lozenge handles again. Alternatively, press the **SPACEBAR** to move the hue cube and press **SHIFT** to zoom.

Use any of the Define controls to reset settings and move the lozenges to the appropriate area of the hue cube. For example, if you click Y, the lozenges move to the yellow area of the hue cube. Define specifies your start-up condition.

Using the Luma Range



When you use Plot, a red line appears in the luma range. You can then expand or contract the luma range to include (or exclude) the plotted colour. Once the range is defined, you can switch to view Result and apply colour correction from the Basics or Warp|Gam menu.

The tolerance appears as two white lines in the range. All the pixels in the key-in clip with the selected colour value are keyed out.

The yellow lines indicate the limits of the softness range. The pixels in the front clip with colour values at the centre of the range are black (transparent) in the matte. As you move away from the centre, the pixels become more opaque.

Use the numerical fields to set maximum and minimum softness and tolerance ranges.

Adjusting Shadows, Midtones, and Highlights

Use the Sha, Mid, and Hig buttons to adjust the entire image's shadows, midtones, and highlights. These three buttons expand the tolerance and softness boundaries to include all ranges of colours in the image.

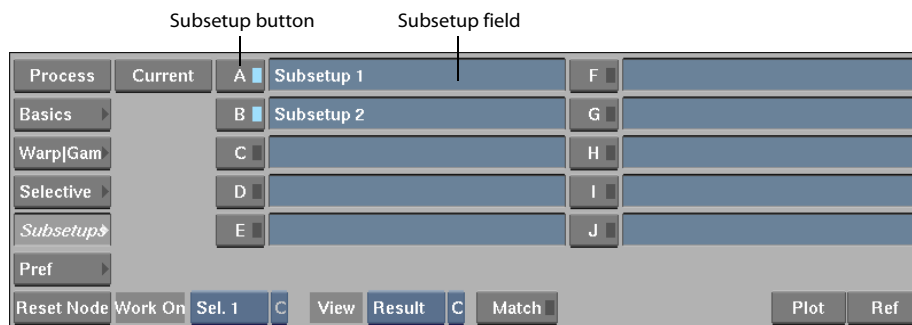
To desaturate highlights:

1. Select Sel1 from the Work On option box. Only the region defined for Selective 1 will be affected.
2. Click Hig to select highlights.
3. Use either the Luma field or the Saturation field and Highlight trackball in the Basics menu to desaturate the highlights.

NOTE: To adjust shadows, click Sha and use the Basics menu. To adjust midtones, click Mid and use the Basics menu. For more information on using the Basics menu, see "Basics Menu" on page 542.

Subsetups

Subsetups are Colour Warper setups selected at a specific moment at a specific frame. Subsetups are for the current frame only and do not include entire animations. You can select up to 10 different subsetups, and can compare colour corrections in other subsetups.



Use the Subsetups menu to:

- Selectively store intermediate setups.
- Compare any subsetup to the current setup.
- Share subsetups between Colour Warper nodes.

Storing Setups

To store the current setup for later retrieval, press and hold one of the subsetups buttons until a message appears indicating “Storing subsetup <subsetup letter>” and release. Use the setup at another frame or in another node by pressing the corresponding button. LED indicators on each subsetup button indicate active subsetups. If you have not stored anything into a subsetup, the active light of the button will be off.

NOTE: Subsetups are shared between multiple Colour Warper nodes.

Naming and Comparing Setups

You can name your subsetups. This allows you to store setups and later compare them to your current setup, or to apply the same correction to different points in time in a clip. Click in a subsetup field, type in a name or description, and press **ENTER** to store the setup.

Deleting and Replacing Setups

To discard the current setup, click Current. Click and hold to overwrite and replace subsetups. A keyframe is set and the current frame is saved.

Click Current to return to the settings you used before trying out various subsetups. To set the Current setup, perform an operation such as advancing a frame or changing a value in the Basics or Warp|Gam menu and your present settings will be displayed the next time you click Current.

NOTE: Subsetups cannot be saved and are deleted when you close **flame**. If you do a commit by doing an operation, you cannot undo it.

Hot Keys

Press:

In the Basics menu, Click+**Z**

In the Basics menu, Click+**X**

In the Basics menu, Click+**C**

O

R

CTRL+G

To:

Adjust Shadows (black).

Adjust Midtones.

Adjust Highlights (white).

Plot a colour.

Plot a colour for reference.

Enable Grab in the reference buffer.

[illegible]

28

Filters

Not just rose-coloured glasses

Use filters to add visual interest to your clips, such as sharpening or softening, saturating or desaturating, or to add texture and patterns. Use a filter as-is, or modify and customize it.

Summary

In this chapter, you learn about:

- “Accessing the Filter Menu” on page 555
- “The Filter Library” on page 557
- “Applying a Filter to a Clip” on page 556
- “Custom Filters” on page 558

About Filters

You can customize filters either by creating your own filter from scratch or by modifying an existing filter. Any filter that you create can be saved in the filter library and later loaded for use in another work session. You can also combine filters to produce complex effects such as pixel turbulence.

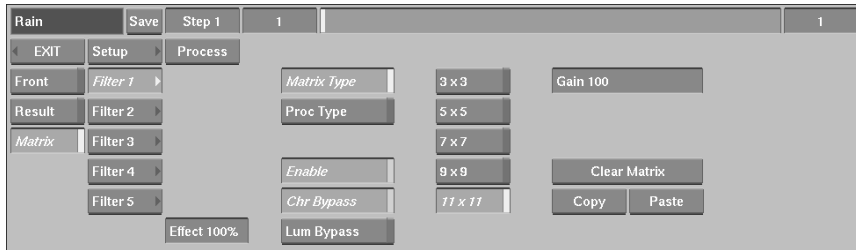
Accessing the Filter Menu

Use the Filter menu to work with filters.

To access the Filter menu:

1. Click the Filter button in the Processing menu.
2. Select a source clip.
3. Select a destination.

The Filter menu appears.

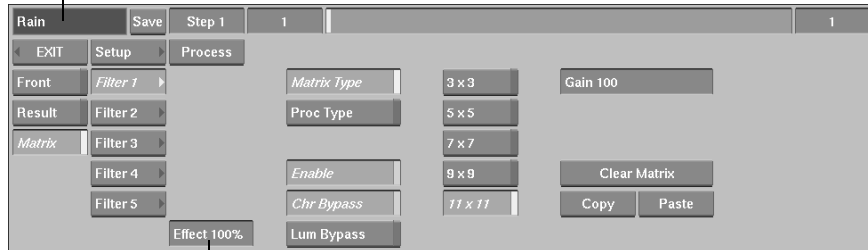


Applying a Filter to a Clip

To apply a filter to a clip, you select the filter from the filter library. You can control the level of filtering applied. Decreasing the value reduces the filtering effect, while increasing the value magnifies the filtering effect.

NOTE: When you use the Filter menu, the filter is applied across each frame in the source clip. To apply a filter to selected areas of an image, use the Paint module. For more information on using filters in Paint, see Chapter 51, "Paint: Using Filters and Special Effects Media."

Filter Title field



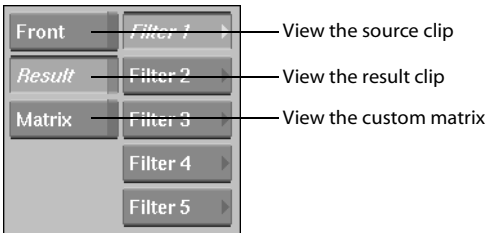
Effect field

To apply a filter to a clip:

1. In the Filter menu, click Setup to open the Filter Setup menu.
2. Click Load to open the filter library.
3. Select the filter that you want to use from the filter library. The Filter Setup menu returns. The name of the selected filter appears in the Filter Title field.
4. Click Filter 1 to return to the Filter menu.
5. Click Result to preview the effect of applying the filter to the source clip.
6. Adjust the value in the Effect field to get the level of filtering that you require.
7. Click Process to apply the filter to the clip.

Viewing Source and Result Clips

Use the clip view buttons to view either the source clip or the result clip. You can also display the matrix that you use when making custom filters.



The Filter Library

The filters supplied with **flame** are stored in the filter library. Additional filters you create can also be stored there. To open the filter library, click Load, Save, or Remove in the Filter Setup menu.

The following table describes some of the filters available in the filter library.

Use:	To:
Softening filters	Cause the image to appear less focused, by decreasing the contrast between adjacent pixels.
Sharpening filters	Provide light, medium, and heavy levels of image sharpening. These filters improve the clarity and focus of an image by increasing the contrast between adjacent pixels.
Outline filters	Outline areas where sudden colour changes occur in an image.
Saturate and Desaturate filters	Decrease or increase the amount of grey in the colour of an image. Saturation refers to the colour purity, or the amount of grey in the colour of the image. Less grey creates a more saturated image.
Scanner Correct filters	Remove undesirable effects encountered with scanned images.
Fragmentation and Reticulation filters	Displace pixels in the source image.
Negative filters	Produce a negative of the source image.

Custom Filters

Although the filter library contains a number of special effects filters, you may need custom filters for specialized applications.

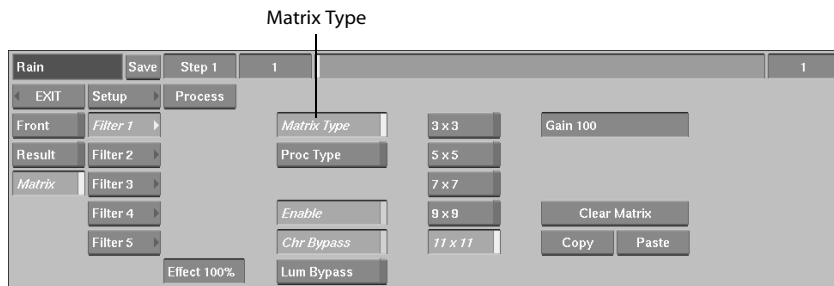
When a filter is applied to an image, the colour value of each pixel in the image is evaluated individually and is modified in one of two ways:

- A matrix-based spatial filter modifies each pixel according to calculations performed on its surrounding pixels.
- A procedural filter modifies each pixel in a predefined way.

A filter is composed of up to five matrix or procedural image operations. These operations are performed in sequence so that the output from one operation becomes the input to the next. When these operations are combined, they are called *filter elements* to distinguish them from the final composite filter. Matrix and procedural filter elements can be combined within a single filter.

Matrix Filter Elements

A matrix specifies the relative influence of each pixel's colour value in the filter calculation. Each matrix consists of a grid of fields. The field at the center of the grid represents the pixel that is currently being evaluated (the current pixel). The remaining fields represent the pixels surrounding the current pixel. The value that you set in each field determines the relative influence of the corresponding pixel in the filter calculations. A pixel with a higher value is more influential in the calculations than a pixel with a lower value.



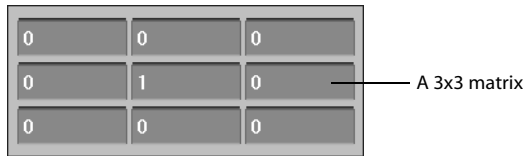
To use matrix elements in a custom filter:

1. You can use up to five filter elements in a single custom filter. Select the number of the current filter element by clicking Filter 1, Filter 2, Filter 3, Filter 4, or Filter 5.
2. To set up a matrix filter element, click the Matrix Type button.
3. Select the matrix dimensions.

The matrix dimension buttons are labelled 3x3, 5x5, 7x7, 9x9, and 11x11. The 3x3 button opens a matrix of three rows by three columns; the 5x5 button opens a matrix of five rows by five columns, and so on.

4. Click Matrix.

The Matrix grid appears.



5. Click Enable to activate the filter element in the final custom filter.
6. Set the values in the matrix fields for all the pixels you want to be considered. These fields are numeric. Permissible values range from -255 to +255 in 8-bit mode, and from -4095 to +4095 in 12-bit mode.
7. To bypass the chrominance (hue and saturation) channels or the luminance (brightness) channel of an image, enable the Chr Bypass or Lum Bypass button.
8. Use the Gain field to boost or reduce the colours in the image. Gain is expressed as a percentage value. Values less than 100% subdue the colours and values greater than 100% boost the colours.
9. Repeat the above steps for each matrix that you want to include in the custom filter.
10. Click Result to preview the result of applying the new filter to the source clip. Make changes to the filter elements as required.

To save your custom filter:

1. When you are satisfied with your custom filter, save the new filter in the filter library. Click Save As in the Setup menu. Type the name of the new filter, and click Enter.
2. Click Process in the Filter menu to apply the new filter to the clip.

Using Copy and Paste

Use the Copy and Paste commands in the Filter Matrix menu to copy a setup from one filter and apply it to a custom filter. The Copy command copies the displayed matrix as well as the status of the Chr Bypass and Lum Bypass buttons.

To copy and paste setups:

1. Load the filter you want to modify.
2. Display the matrix setup you want to copy and click Copy.
3. Select the filter element where you want to paste the matrix setup by clicking Filter 1, Filter 2, Filter 3, Filter 4, or Filter 5.
4. Click the Matrix Type button.
5. Click Matrix to display the matrix.

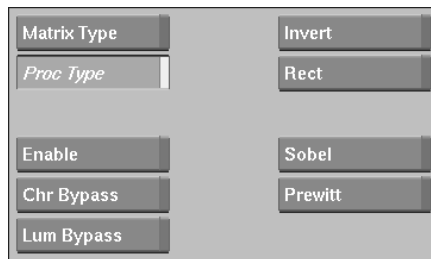
- Click Paste. The setup you copied is pasted into the current filter element.

Clearing the Matrix

Click Clear to reset all boxes in the displayed matrix to 0.

Procedural Filters

You can combine up to five matrix or procedural filter elements into a custom filter. Procedural filters use a predefined procedure instead of using matrix calculations to alter the image.



To use a procedural filter:

- Select the number of the current filter element by clicking Filter 1, Filter 2, Filter 3, Filter 4, or Filter 5.
- Click the Proc Type button.
The Procedural Filter menu appears.
- Select the procedural filter you want to use.

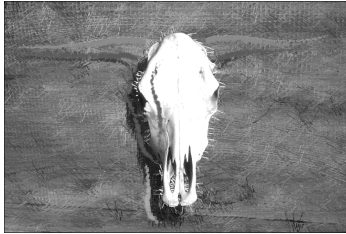
Select:	To:
Invert	Invert the image.
Rect	Blur the image. Use the Height and Width fields to adjust the direction and intensity of the blur. These fields appear when you select the Rect option.
Sobel	Use an edge-detection filter. Select the direction in which the filter is applied from the field that appears in the menu. The options are DiagRight (diagonal right), DiagLeft (diagonal left), and Horiz/Vert (horizontal/vertical).
Prewitt	Use an edge-detection filter. As with the Sobel filter, you can select the direction in which the filter is applied.

- Click Enable to activate the filter element in the final custom filter.
- To bypass the chrominance (hue and saturation) channels or the luminance (brightness) channel of an image, click Chr Bypass or Lum Bypass.
- Repeat the above steps for each procedural filter element that you want to include in the custom filter.

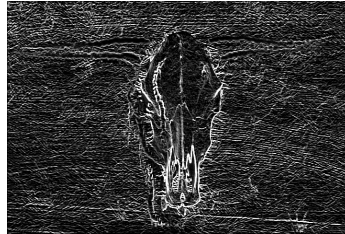
7. Click Result to preview the effect of the new filter on the source clip. Make changes to the filter element as required.

To save your custom filter:

1. When you are satisfied, save the new filter in the filter library. Click Save As in the Filter Setup menu. Type the name of the new filter, and click Enter.
2. Click Process in the Filter menu to apply the new filter to the clip.



The original image



After applying the Sobel procedural filter

This image shows a full page of white paper with horizontal dashed lines, resembling notebook paper. The lines are evenly spaced and run across the width of the page. There is no handwriting or other markings on the paper.

29

Batch Processing

One-shot deal

Use Batch to streamline workflow processes for colour correction, compositing, animation, and special effects.

Summary

In this chapter, you learn about:

- “Assembling a Process Tree” on page 565
- “Manipulating Nodes” on page 572
- “Previewing Results” on page 580
- “Processing Batch Jobs” on page 586
- “Setting Batch Options” on page 589

About Batch

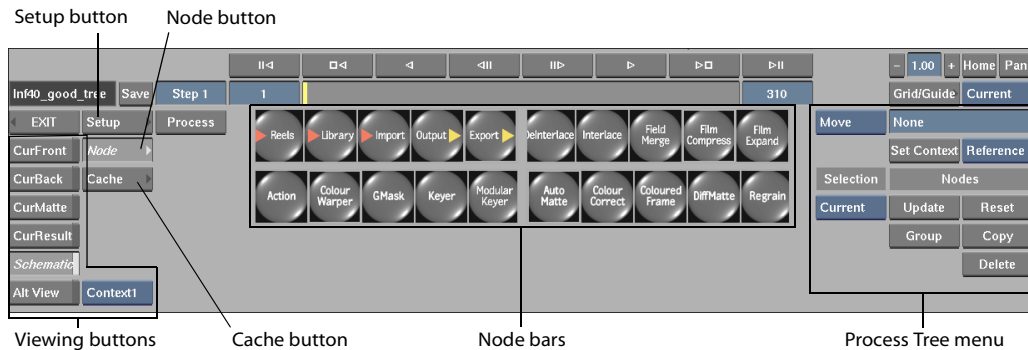
You use Batch to assemble a series of tasks such as colour correction, keying, multilayer composites with animation, and regraining where the result of each task serves as the source for the next one. As you create a complex effect, you can eliminate the amount of processing time required for intermediate steps by building a process tree in Batch. You process the clip only at the end, and any prerendered frames stored in the cache are not reprocessed.

To open Batch:

1. In the Processing menu, click Batch.
2. Select a destination reel on the desktop.

Alternatively, hold **ALT** while selecting a destination reel to open Batch with an empty schematic.

The Batch menu appears.



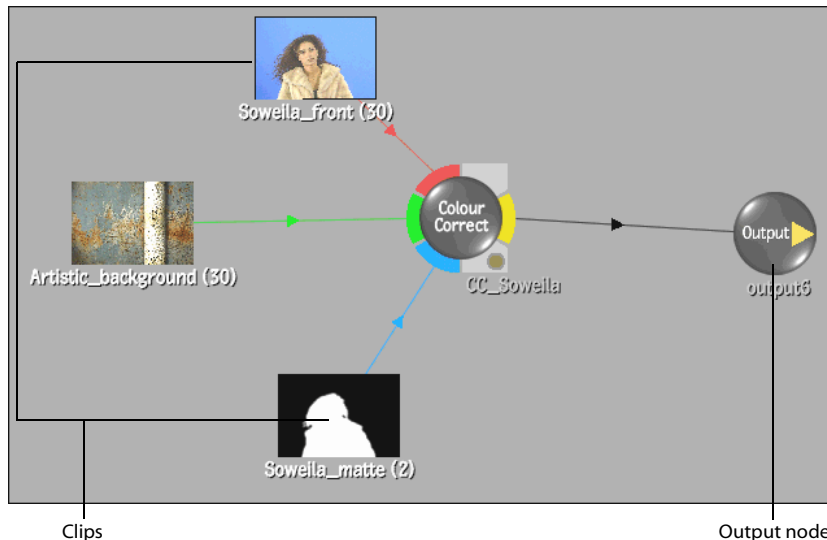
The Batch controls are described in the following table.

Select:	To:												
Setup button	Load and save Batch setups and set Batch options.												
Node button	View the menu for a selected node in the schematic. Each node has its own menu and depending on the node you select, you can access the full module by clicking Edit.												
Cache button	Manage the cache of any frames you render in Batch.												
Viewing buttons	Set the view in the image window for the selected node: <table> <tr> <td>CurFront</td><td>View the front of the currently selected node. This hot key is F1.</td></tr> <tr> <td>CurBack</td><td>View the back of the currently selected node. This hot key is F2.</td></tr> <tr> <td>CurMatte</td><td>View the matte of the currently selected node. This hot key is F3.</td></tr> <tr> <td>CurResult</td><td>View the result of the currently selected node. This hot key is F4.</td></tr> <tr> <td>Schematic</td><td>View the Batch schematic. This hot key is ~ and acts as a toggle between the schematic and the current view.</td></tr> <tr> <td>Alt View</td><td>View the node for which the context is set. Select either Context1 or Context2. The hot keys are 1 and 2, respectively</td></tr> </table>	CurFront	View the front of the currently selected node. This hot key is F1 .	CurBack	View the back of the currently selected node. This hot key is F2 .	CurMatte	View the matte of the currently selected node. This hot key is F3 .	CurResult	View the result of the currently selected node. This hot key is F4 .	Schematic	View the Batch schematic. This hot key is ~ and acts as a toggle between the schematic and the current view.	Alt View	View the node for which the context is set. Select either Context1 or Context2. The hot keys are 1 and 2 , respectively
CurFront	View the front of the currently selected node. This hot key is F1 .												
CurBack	View the back of the currently selected node. This hot key is F2 .												
CurMatte	View the matte of the currently selected node. This hot key is F3 .												
CurResult	View the result of the currently selected node. This hot key is F4 .												
Schematic	View the Batch schematic. This hot key is ~ and acts as a toggle between the schematic and the current view.												
Alt View	View the node for which the context is set. Select either Context1 or Context2. The hot keys are 1 and 2 , respectively												
Node bars	Place a node in the schematic. Drag a node to the schematic to build a process tree.												
Process tree menu	Manipulate and organize the process tree in the schematic. See "Manipulating Nodes" on page 572.												

Assembling a Process Tree

You use Batch to assemble a series of tasks using nodes. Each node represents a specific **flame** function. You connect clips with nodes and use the result of one node as the source for the next node. As you assemble this arrangement in the Batch schematic, you are building a *process tree* that will process as many output clips as you want as well as intermediate clips.

A process tree begins with a clip, contains at least one node, and ends with an Output or an Export node. For example, the following figure shows a simple process tree that performs a colour correction using a front, back, and a matte clip.



To build a process tree:

1. In the Batch menu, click Schematic.
2. Add the clips and nodes in the schematic. See “Adding Clips to the Schematic” on page 566.
3. Connect the clips and nodes in the required sequence for your effect.
See “Connecting Clips and Nodes” on page 568 and “Working with Multiple Connections” on page 570.
4. Connect a node to an Output or Export node.
Use an Output or Export node at the end of a process tree to render as many results or intermediate results as required. See “Completing the Process Tree” on page 572.
5. Preview your process tree, then click Process.
The final clip is stored on the destination reel. See “Previewing Results” on page 580 and “Processing Batch Jobs” on page 586.

Adding Clips to the Schematic

You can add clips to the Batch schematic in the following ways:

- Use the input nodes to load clips from the desktop reels or a clip library, or import a clip from a different format.
- Double-click on a node's source tab to load clips from the desktop reels.

When you load or import clips from remote clip libraries, Batch does not duplicate storage on the local framestore. This technique is called “on demand” import where the import process occurs one frame at a time as you jog the timeline in Batch. In this way, the framestore is not cluttered with duplicate information.

To add clips to the schematic:

1. In the Batch menu, click Schematic and drag one of the following nodes to Batch schematic:

Select:	To:
Reels Node	Add clips from the desktop reels. You can select up to six clips. Click Exit Clip Select to return to Batch. See “Reels Node” on page 612.
Library Node	Load clips from a local or remote clip library. Select the clips and click Load to return to Batch. See “Library Node” on page 604.
Import	Import images and clips using the Import Image module. See “Import Node” on page 602.

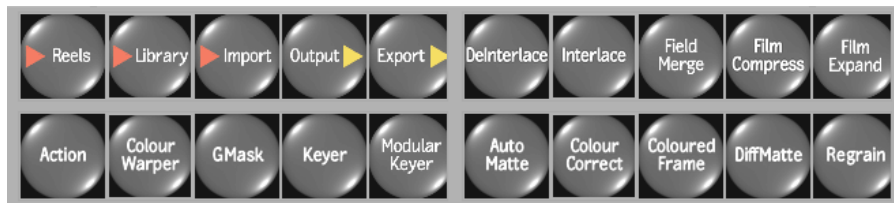
2. When you return to the Batch schematic, click a clip.
3. In the Batch menu, click Node.
The Slip field appears in the Batch menu. If the node bar appears, swipe the bar at the left or right of the Batch menu.
4. Change the value in the Slip field to slip the starting frame of the clip.
For example, a slip value of -15 holds the clip at the first frame and repeats it 15 times before the clip begins. A slip value of 10 begins the clip at frame 10.

Adding Nodes to the Schematic

You can add as many nodes as you need to create complex effects. Since Batch is designed to match your workflow, you assemble process trees in a similar order as you would by going from module to module.

To add a node to the schematic:

1. In the Batch menu, click Schematic.
2. Click Node and swipe the bar at the left or right of the menu to view the node bars.
3. From one of the four node bars, select a node.

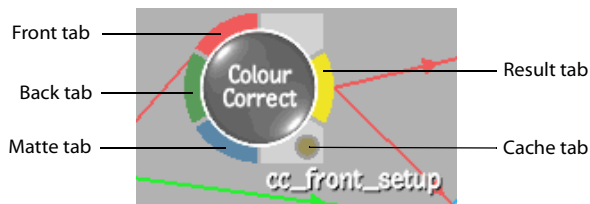


4. Drag in each node bar to scroll the nodes and select the node of your choice.
 5. Drag the node from the node bar and place it in the schematic.
To use the node in the process tree, you must connect it to other nodes or clips. See “Connecting Clips and Nodes” on page 568.
 6. In the schematic, click the node, and if the node bars appear, swipe the bar at the right or left side of the menu.
If the selected node is a command such as Logic Ops, its menu appears in Batch. If the selected node is an existing module in **flame**, an Edit button appears in the Batch menu.
 7. Click Edit to access the full module for the selected node.
For example, when you select a Keyer node in the schematic and you want to pull a key, click Edit to access the full Keyer module.
 8. In the module you accessed through Batch, click Save to return to Batch with the latest unprocessed setup for the selected node.
You repeat this procedure for all the nodes you set up in your process tree.
- For a description of each Batch node, see “Batch: Node Reference” on page 593.

Connecting Clips and Nodes

As you add nodes, you can connect them either by linking the result from one node and using it as a source (front, matte, or back) for the next node in the process tree or by linking backward from the source of a node to the output of another. The only exceptions are the Output and Export nodes, from which you cannot link from their result.

To connect nodes, you use the coloured tabs on the node's left side; these tabs are called *source tabs*. The colours of the source tabs correspond with the cursor colours when selecting clips from the desktop reels. The yellow tab on the node's right side is called the *result tab*. You use the result tab of a node to connect its result to the front, back, or matte tab of another node.



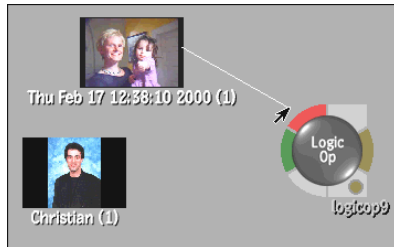
Node source tabs are described in the following table.

Select:	Colour:	To:
Front	Red	Connect a front clip to a node.
Back	Green	Connect a back clip to a node.
Matte	Blue	Connect a matte clip to a node.
Result	Yellow	Connect the result of a node to other nodes.
Cache	Grey and yellow circle	Enable, disable, or set cache as read only for the selected node. "Working with Cache" on page 584.

The available source tabs depend on the node. If the node accepts a front, back, and matte clip, all coloured source tabs are available. If the node only accepts a front clip, the red source tab is available and the other source tabs are grey. The result tab is always available. When a source tab or result tab is not connected to a clip or to another node, the coloured tabs are dimmed. See Chapter 30, "Batch: Node Reference."

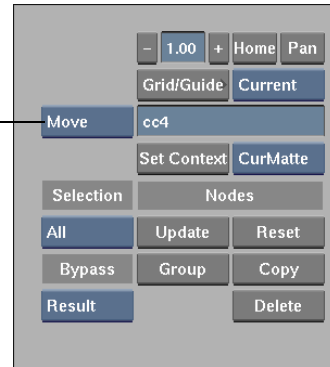
To connect a clip to a node:

1. Click the clip and drag the cursor to a source tab. Alternatively, select Parent from the Edit Mode box and drag the cursor to a source tab.



Example of dragging from a clip to the front tab of a Logic Op node.

Edit
Mode
box



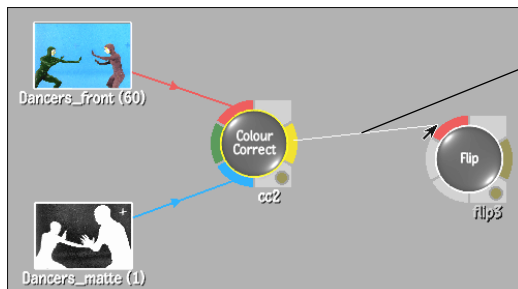
An arrowed line is drawn from the clip to the source tab. The colour of the line corresponds to the colour of the source tab to which the clip is connected.

For example, if you want a clip to be used as the front clip for the Colour Corrector node, a red arrowed line is drawn in the schematic connecting the clip and the node.

To connect two nodes:

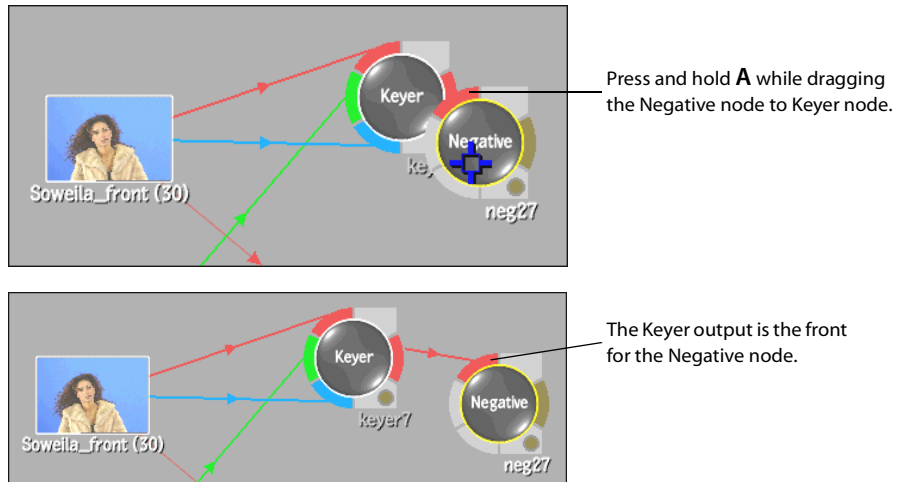
1. Click the Result tab of a node whose result you want to use and drag the cursor to one of the source tabs of the next node in the process tree.

An arrowed line is drawn from the node to the source tab. For example, click the result tab of the Colour Correct node and drag the cursor to the front tab of the Flip node to flip the result of the colour-corrected clip.



Drag from the Colour Correct
result tab to the Flip front tab.

Alternatively, hold **A** and drag a node to another node so their tabs touch.



Or, as another alternative, select Parent from the Edit Mode box and drag the cursor from the result tab to a source tab.

To disconnect nodes or clips:

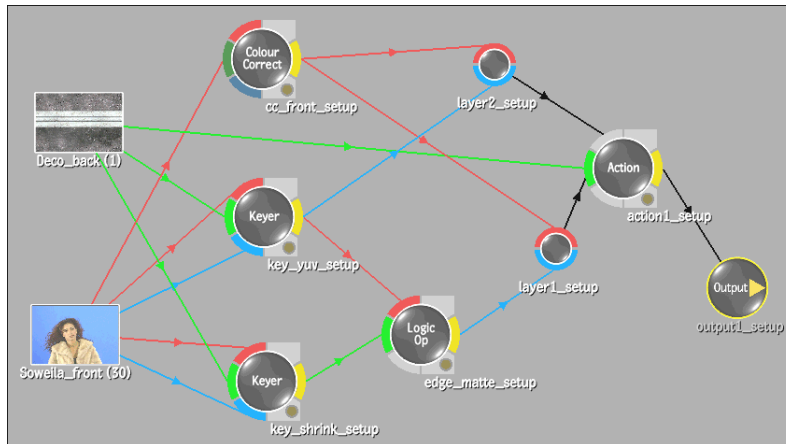
1. Drag the cursor across the connecting line between a clip and a node or between two nodes. The arrowed line is cut and the source tab is dimmed.
2. You can cut multiple connections in a single stroke by dragging over several connecting lines in the schematic.

Working with Multiple Connections

When connecting clips and results, you can have single or multiple connections. This means that you can connect a clip to many different nodes. You can also connect a node's result tab to more than one node.

For example, the Soweila_front clip is connected to the front tab of a Colour Correct node and two Keyer nodes. The Deco_back clip serves as the back for both Keyer nodes and the Action node. The result of the Colour Correct node is connected to two Action layers nodes, one of

which has a Logic Op node result feeding into it. Finally, the entire process tree feeds into one final output.



To add a connected clip:

1. Double-click a coloured source tab.

The desktop appears. The cursor is the same colour as the first unassigned tab.

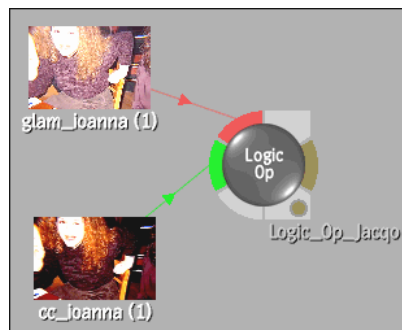
2. Select a clip.

The arrow changes colour if the node has other unlinked source tabs.

3. Continue to select clips for unlinked tabs or click Exit Clip Select.

You are returned to Batch where the selected clips are added to the process tree and connected to the node.

For example, the node on the left shows a Logic Op node before double-clicking a source tab. The process tree on the right shows the same Logic Op node after double-clicking the red source tab and selecting a front clip and a back clip from the desktop.



Completing the Process Tree

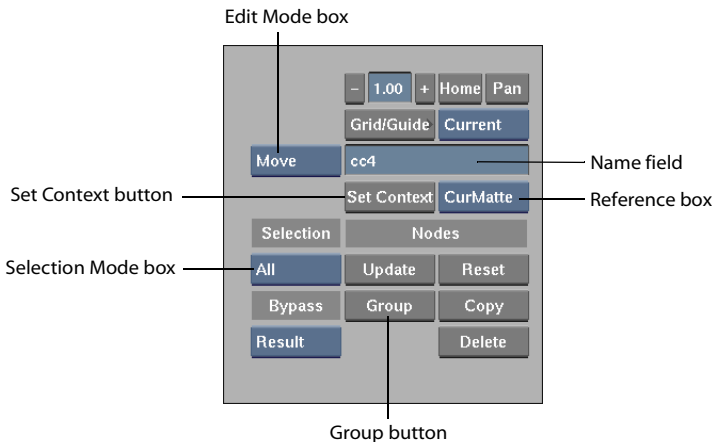
Before you process, a process tree must end with either an Output or an Export node. Batch uses these nodes to process the result of the process tree to the specified destination in the Queue Manager.

See “Output Node” on page 611, “Export Node” on page 599, and “Using the Queue Manager” on page 586.

Manipulating Nodes

As you build your process tree, you should reposition the nodes to keep track of the processing direction. For example, you may want to set up the tree so that the processing direction is from the top down, with the Output or Export node at the bottom. Or, you may want to process from left to right, with the Output or Export node at the right.

You will also want to give nodes more descriptive names, copy process trees, and delete unnecessary nodes. You must select a node before moving, copying, or deleting it. Batch lets you select a single node, a branch with several nodes, an entire process tree, or all the nodes. The controls for carrying out these tasks are contained in the Process tree menu.



The Process tree controls are described in the following table.

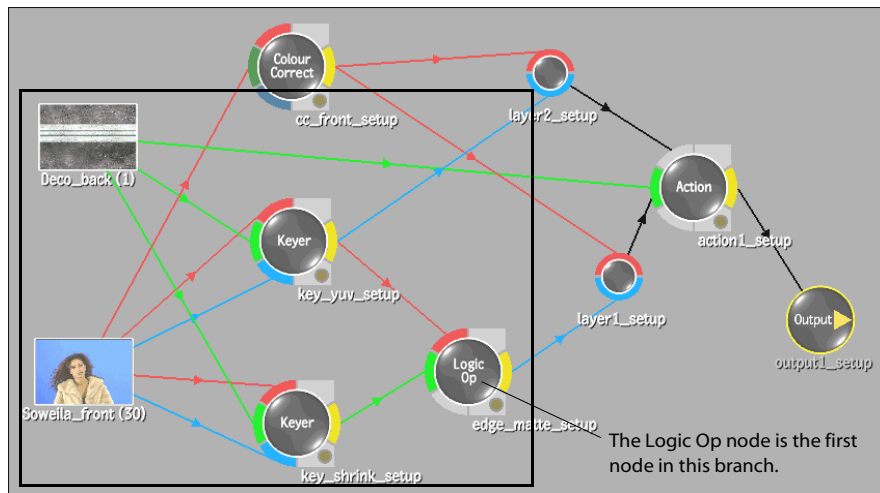
Select:	To:
Edit Mode box	Set the manipulation mode in the schematic.
	Move Move nodes, branches, and process trees in the schematic. In this mode, you can also parent and cut connections gesturally in the schematic.
	Parent Connect and disconnect clips and nodes.
	Delete Delete clips and nodes in the schematic, one at a time.

Select:	To:
Name field	Assign a unique and descriptive name to nodes and clips. Any clip names you modify here are also modified on the desktop reels and in the clip library where the clip originated.
Set Context button	Use the currently selected node as a context for another node. See “Viewing Clips In Context” on page 582.
Reference box	For the currently selected node, display one of the following in the reference portion, “R” side, of the split bar: <div> <div>CurFront</div> <div>View the front for the currently selected node.</div> </div> <div> <div>CurBack</div> <div>View the back for the currently selected node.</div> </div> <div> <div>CurMatte</div> <div>View the matte for the currently selected node.</div> </div> <div> <div>CurResult</div> <div>View the result of the currently selected node.</div> </div> <div> <div>Context1</div> <div>View the result your changes to a node are having on a node set as context1 further along in the process tree. See “Viewing Clips In Context” on page 582.</div> </div> <div> <div>Context2</div> <div>View the result your changes to a node are having on a node set as context2 somewhere else in the process tree.</div> </div> <div> <div>Reference</div> <div>View the reference frame in the Reference Buffer.</div> </div> In the Setup menu, select Split On. The other side of the split bar is determined by the view you select on the left side of the Batch menu (Viewing buttons). See “Setting Batch Options” on page 589 and “The Reference Buffer” on page 81.
Selection Mode box	Determine the node or set of nodes that you want to select: Current, Branch, Graph, or All.
Update button	Force an update of nodes that might not have processed the current frame.
Group button	Create a group with sets of nodes, branches, or process trees. See “Grouping Nodes” on page 575.
Reset button	Reset a node, a branch of nodes, or an entire process tree according to what is selected in the Selection Mode box. The options for each selected node are reset to their default values after you click Confirm. However, if one the selected nodes is linked to a module such as Action or the Keyer, the options and settings for these modules are not reset. To reset a module, you must access the module and use its reset options exclusively.
Copy button	Copy selected items in the schematic.
Delete button	Delete selected items in the schematic.

To select nodes:

1. From the Edit Mode box, select Move.
2. In the Selection Mode box, select a mode. Alternatively, use the hot keys.

Select:	To:
Current	Select one item in the schematic such as a clip or a node.
Branch	Select the nodes and clips contained in a branch. A branch is defined by a node and all the clips and nodes before it in the process tree.



Graph	Select an entire process tree.
All	Select all process trees in schematic, in other words, the entire schematic.

3. In the schematic, make a selection.

To select:	Click:
An item	The node or image.
A branch	The first node in the branch you want to select. All nodes before it (upstream) in the tree are selected.
A process tree	Any node in the process tree.
All process trees	Any node in any process tree.

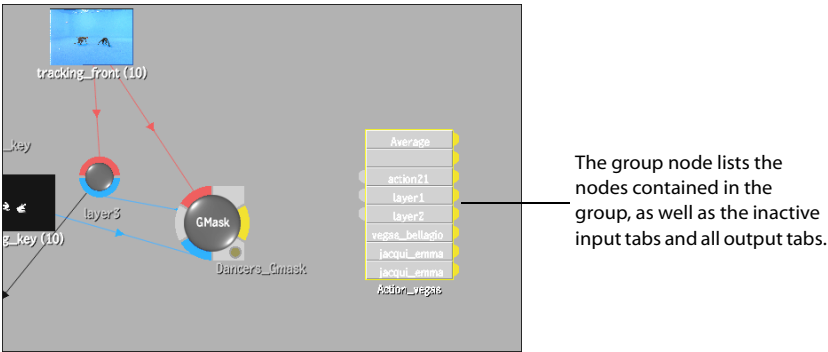
The selected nodes are highlighted. To differentiate between the clicked node and other nodes in the selection, the clicked node is outlined in yellow and the other selected nodes in white.

Grouping Nodes

You can group nodes to simplify a cluttered schematic. When you group nodes, they collapse into a group node, taking up less space in the schematic. You can create several groups and organize them in your schematic. You can then work on individual groups. For example, you can group a keying and colour correction branch separately from an Action and filtering branch, and then work on each branch independently of the other.

To group nodes:

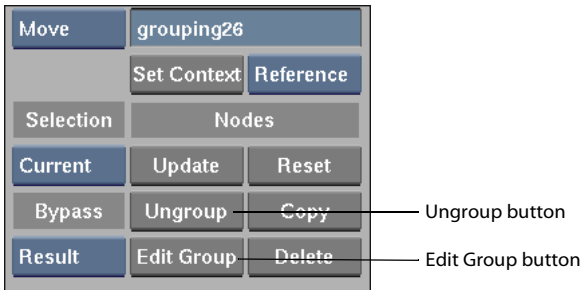
1. Select the nodes you want to group by pressing **CTRL** and dragging to select multiple nodes.
Selected nodes are outlined in white.
2. In the Process tree menu, click Group.
The selected nodes collapse into a group node.



3. In the Name field, type a name for the group and press **ENTER**.
The new group name is shown in the schematic.

To edit a group:

1. Select the group you want to edit.
When you select a group node, the Process tree menu changes to show Ungroup and Edit group buttons.



2. Click Edit Group.

The nodes contained in the group appear in an isolated schematic and the Process tree menu changes to show the Exit Group button.

3. Modify the nodes as required.

4. In the Process tree menu, click Exit Group to return to the schematic.

The group node reappears in the schematic and the Process tree menu changes back to show the Ungroup and Edit Group buttons.

5. Click Ungroup to expand the group of nodes to their pre-group positions in the schematic.

Moving Nodes

You can move a node, a branch, group, or an entire process tree. Typically, you move a node by clicking it and dragging it in the schematic. You can move several nodes at a time.

To move nodes:

1. In the Edit Mode box, select Move.
In the schematic, the cursor changes to a blue crosshair.
2. In the Selection Mode box, select the nodes and clips you want to move.
3. Press **ALT** and click a node to move all selected nodes.

Copying Nodes

You can copy a node, a branch, or an entire process tree. You can also copy clips.

To copy a selection:

1. In the Edit Mode box, select Move.
2. In the Selection Mode box, select the nodes and clips you want to copy.
3. In the schematic, make your selection.
4. Click Copy to copy the selection.
A copy of the selection appears in the schematic.

Deleting Nodes

You can delete nodes or clips individually; or you can delete branches, process trees, as well as the entire schematic.

To delete a single node or clip:

1. In the Edit Mode box, select Delete.
In the schematic, the cursor changes to a red crosshair.
2. Click the node or clip you want to delete and when prompted, click Confirm. Alternatively, press **ALT** and click the node or clip to override the delete confirmation prompt.
The selected item is deleted.

NOTE: In Delete mode, you can only delete one item at a time.

To delete a branch:

1. In the Edit Mode box, select Move.
2. In the Selection Mode box, select Branch.
3. In the schematic, click the first node for the branch you want to delete.
All nodes and clips feeding into the clicked node are outlined in white and the leading node in the branch is outlined in yellow.
4. Click Delete and when prompted, click Confirm. Alternatively, press **ALT** and Delete to override the delete confirmation prompt.
The selected branch is removed from the schematic.

To delete a process tree or an entire schematic:

1. In the Edit Mode box, select Move.
2. In the Selection Mode box, select Graph or All depending on what you want to delete.
3. In the schematic, click a node in the process tree you want to delete or if you want to delete the entire schematic, click any node.
4. Click Delete and when prompted, click Confirm. Alternatively, press **ALT** and click Delete to override the delete confirmation prompt.
The selected process tree or schematic is deleted.

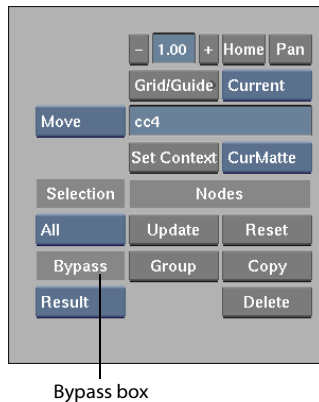
To delete by dragging items out of the schematic:

1. In the Edit Mode box, select Move.
2. Select an option from the Selection Mode box.
3. In the schematic, click a node and drag to the bottom of the schematic.
The cursor changes to a green recycling icon.
4. When prompted, click Confirm. Alternatively, press **ALT+CTRL** and drag to override the delete confirmation prompt.

Bypassing Nodes

You can improve your workflow in Batch by deactivating certain nodes and rendering only the nodes that you want to process right away.

For any node, you can pass either the front, back, matte, or key-in clip—depending on the type of node you select and its source tab inputs—as the input to the next node in a branch. The bypass controls are the Result button and Bypass box in the Batch menu.



To bypass a node:

1. In the schematic, select the node that you want to bypass.

Typically, the bypass node will be feeding its result into a node that you want to render or work with more intensively, such as an Action node. For example, click a Keyer node that has front, back, key-in clips feeding into it.

2. In the Bypass box, select the clip you want to pass as the input to the next node.

The options that appear in the Bypass box depend on the inputs of the node you select in step 1. For example, the Keyer node has Front, Back, and Key in clips as inputs to its source tabs, therefore they appear in the Bypass box.

Previewing Results

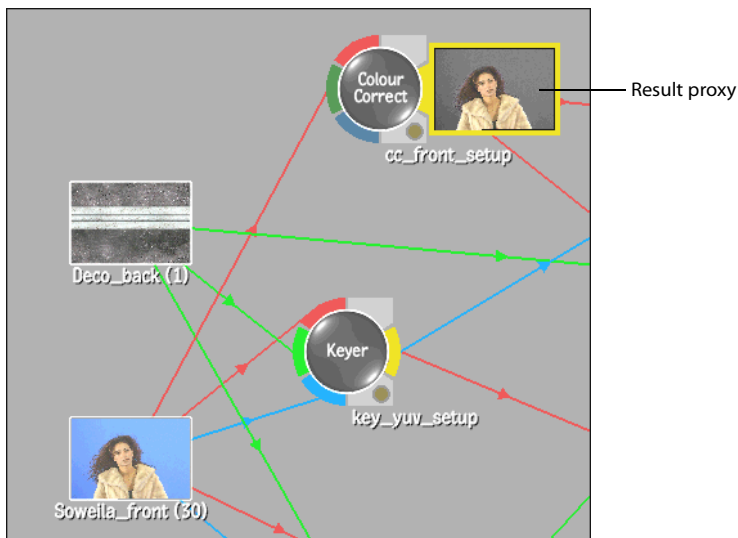
While creating your process tree, you can preview results at any time. If you do not like the result of one operation, you can modify or delete it without affecting the other operations in the sequence. You can preview your results in Batch by:

- Using proxies in the schematic
- Clicking one of the viewing buttons to view results in the image window
- Setting a context node in schematic view and viewing the results in context in the image window

To preview results using proxies:

1. In the Setup menu, enable Auto Update.
2. In the schematic, select the node whose result you want to preview.
3. Double-click the result tab. Alternatively, press **T** as you click the result tab.

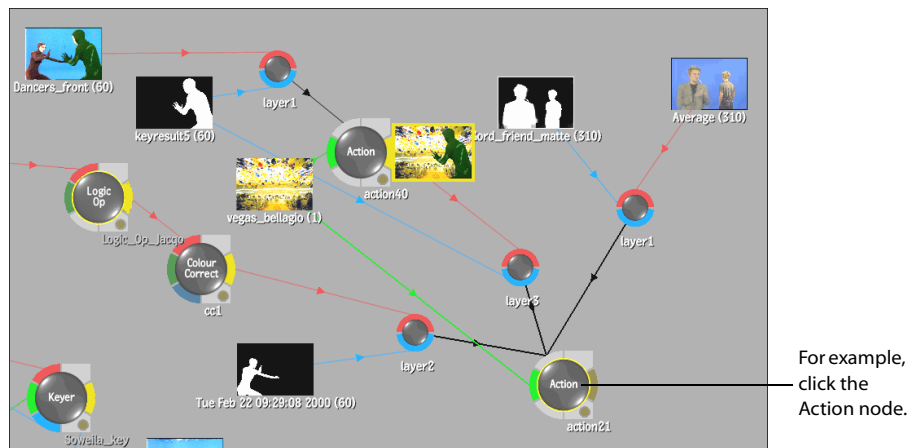
The result proxy appears and shows the result up to and including the selected node.



4. To close the proxy, double-click the result tab again, or press **T** as you click the result tab.

To preview results in the image window:

1. Select the node whose results you want to preview.



2. In the Batch menu, click CurResult.

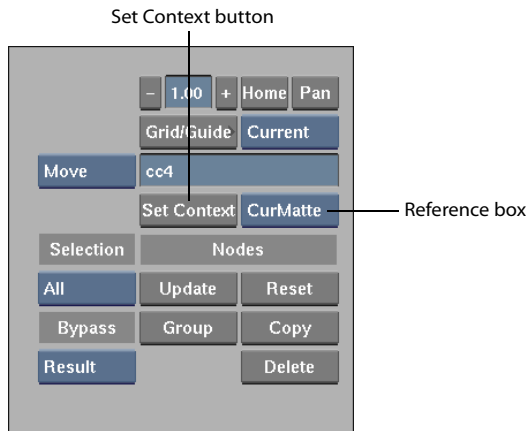
The intermediate nodes are processed up to and including the selected node and the result of the selected node is displayed in the image window.



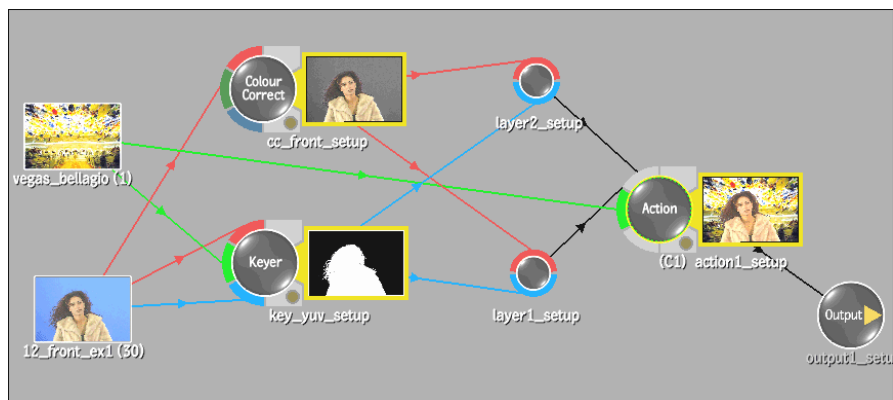
Viewing Clips In Context

You can view a node in context with another node to compare intermediate results throughout the process tree. By working with context points, you can modify nodes in the process tree and immediately view the impact those changes have on the nodes further along in the process tree.

You can set two different context points in a process tree. The Process tree menu includes the Set Context button and the Reference box for setting the context of a node operation.



The following example has a process tree that includes a Colour Correct node that feeds into an Action node. Two Keyer nodes also feed into the same Action node. You can set the context on the Action node and continue modifying the Colour Correct node, while viewing the result of the Colour Correct node, the Keyer node, or the Action node.

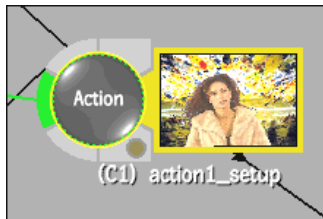


To view nodes in context:

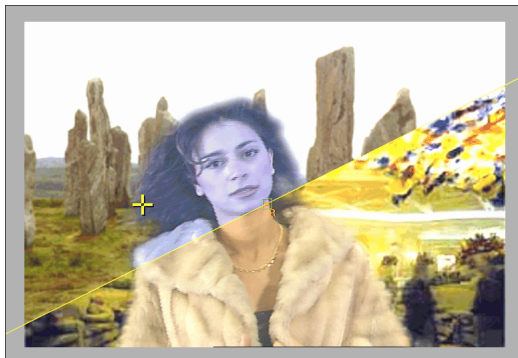
1. In the schematic, select the node you want as the context. Using the preceding example, select the Action node.
2. In the Process tree menu, click Set Context.

NOTE: You can also set the Context node by holding the = key and clicking the node you want to use. This method retains the current node selection.

(C1) appears next to the Action node name in the schematic.



3. In the schematic, select the node you want to edit. Using the preceding example, select the Colour Correct node and modify the setup.
4. View your colour correction changes in the context of Action by:
 - Selecting Context 1 from the Reference box.
 - Clicking Alt View and selecting Context1 or pressing **F4**.



In this way, you are colour correcting “in context” of the intermediate result. Furthermore, as you move to different frames in the module, Context 1 also displays the result at the same frame. For instance, if in the Colour Corrector you are at frame 15, Context 1 (Action) is also at frame 15. (Exceptions to this occur when using time-based nodes such as Film Compress, Film Expand, Interlace, and Deinterlace.)

- If you want to set a second context, select another node in the schematic—such as the Keyer node—and click Set Context.
(C2) appears next to the Keyer node name.
- Repeat steps 3 and 4.

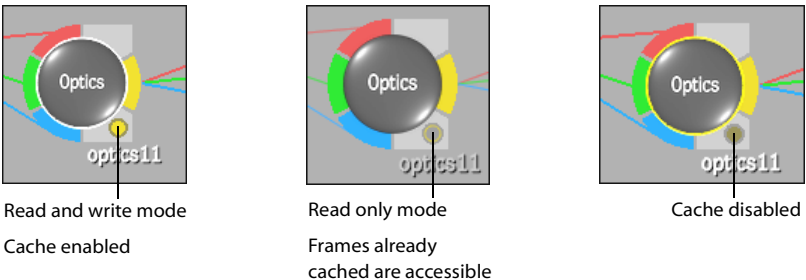
Working with Cache

In Batch, you can avoid reprocessing the same nodes as you navigate in the timeline by specifying which nodes should be cached. Cached frames are stored on the system’s framestore, allowing you to work more efficiently.

The Cache menu displays the number of rendered frames contained in the cache for the selected node. You can also view the total space that is available on the framestore, preventing you from using up all the disk space.

When you load a Batch setup, the nodes in the setup retain the cached frames’ location. Cached frames are stored with the clip ID until you remove them. You enable and disable node caching for all nodes in the Cache menu.

Each node has a cache tab on the lower-right corner of the node.

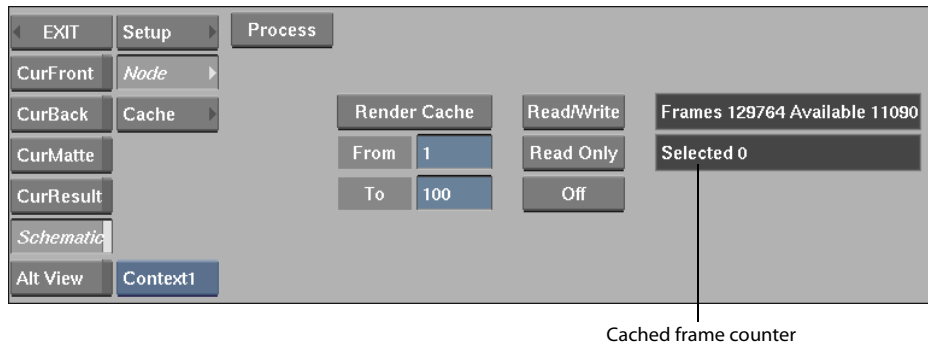


Click the cache tab to activate any of the following states.

State	Colour	Description
Off	Grey	Disable the cache. You can also press ALT-CLICK to disable and remove cached frames for the selected node.
Read and write	Yellow	Save and load data to and from the cache.
Read only	Amber	The indicator is shown with a yellow outline. No data can be saved to the cache, but rendered frames remain stored. You might use this state to prevent framestore from getting full.

Changing or cutting an input or any node further along the process tree that has cached frame clears the cache buffer. The Cache menu shows how many frames are cached for the selected

node, the number of frames available on the framestore, and allows you to clear the cache of the selected node or all cached frames.



To cache frames:

1. In the Batch processing tree, select a node and click the cache indicator.
The messages bar displays “Batch: Node cache set to read-write,” and the cache indicator changes to bright yellow.
2. In the Cache menu, specify the frames you want to render and store in cache using the From and To fields and click Render Cache.

The cached frames are stored in the framestore and added to the cached frame counter.

HINT: You do not have to follow step 2 to actually cache frames. Typically, you use *intelligent* caching, which automatically caches frames as you move through the timeline. Caching occurs as the frame is displayed and the node operation is applied. A cached node automatically saves frames as you display—or visit—them and then reloads them each time you visit the frame again. Your workflow speed is increased.

Stored cache clips are contained in a `_Cache_` library and correspond to the clip IDs in the setups. These cache library files are named according to setup name and project name.

3. To turn off the cache and remove cached frames, **ALT-CLICK** the cache indicator for the selected node.

Processing Batch Jobs

As you build process trees in the schematic, each Output or Export node is added to the Queue Manager. Once in the Queue Manager, you can consider each output or export process as a job that has various instructions for how the final output is rendered. Each job can be active or inactive, has a unique name, specific start and end frames, and a specific destination. You can execute jobs immediately or at a later time using a Batch script, as well as set job priorities. The list of jobs that appear in the Queue Manager are saved in the Batch setup file.

When you are ready to generate the final clip or an intermediate result, click the Process button. All process trees in the schematic are processed, and one clip is written to the destination specified by each Output node or Export node. The final output clips bear the name you specify in the Node name field in the Queue Manager.

Using the Queue Manager

With the Queue Manager, you can manage several output and export processes at a time. You can set priorities for particular output and export processes, specify the destination for processed output, attach a script, as well as determine the frames at which to start and end an export or output process. Furthermore, you can attach a script that contains elaborate instructions such as converting processed images to a movie file (.mpg) and sending notification messages to alert the status of various Batch processes.

In Batch, the Queue Manager is accessed through the Queue menu. To use the Queue Manager, you must add at least one Export or Output node to the Batch schematic.

P	Node Name	Start	End	Dst	Priority	Script
▶	callanish	1	30	Reels	1	None
▶	exportfile28	1	310	Export	2	None
▶	vegas_bellagio_gordo	1	310	Export	1	None
▶	ballybunion	1	310	Reels	2	None

The Queue Manager consists of several columns and rows, and each row contains details about a Batch job. You set these fields as described in the following table.

Select:	To:
Process indicator	<p>Activate the Output or Export node for processing.</p> <p>The arrow in the P column is bright yellow when a job is active. When you click Process, all active jobs are processed.</p>
Node Name	Provide a descriptive name for the job represented by an Output or Export node.
Start and End Frames	<p>Specify the range of frames you want rendered for the job and in the processed result.</p> <p>This means that 1 Output node could be set to process first 30 frames of a 60-frame clip and another Output node could process the remaining 30 frames.</p> <p>The Start and End fields override the total number of frames in the timeline. For example, if the timeline shows a 60-frame clip and an Output node is set to start at frame 20 and end at frame 100, then 80 frames are processed.</p>
Dst	Specify the destination for the output or exported result. For Output nodes, the destination is either the desktop reel or clip library.
Priority	Specify which jobs you want processed in an absolute order.
Script	<p>Attach a script file whose content is executed when the Output or Export node is finished rendering. Scripts must have the file extension <i>.bscript</i>.</p> <p>Click this field to load an existing script from the file browser.</p>

To access the Queue Manager:

1. In the schematic, click an Output node or an Export node.
2. In the Batch menu, click Node to display the menu for the selected node. If necessary, swipe the bar at the left or right side of the menu.

The Queue Manager appears with the corresponding Output node or Export node controls. Once an Output or Export node appears in the Queue Manager, it is considered a job.
3. In the Node Name field, provide a name for the job.

Using Scripts

Batch scripts are ASCII files that contain instructions related to a Batch job; and they must have the file extension *.bscript* for Batch to recognize them. A Batch script is executed after the Output or Export node to which the script is associated is finished rendering.

A few sample Batch scripts are furnished with **flame** and located in the directory */usr/discreet/example_setups*. You should consult with your UNIX system administrator to create scripts. Scripts are created using either C shell, Korn shell, or Perl. Also, Batch scripts must have the

proper file access, directory access, and execution permissions for the files, directories, and applications contained in the script.

When the script is executed, **flame** continues with its own processes and does not wait for the script to terminate. **flame** imposes no restrictions with executing scripts; however, be sure that no one else is using the system when a script is executed and be sure the script is approved by your UNIX system administrator.

Script variables that you can set are defined in the following table.

Variable	Description
BATCH_NODE_NAME	Corresponds to the name of the Output node or the Export node. This is the same name that you provide in the Queue Manager Node Name field.
BATCH_FIRST_FRAME	Corresponds to the first frame displayed in the Queue Manager for the selected Node Name.
BATCH_LAST_FRAME	Corresponds to the last frame displayed in the Queue Manager for the selected Node Name.
BATCH_ASPECT_RATIO	Applies only to scripts associated with Export nodes. Corresponds to the framestore aspect ratio.
BATCH_FRAME_RATE	This is the frame rate of the final clip result.
BATCH_FRAME_WIDTH	Applies only to scripts associated with Export nodes. Corresponds to the width of the final clip result.
BATCH_FRAME_HEIGHT	Applies only to scripts associated with Export nodes. Corresponds to the height of the final clip result.
BATCH_OUTPUT_DST	Applies only to scripts associated with Output nodes. Indicates the destination of the final clip result as either the desktop reel or a clip library.
BATCH_BIT_DEPTH	Applies only to scripts associated with Export nodes. Specify the bit depth of the final clip result. See “Bit Depth” on page 342.
BATCH_CROP_WIDTH	Applies only to scripts associated with Export nodes. Specify the crop width of the final clip result.
BATCH_CROP_HEIGHT	Applies only to scripts associated with Export nodes. Specify the crop height of the final clip result.
BATCH_FILE_FORMAT	Applies only to scripts associated with Export nodes. Indicates the numbering format of a sequence of exported images in the final clip result. This numbering sequence ensures that the order of frames in the clip is intact when you reimport the clip. See “Changing the Numbering Format” on page 342.

Variable	Description
BATCH_FILE_EXT	Applies only to scripts associated with Export nodes. Specifies the format to which the final clip result is being exported: Alias (.als), Cineon, Dpx(Spirit), Jpeg (.jpg), Pict (.pict), Pixar (.picio), Sgi (.sgi), SoftImage (.pic), Targa (.tga), Tdi/Maya (.iff), Tiff (.tif), or Wavefront (.rla).
BATCH_FILE_DIR	Applies only to scripts associated with Export nodes. Indicates the file location of the final clip result.

Executing Batch from the Command Prompt

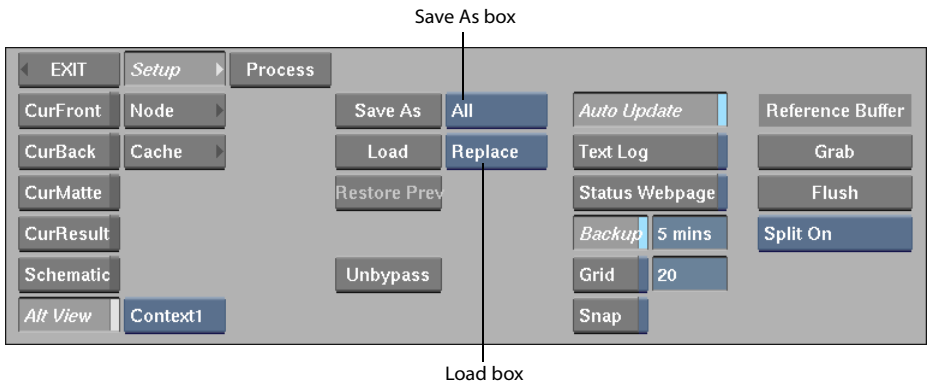
You can execute Batch directly from the UNIX shell so that you can render complex process trees while the machine is not in use, such as overnight. The following command starts **flame** for a specified user, and renders a specified Batch setup:

```
inferno -b batchsetup -U user - J project
```

NOTE: The specified user and project name must be valid.

Setting Batch Options

You use the Batch Setup menu to load and save Batch setups, control backups, enable Batch log files, and display a grid in the schematic.



The Batch Setup controls are described in the following table.

Select:

Save As button
and box

To:

Save a Batch setup by selecting one of the following:

- All Save all nodes in the schematic
- Selected Save selected nodes

By default, Batch setups have the file extension *.batch*.

Select:

Load button
and box

To:

Load a Batch setup by selecting one of the following:

- | | |
|---------|--|
| Replace | Replace the current Batch setup in the schematic. |
| Append | Append the selected setup to the current Batch setup in the schematic. |

Loading a Batch setup also loads the clips used in the processing trees. If Batch cannot find a clip, it displays the name of the clip in red. If you choose Append and the setup contains nodes that have the same name as the current Batch setup, the conflicting nodes in the current schematic are renamed.

You can load Batch setups that contain **flame** and **inferno** specific nodes. These nodes—called render-only nodes or noneditable nodes—cannot be modified in **flame**, however, Batch processes the node operation in the final clip result.

Restore Prev button

Revert to the last saved setup. All changes made since the previous Save operation are undone.

Unbypass button

Disable node bypassing for selected nodes in the current Batch setup.

Auto Update button

Update a node's result proxy automatically. When you change the current frame, the result proxies are updated for all nodes.

Text Log button

Save Batch processing status information in a text file called *batch_log*. These files are located in */usr/discreet/project/effects/[project_name]/batch/log*.

Information is appended on a continuous basis to this file and serves as a permanent log file as long as Text Log is enabled.

Status Webpage
button

Create three HTML files called *batch_log.html*, *batch_log_status.html*, and *index.html* when this button is enabled.

These files are located in */usr/discreet/html*.

The file *batch_log.html* describes what user, project, and date of the current job.

The file *batch_log_status.html* is the actual status of the Output or Export node being processed and includes information such as remaining processing time, the frame currently being processed, the number of frames to process.

The *index.html* combines the content of both the *batch_log* and the *batch_log_status* files into a single file.

A message is displayed if an Output node is skipped at process time. This node will be skipped if the host where the output is Wired is unreachable or if the library is read only.

The contents of the HTML file are cleared after each Batch process.

Select:	To:
Backup button and field	<p>Create a backup copy of your current Batch setup automatically. When enabled, a backup copy of the current setup is saved in the file <code>_session.batch</code> in the <i>batch</i> directory.</p> <p>In the Backup frequency field, set the number of minutes between backups.</p>
Grid and Snap buttons	<p>Display a grid for aligning nodes in the schematic. In the Grid size field, set the number of pixels for grid spacing.</p> <p>Enable Snap to snap nodes to the grid.</p>
Reference Buffer controls	<p>These controls appear in the Setup menu when a view other than Schematic is enabled.</p> <p>Click Grab to grab the current frame and use it as a reference for viewing comparatively with another frame in the Reference Buffer. See “The Reference Buffer” on page 81.</p> <p>When you select Reference as the Reference box, and Split On is selected, the “grabbed frame” appears in the “R” side of the split bar. Click Flush to clear the reference buffer and free up memory.</p>

[illegible]

Batch: Node Reference

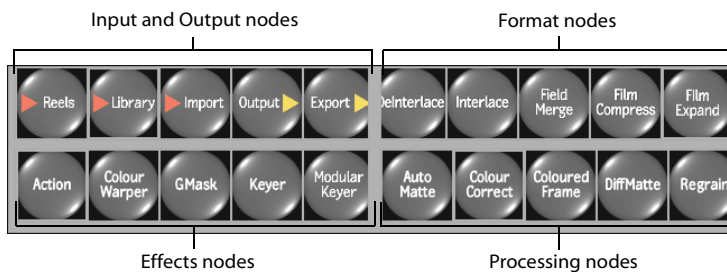
What to do with all those nodes?

This chapter describes how each Batch node works. The nodes are listed in alphabetical order.

About Batch Nodes

Each node in Batch corresponds with one of the **flame** commonly used modules or a command. The **flame** functions that you use in Batch are contained in four sets of node bars:

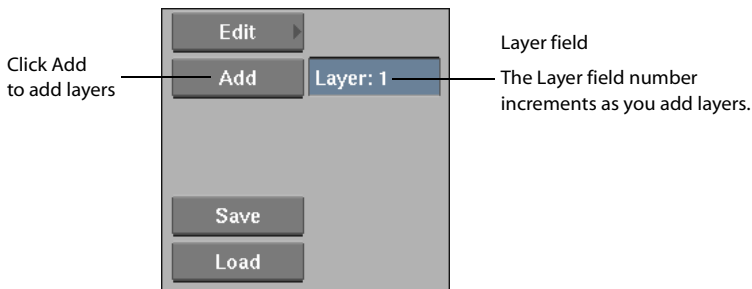
- Input and Output nodes—for loading and importing sources clips, as well as processing and exporting rendered clips.
- Processing nodes—for processing clips using nodes whose functions are found in the Processing menu.
- Format nodes—for formatting the frames in a clip using nodes whose functions are found on the Format menu.
- Effects nodes—for creating special effects using nodes whose functions are found in the Effects menu.



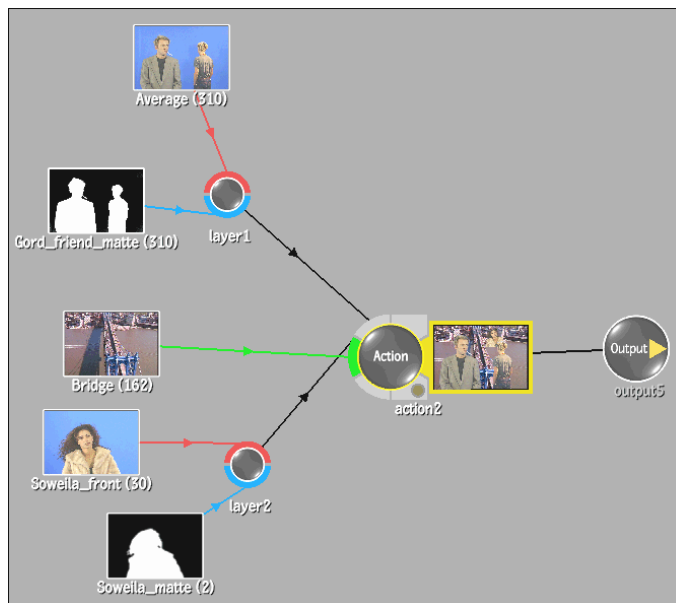
Action Node

The Action node accepts a back clip as input and has its own layer nodes for connecting front and matte clips. You can load and save Action setups directly in Batch.

The Action node is self-contained, which means the specific Action layers are not displayed in the Batch schematic. Use the Layer node to replace clips in your Action setup with clips from Batch.



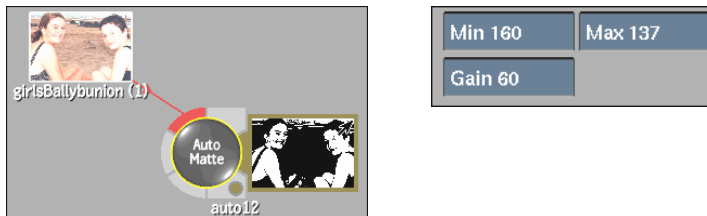
The following example shows an Action module that uses two layer nodes. The clips attached to layer1 and layer2 nodes replace front and matte clips of Layer1 and Layer2 in the Action setup. The Action setup associated with this Action node composites layer1 and layer 2 with the back clip.



See “Action: Layers and Surfaces” on page 907.

Auto Matte Node

The Auto Matte node accepts a front clip as input, which it uses to generate a high-contrast matte. The Auto Matte node menu is similar to the Auto Matte command in the Processing menu.

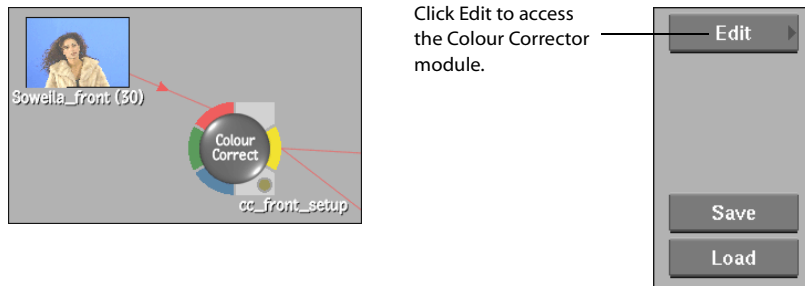


See “Using the Auto Matte Command” on page 500.

Colour Correct Node

The Colour Correct node accepts a front, back, or matte clip as input. In Batch, the results of a colour correction operation depend on which clip or combination of clips are linked to the Colour Correct source tabs. You can load Colour Corrector setups directly in Batch.

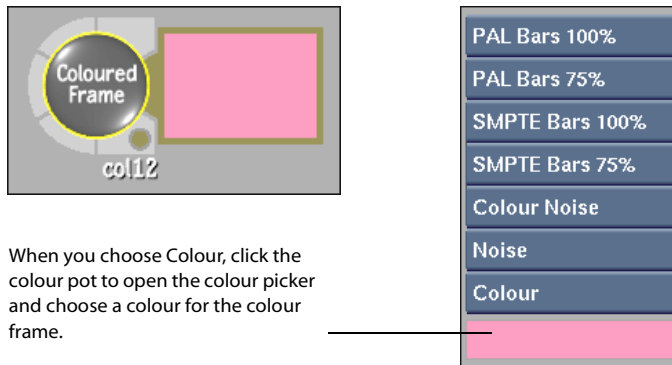
For example, if you connect only a front clip, the colour correction is applied to the entire frame. If, however, you connect both a front and matte clip, the colour correction is only applied to the area defined by the matte.



See “Colour Corrector” on page 515.

Coloured Frame Node

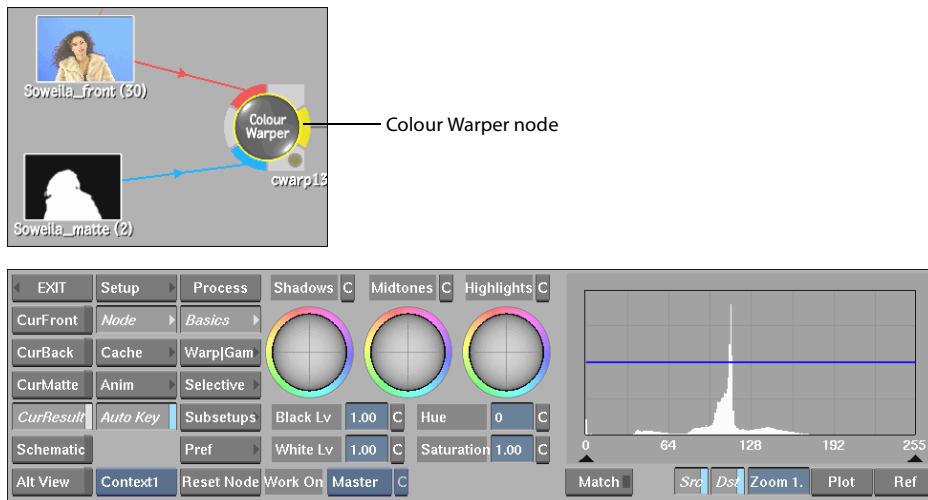
The Coloured Frame node generates a colour bar, noise, or colour clip that can be used as the clip for other nodes. The Coloured Frame node menu is similar to the Coloured Frame command in the Processing menu.



See “Creating Coloured Frames” on page 503.

Colour Warper Node

The Colour Warper node accepts a front clip and a matte clip as input. Use the Colour Warper node to access the Colour Warper module. In Batch, enable CurResult to display the image and the vectorscope in the image area.



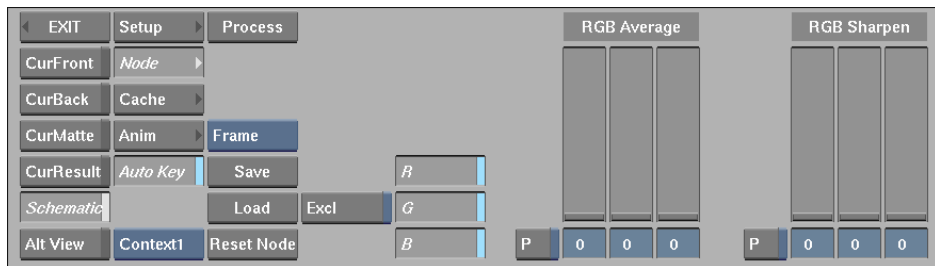
Colour Warper menu in Batch

See “Colour Warper” on page 533.

Degrain and Regrain Nodes

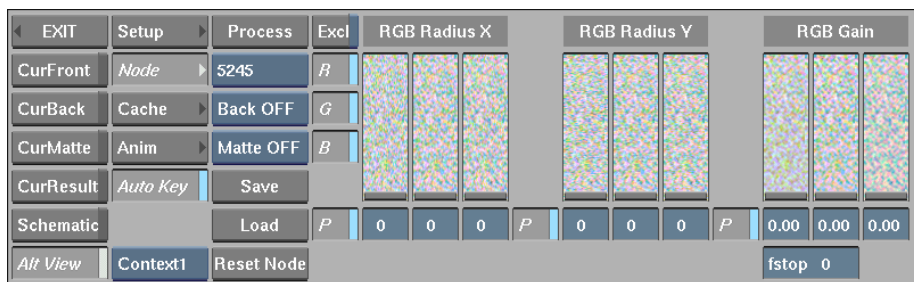
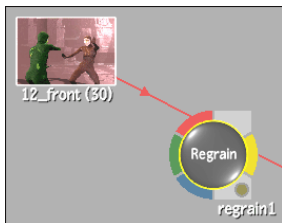
Use the Degrain and Regrain nodes to add or remove grain from the RGB channels of a selected colour in any image. The Degrain and Regrain nodes are the same as the Degrain and Regrain commands in the Processing menu.

The Degrain node accepts only a front clip as input.



Degrain menu in Batch

The Regrain node accepts front, back, and matte clips as input.



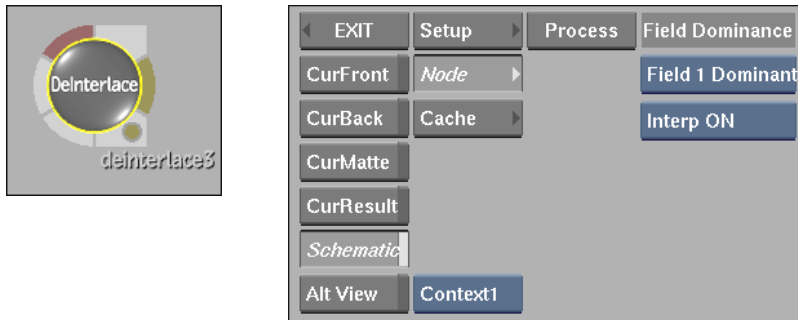
Regrain menu in Batch

See “Degraining and Regraining Clips” on page 504.

DeInterlace and Interlace Nodes

The DeInterlace node separates the odd and even scanlines of a clip. For each frame of the clip, the result clip contains one frame with odd scanlines (Field 1) and one frame with even scanlines (Field 2). See “Deinterlacing Fields” on page 620.

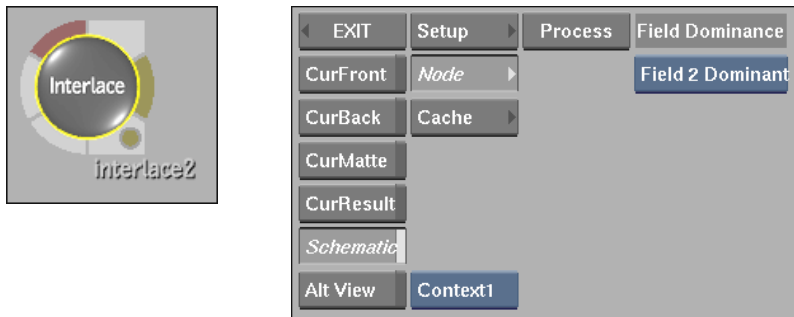
The DeInterlace node accepts a front clip as input.



DeInterlace menu in Batch

The Interlace node interlaces the odd and even scanlines of a clip. For each pair of frames in the input clip, the Field 1 scanlines of one frame are interlaced with the even scanlines of the second frame to produce a single frame in the generated clip. See “Interlacing Fields” on page 623.

The Interlace node accepts a front clip as input.



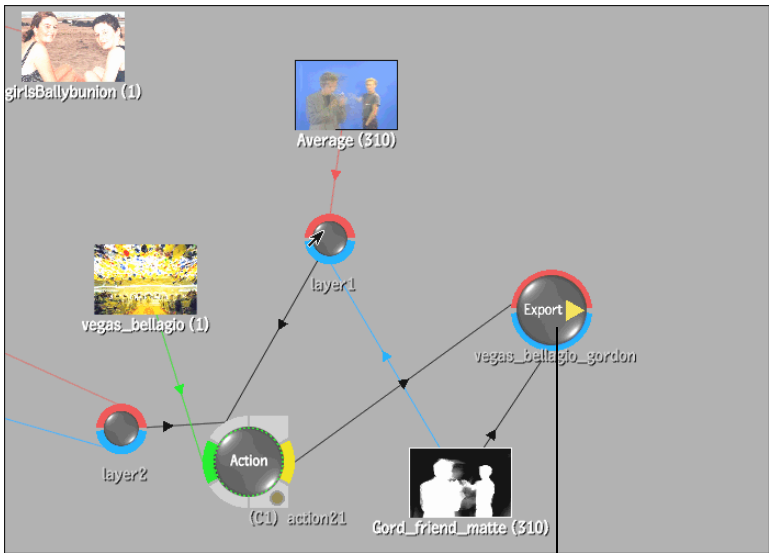
Interlace menu in Batch

Export Node

Use the Export node to export the final processed result directly to disk without saving the clip to the framestore.

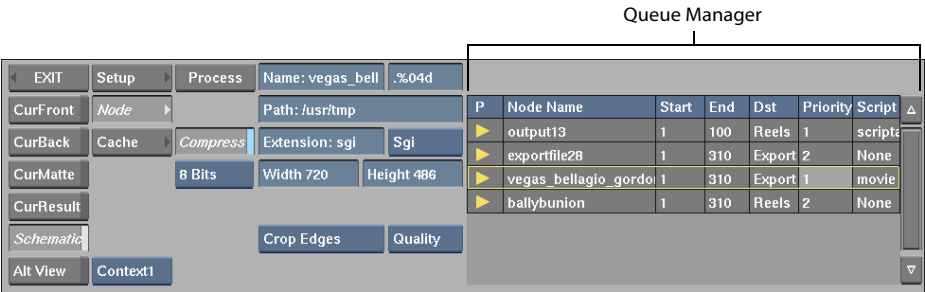
The Export node accepts a front clip and its matte source tab accepts an alpha channel as long as the front tab has an input and if an RGBA supported format such as .sgi is selected for export.

You can export a final matte by connecting the input to the matte tab on the Export node. Connecting the input to the front tab on the Export node produces a final result clip.



You can export a final matte as well as a final result (front).

When you select the Export node, the Image Export controls and Queue Manager appear in the Batch menu.



See “Exporting Images” on page 338 and “Using the Queue Manager” on page 586.

Field Merge Node

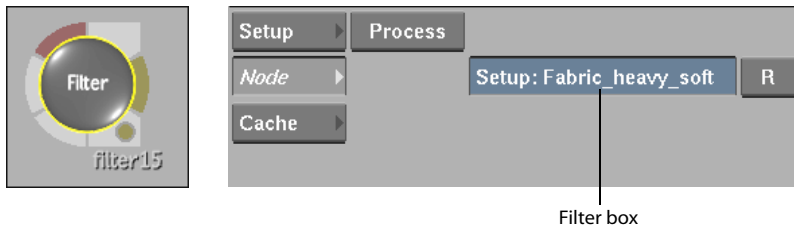
The Field Merge node accepts a front clip. You use the Field Merge node to remove field jitter by merging the fields of a clip.



See “Merging Fields” on page 624.

Filter Node

The Filter node applies different effects to a clip, including textures, blurring, edge detection, embossing, sharpening, or a combination of effects.



To load a filter:

1. Add a Filter node and attach a front clip.
2. Select the Filter node in the schematic.
The menu for the Filter node appears.
3. Click the Filter box.

The file browser appears showing a list of filters. These are the same filters available in the Filter module and Paint.

4. Select a filter.

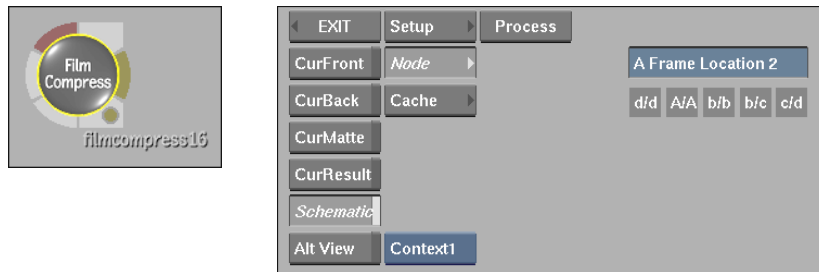
You return to Batch and the name of the selected filter appears in the Filter box.

NOTE: You cannot create or modify filter setups through Batch.

Film Compress and Film Expand Nodes

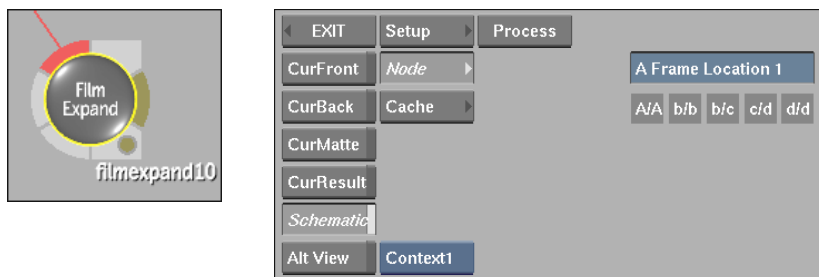
The Film Compress and Film Expand nodes each accept a front clip as input.

The Film Compress node removes field artifacts introduced in a 24-to-30 fps conversion process.



See “Compressing Clips from 30 to 24 fps” on page 618.

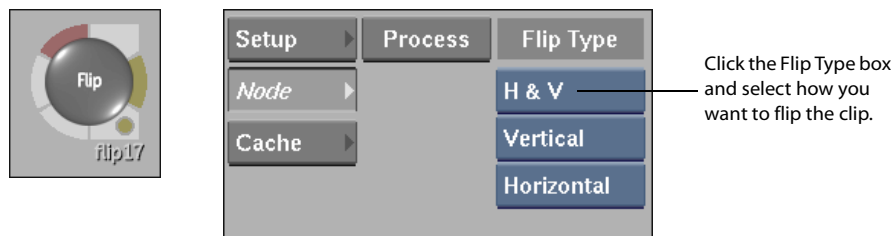
The Film Expand node produces the appropriate field sequence for converting a clip from 24 to 30 fps.



See “Expanding Clips from 24 to 30 fps” on page 621.

Flip Node

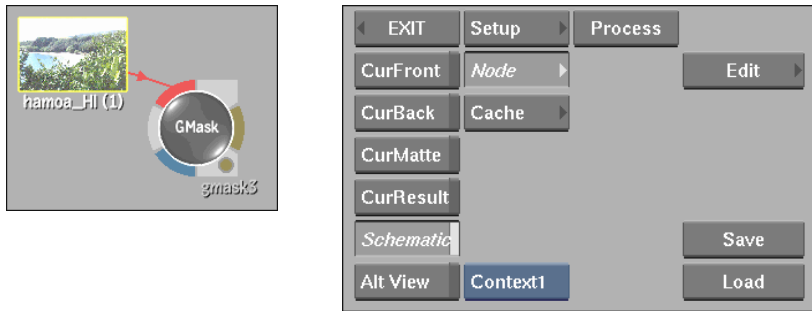
The Flip node accepts a front clip as input. Use the Flip node to generate a mirror image of a clip. You can also flip the frames in a clip horizontally, vertically, or both.



See “Flipping Images” on page 494.

GMask Node

The GMask node accepts a front and a matte clip as input. The GMask node accesses the Garbage Mask menu and includes the Tracer and Region of Interest (ROI) functions, which are found in the Modular Keyer module.



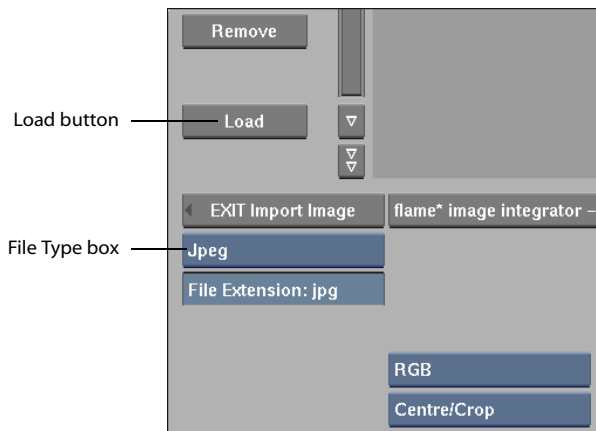
You can save and load GMask setups directly in Batch. See “The Keyer” on page 631.

Import Node

The Import node accesses the Import Image module and lets you import images and clips directly into Batch.

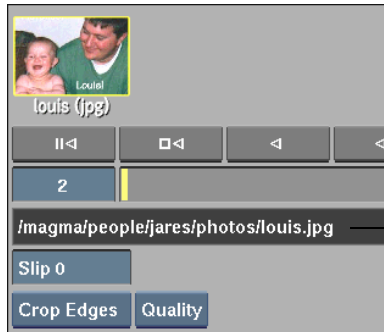
To import images into Batch:

1. In the Batch menu, click Node. If the node bar does not appear, swipe the bar at the left.
2. In the node bar, select Import and drag it to a location on the schematic.
The Import Image menu and file browser appear.
3. In the Image Import menu, select the file type you want to import from the File Type box.



4. In the file browser, select the file(s) you want to import.
5. Select other options on the Import Image menu as required. See “Importing Images” on page 322.
6. Click Load.

You return to Batch and the filename and file type of the image you import appear in the schematic as shown in the following example.

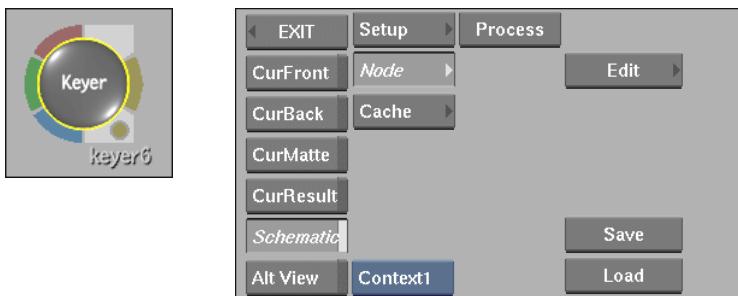


The filename and path appear in the status bar when you click the image.

When you click elsewhere in the schematic, the path name no longer appears.

Keyer Node

The Keyer node accepts front, back, and matte clips as input. When you add a Keyer node, you can choose to generate a result, matte, or composite clip. Use the Result box to select the resulting clip.



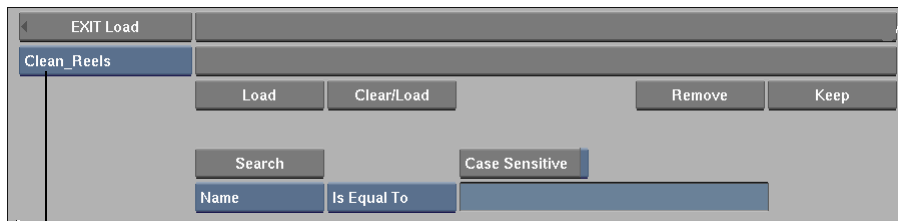
See “The Keyer” on page 631.

Library Node

The Library node accesses the clip library, letting you load clips directly into Batch.

To load clips from the clip library into Batch:

1. In the Batch menu, click Node.
If the node bar does not appear, swipe the bar at the left.
2. In the node bar, select Library and drag it to a location on the schematic.
The clip library appears.
3. In the Library box, select the library you want to browse.

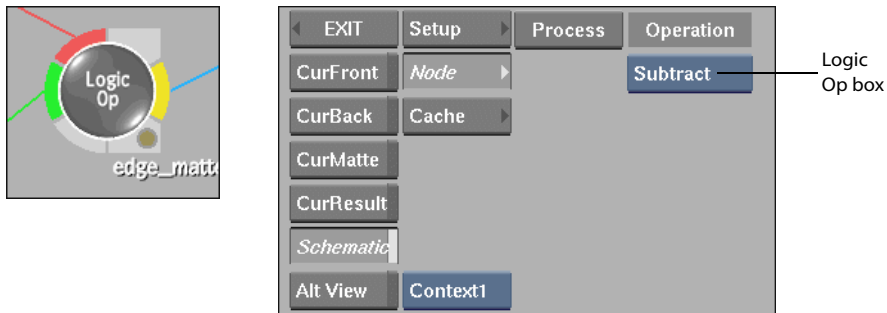


Library box

4. Select the clips you want to load and click Load.
To search the clip library for a specific clip, see “Clip Libraries” on page 197.
You return to Batch and the selected clips appear in the schematic.

Logic Op Node

The Logic Op node accepts a front clip and the Logic Op box lists the logical operations that you can perform on the front clip. The Logic Op node menu is similar to the Logic Op command in the Processing menu.



See “Using Logical Operations” on page 495.

LUT Editor Node

When you import or export Cineon or DPX files, Batch automatically appends a LUT Editor node to the Import or Export Node. A LUT converts a logarithmic image to a linear image or linear images to logarithmic images, while maintaining colour accuracy.

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

NOTE: Refer to “Using a Lookup Table” on page 330 for more information on LUTs.

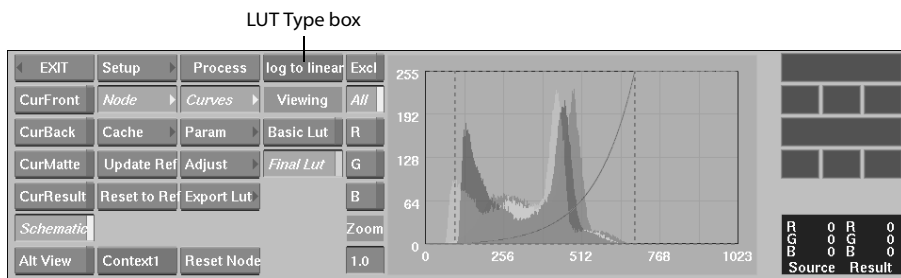
Importing a LUT

You can load previously defined LUTs for use on clips that you are exporting or importing.

To load a predefined LUT:

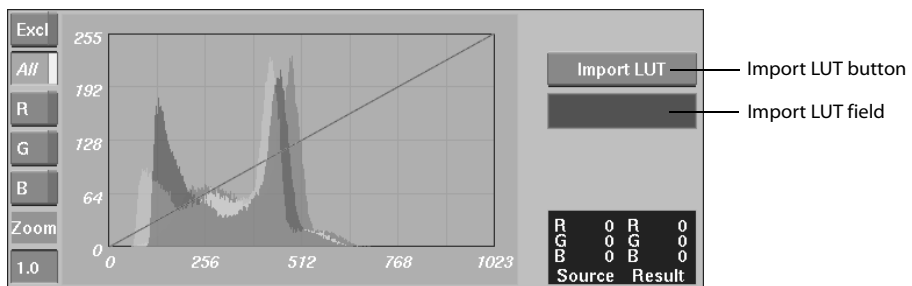
1. Select the LUT Editor node.

The LUT Editor appears.



2. Select Import in the LUT Type box.

The Import LUT button appears.



3. Click Import LUT.

Existing LUTs appear in the file browser.

4. Click on a LUT to select it.

The Export Image or Import Image menu reappears and the name of the selected LUT appears in the field below the LUT button.

Creating a Custom LUT

flame combines the Basic LUT and the Adjust LUT to create the Final LUT. The Final LUT is the LUT that is applied to the image.

To create the Final LUT you need to:

- Set up the Reference image.
- Modify the Basic LUT using the Param or Curves menu.
- Modify the Adjust LUT using the Adjust menu.

NOTE: You can display the Basic LUT and the Adjust LUT curves in the histogram at anytime by enabling the Basic LUT or Adjust LUT buttons.

To create the Final LUT:

1. Select the LUT Editor node.

The LUT Editor appears.

2. Click CurResult.

The image that appears in the image window is the result of the 10-to-8-bit conversion (or 10-to-12-bit conversion, if you are using a 12-bit partition). This conversion takes place according to the default LUT parameters.

3. Click the Update Ref button.

The Reference image is updated with the default CurResult image. Now you can make changes to the LUT using the CurResult image, and compare the result with the default in the reference buffer.

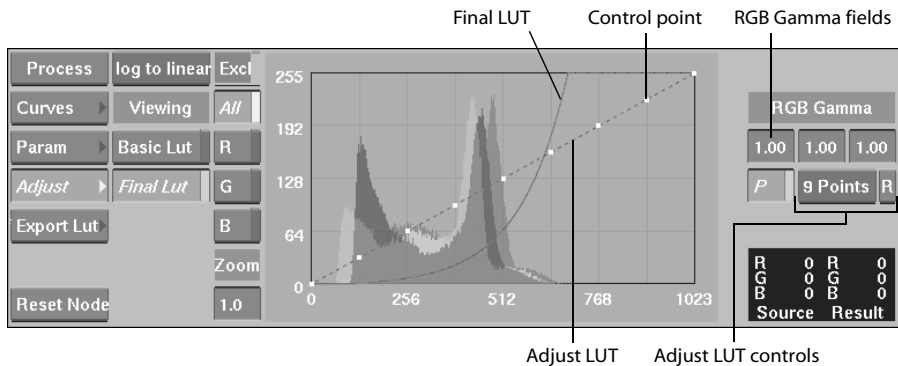
4. Click the CurResult button.

5. Drag the split bar so that you can see both the Reference and CurResult image at the same time.

6. Click Param and modify the Basic LUT parameters, or click Curves to modify the parameters by adjusting the curves using a reference histogram. See “Using the Histogram and RGB Curves” on page 335.

7. Click Adjust.

The Adjust menu appears.



8. Modify the Adjust LUT by dragging the control points on the curve and modifying the values in the RGB Gamma fields (click P to modify all Gamma Field channels simultaneously).

The RGB Gamma and Adjust LUT curve settings are cumulative. The RGB Gamma values shift the entire curve while the control points let you fine-tune specific curve segments. Click Basic LUT and Final LUT to view both the Basic and Final LUTS curves in the histogram.

Editing the Adjust LUT Using Control Points

The following options are available to edit the Adjust LUT with the control points:

- Enable the P button to modify red, green, and blue channels simultaneously.
- Click the Control Point option box to toggle between 3 and 9 control points.
- Click the R button to reset the current curve (All/R/G/B).

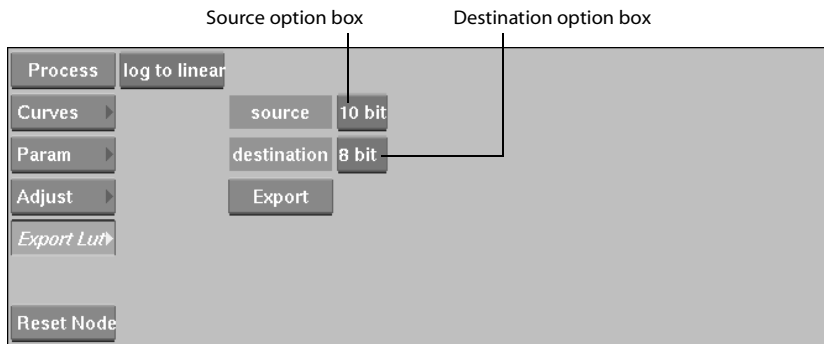
Exporting a LUT

You can export the LUTs you create with the LUT Editor.

To export a LUT:

1. Click Export LUT in the LUT Editor.

The Export LUT menu appears.



2. Use the Source and Destination option boxes to enter the appropriate bit depths.
3. Click Export.
The file browser appears.
4. Name the clip and click Save.

Creating an Isomorphic LUT (1:1)

This procedure lets you define a 1:1 LUT that does not alter the appearance of the clip.

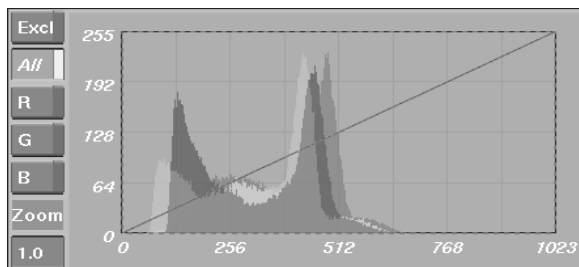
To create an isomorphic LUT:

1. Select the LUT Editor node.

The LUT Editor appears.

2. Select Gamma in the LUT Type box.

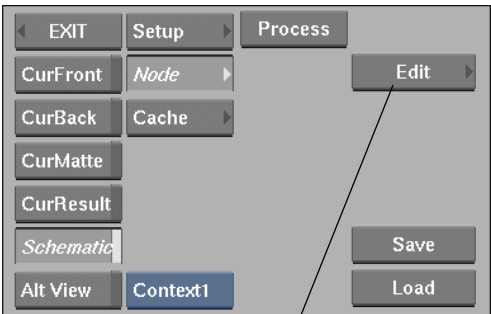
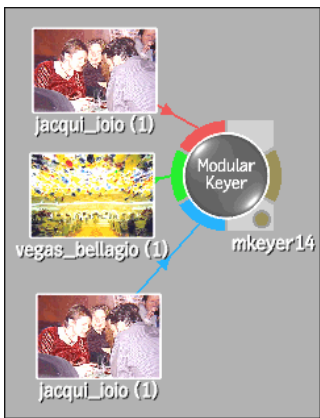
A straight line (1:1) appears in the histogram.



Modular Keyer Node

The Modular Keyer node accepts front, back, and key-in clips as input. You use the key-in clip, which is generally the same as the front clip, to create a matte for the front clip.

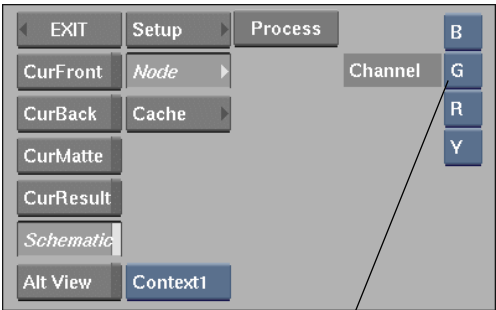
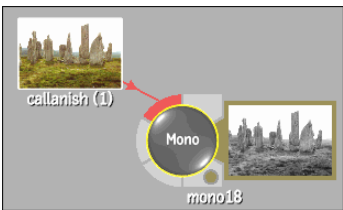
You can save and load Modular Keyer setups directly in Batch.



Click Edit to access the Modular Keyer module.

Monochrome Node

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



Channel box

Negative Node

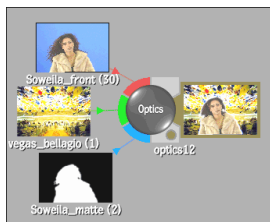
The Negative node generates a negative copy of the front clip. The Negative node does not have a node menu.



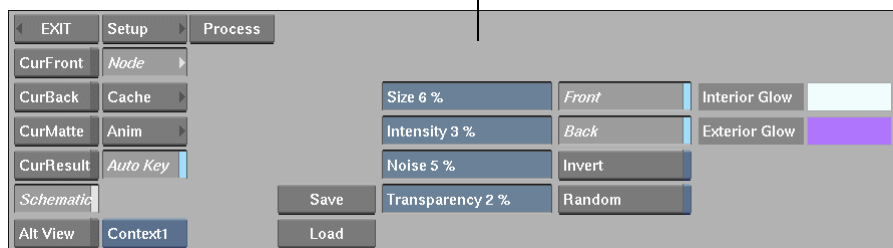
See “Creating a Negative Clip” on page 502.

Optics Node

The Optics node accepts front, back, and matte clips as input. Use the Optics node to add a glow effect to the clip in the process tree.



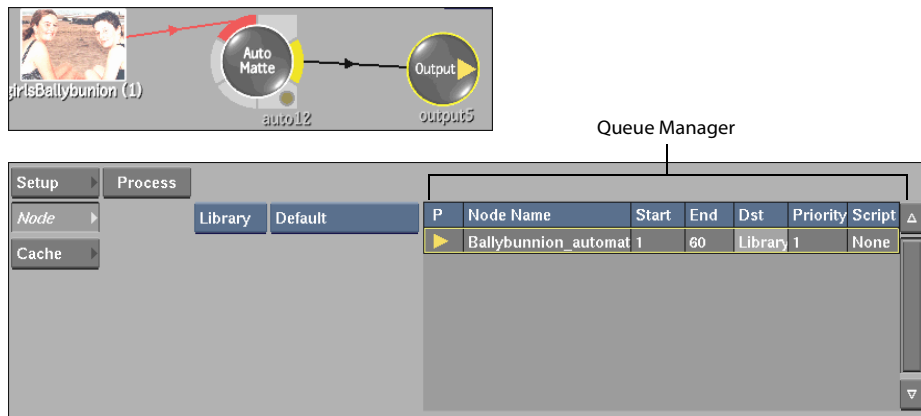
When you click the Optics node, the Optics menu appears in Batch.



See “The Quick Composite Module” on page 786.

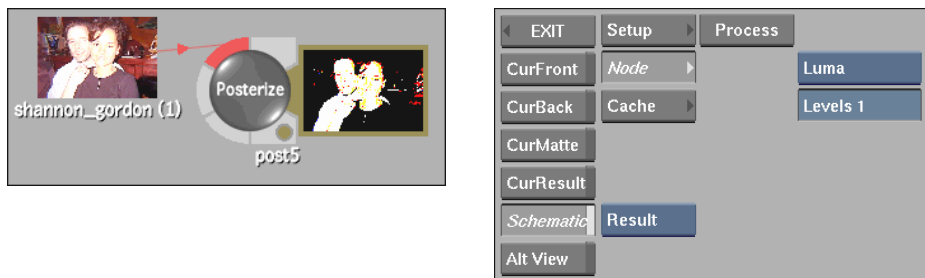
Output Node

The Output node accepts one input from any node. When you select the Output node, the Destination box and Queue Manager appear in the Batch menu. You can choose to process the resulting output clip to the desktop reel or the clip library.



Posterize Node

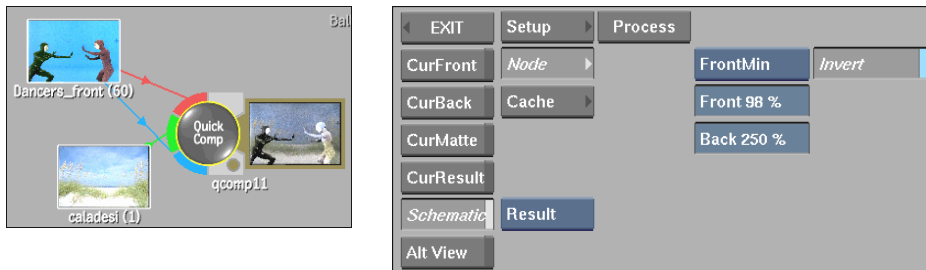
The Posterize node accepts a front clip and produces a posterized result by reducing the number of luminance or chrominance levels in the clip. The Posterize node menu is similar to the Posterize command in the Processing menu.



See “Modifying Luma/Chroma of a Clip” on page 502.

Quick Composite Node

The Quick Composite node accepts front, back, and matte clips. Use the Quick Composite node to composite a front clip and a back clip using a matte clip. The Quick Composite node menu is similar to the Quick Composite module in the Effects menu.



See “The Quick Composite Module” on page 786.

Reels Node

The Reels node accesses the desktop, letting you load clips from the reels into Batch.

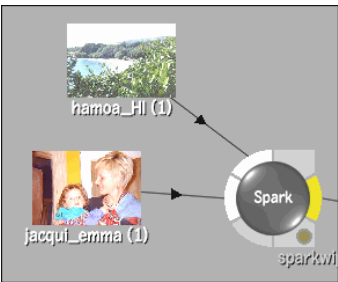
To load images from the desktop reels:

1. In the Batch menu, click Node.
If the node bar does not appear, swipe the bar at the left.
2. In the node bar, select Reels and drag it to a location on the schematic.
The desktop reels appears.
3. Select up to six clips.
4. If you select less than six clips, click Exit Clip Select or click an empty reel to return to Batch.
You return to Batch and the selected clips appear in the schematic.

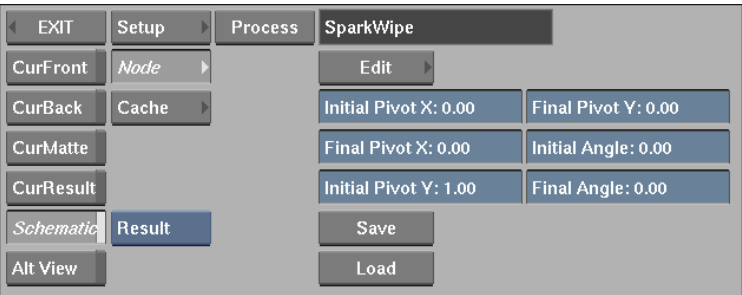
Spark Node

The Spark node accesses the Sparks file browser, letting you add Sparks to the process tree. The number of inputs that a Spark node accepts depends on the Spark. A Sparks node has light grey coloured source tabs since each spark varies in the type of clips it uses.

For example, the following spark wipes from one input clip to another input clip.



SparkWipe menu



NOTE: When you attempt to use a spark, the following error message “SPARK IS NOT SUPPORTED” may appear. This means the selected spark cannot be used with Batch. Contact the company who developed the spark for information on obtaining a compatible version.

[illegible]

Use **flame**'s formatting tools to fine-tune the image quality of your source clips.

Summary

In this chapter, you learn about:

- “Changing Timecode” on page 617
- “Combining RGB Channels in a Clip” on page 618
- “Compressing Clips from 30 to 24 fps” on page 618
- “Dealing Clips” on page 619
- “Deinterlacing Fields” on page 620
- “Expanding Clips from 24 to 30 fps” on page 621
- “The Field Dominance Box” on page 622
- “Interlacing Fields” on page 623
- “Interleaving Two Clips” on page 623
- “Merging Fields” on page 624
- “Reversing Field Dominance” on page 625
- “Separating RGB Channels in a Clip” on page 625
- “Stamping Timecode or Frame Number” on page 626

About Clip Formats

Use the commands in the Format menu to format source material, create visual effects, or solve image-quality problems. To display the Format menu, click the Format button in the Main menu. A brief description of each Format command is provided.

LIBRARY				
EDITING	Deinterlace	Film Compress	Separate	Delete
PROCESSING	Interlace	Film Expand	Combine	Name
EFFECTS	Field Merge	Burn In	Interleave	Move
FORMAT	Reverse Dominance	Change Timecode	Deal	Copy
SYSTEM	Field 1 Dominant	30 Frames Per Second		Search

Burn In — Stamps a timecode or frame numbers on a clip. See “Stamping Timecode or Frame Number” on page 626.

Change Timecode — Changes the timecode for a clip. See “Changing Timecode” on page 617.

Combine — Combines the red channel from one clip, the green channel from another clip, and the blue channel from a third clip to produce a single clip. See “Combining RGB Channels in a Clip” on page 618.

Deal — Divides the number of frames in a source clip equally into a number of smaller clips. See “Dealing Clips” on page 619.

Deinterlace — Separates the odd and even scanlines of a video clip. For each frame of the source clip, the result clip contains: one frame with the odd scanlines (field 1) and one with the even scanlines (field 2). See “Deinterlacing Fields” on page 620.

Field Dominance — Allows you to set the field dominance for the system to field 1 or field 2. See “The Field Dominance Box” on page 622.

Film Compress — Removes field artifacts introduced in a 24-to-30-frames-per-second conversion process. See “Compressing Clips from 30 to 24 fps” on page 618.

Film Expand — Produces the appropriate field sequence for converting a clip from 24 to 30 frames per second. See “Expanding Clips from 24 to 30 fps” on page 621.

Field Merge — Merges the fields of a clip to remove field jitter. See “Merging Fields” on page 624.

Interlace — Interlaces the odd and even scanlines of a video clip. For each pair of frames in the source clip, the odd scanlines of one frame are interlaced with the even scanlines of the second frame to produce a single frame in the generated clip. See “Interlacing Fields” on page 623.

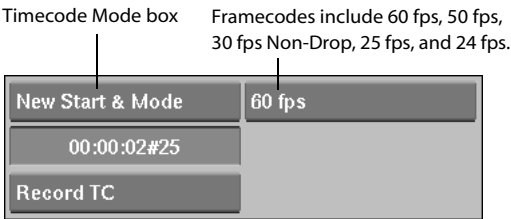
Interleave — Interleaves the frames of two clips to produce a single clip. See “Interleaving Two Clips” on page 623.

Reverse Dominance — Inverts the current field dominance of a selected clip. See “Reversing Field Dominance” on page 625.

Separate — Separates the individual red, green, and blue channels of an image. See “Separating RGB Channels in a Clip” on page 625.

Changing Timecode

Use the Change Timecode command to change the timecode for a clip. This option only applies to stamping a timecode on a clip; the clip's play speed is not altered in any way. When you choose Change Timecode from the Format menu, the Timecode Mode box appears along with framecode, and start options for either Source or Record.



To change the timecode:

1. In the Format menu, click Change Timecode.
2. In the Timecode Mode box, select the timecode mode: either New Start & Mode or New TC Mode.

If you select New TC Mode, the following Start TC Mode box appears.

Select:	To:
Same Start TC	Translate the start timecode according to the start timecode of the selected clip.
Same Start Frame	Translate the start timecode using the same number of frames as the selected clip.

3. Enter the new value in the timecode field, or select the start option.

NOTE: In NTSC mode, you can use the ;/: button on the numeric keypad to switch a timecode between drop and non-drop mode.

4. In the Source TC/Record TC box, select either Source TC or Record TC.
5. Select the clip or sequence for which you want to change the start timecode.
The start timecode changes to the new value.

Combining RGB Channels in a Clip

Use the Combine command to combine individual channels of three different source clips. To generate the new clip, the Combine command uses:

- The red channel information of the first source clip selected
- The green channel information of the second source clip selected
- The blue channel information of the third source clip selected

To combine colour channels:

1. In the Format menu, click Combine.
2. Select the first source clip. The red channel information of this clip will be used for the generated clip.
3. Select the second source clip. The green channel information of this clip will be used in the generated clip.
4. Select the third source clip. The blue channel information of this clip will be used in the generated clip.
5. Select the destination reel.

The generated clip appears on the destination reel.

Compressing Clips from 30 to 24 fps

The Film Compress command removes field artifacts introduced in a 24-to-30 frames-per-second (2:3) conversion. You must use a clip that has already gone through the 2-to-3 conversion process.

To compress clips from 30 to 24 fps:

1. In the Format menu, click Film Compress.

The Timecode Mode box appears.



2. In the Timecode Mode box, select Change TC Mode or Keep TC Mode.

If you select Change TC Mode, the following Start TC Mode box appears:

Select:	To:
Same Start Time	Translate the start timecode according to the start timecode of the selected clip.
Same Start Frame	Translate the start timecode using the same number of frames as the selected clip.

3. Select the source clip.

You must select the frame immediately following the first two frames of jitter in the source clip.

4. Select the destination reel.

The compressed clip appears on the destination reel.

Dealing Clips

The Deal command divides one clip into a number of smaller clips. The source clip frames are “dealt out” between the generated clips. For example, if you divide a 9-frame source clip into four smaller clips, the result clips will be made up of the following frames of the source clip:

- The first clip will contain frames 1, 4, and 7.
- The second clip will contain frames 2, 5, and 8.
- The third clip will contain frames 3, 6, and 9.

1	2	3	4	5	6	7	8	9
---	---	---	---	---	---	---	---	---

Source clip

1	4	7
---	---	---

First result clip

2	5	8
---	---	---

Second result clip

3	6	9
---	---	---

Third result clip

To use the Deal command:

1. In the Format menu, click Deal.

The Clips field appears.



2. Enter the number of clips to be generated in the Clips field.
3. Select the source clip.
4. Select the destination reel.

The generated clips appear on the destination reel.

Deinterlacing Fields

When you are working on interlaced clips, some field information may be lost. This may occur, for example, when you rotoscope in fields or filter a clip. To preserve field information, you can deinterlace the source clip before processing, and then interlace the fields of the processed clip.

The Deinterlace command is used for separating fields 1 and 2 of a source clip. The result clip is twice as long as the original source clip. There are two frames in the result clip for each frame of the source clip: one frame contains field 1, and the other frame contains field 2.

You can deinterlace the fields in a clip with interpolation on or off:

- In most cases, you will deinterlace a clip with interpolation off. The system repeats the fields to fill in the scanlines that separate the fields in the resulting frames.
- To help reduce aliasing artifacts, deinterlace with interpolation on. The system will interpolate between the fields to fill the scanlines that separate the fields in each resulting frame.

To deinterlace a clip:

1. In the Format menu, click Deinterlace.

The Interpolation box appears.



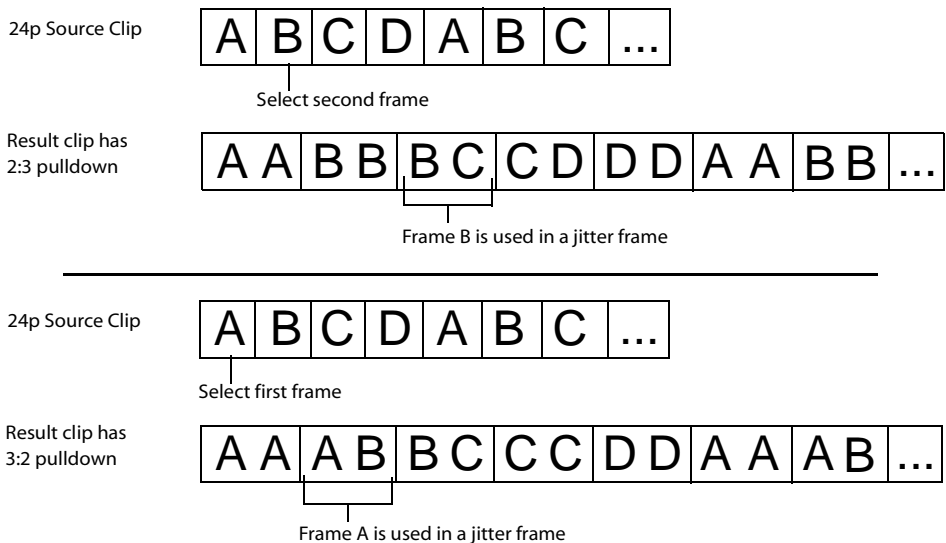
2. In the Interpolation box, select Interpolation On or Interpolation Off.
 3. Select the source clip.
 4. Select the destination reel.
- The deinterlaced clip appears on the destination reel.

Expanding Clips from 24 to 30 fps

Use the Film Expand command to create a 30 fps clip from a 24 fps clip by adding fields in a 2:3 or 3:2 sequence. You can specify where in the clip the pulldown sequence will start by selecting the first frame to be used in a jitter frame. For example, you can create a 30 fps clip with either 2:3 or 3:2 pulldown by selecting for the jitter frame:

- The second frame in the clip, to get a 2:3 pulldown.
- The first frame in the clip, to get a 3:2 pulldown.

The following diagram below shows the result of selecting the first or second frame.



To add a pulldown:

1. For the clip to which you are adding pulldown, display the second frame for a 2:3 pulldown or the first frame for a 3:2 pulldown (either expand the clip or go to the frame).
2. Click Film Expand.



3. Select the Timecode Mode:
 - Select Change TC mode for a 30 fps clip.
 - Select Keep TC mode for a 24 fps clip (for example, for special effects).

When you choose Change TC Mode, you can specify the Start Mode:

Select:	To:
Same Start Time	Convert timecode mode according to start timecode of selected clip.
Same Start Frame	Convert timecode mode using same number of frames as selected clip.

4. Click in the upper left corner of the frame you selected.
5. Select the destination reel.

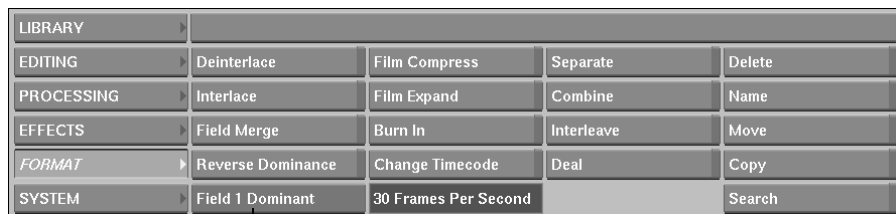
The generated clip appears on the destination reel.

NOTE: To remove 2:3 or 3:2 pulldown, use the Film Compress option. See “Compressing Clips from 30 to 24 fps” on page 618.

The Field Dominance Box

Use the Field Dominance box to set the default field dominance for clips generated during the work session. Every command that renders in fields uses the specified field dominance.

The Field Dominance box is also used in conjunction with the Reverse Dominance command to change the field dominance of a selected clip.



Field Dominance box

Interlacing Fields

Use the Interlace command to recombine fields 1 and 2 of a deinterlaced clip. The result clip is half as long as the original source clip. All odd fields in the first frame of the source clip are interlaced with the even fields in the second frame of the source clip to produce a single frame in the result clip. This processing is repeated for each pair of frames in the source clip.

To interlace a clip:

- 1. In the Format menu, click Interlace.
- 2. Select the source clip.
- 3. Select the destination reel.

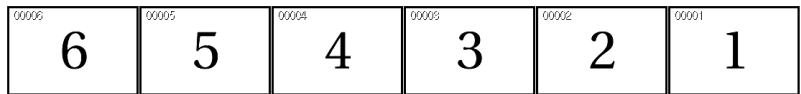
The interlaced clip appears on the destination reel.

Interleaving Two Clips

Use the Interleave command to interleave the frames of two source clips. The interleaving can be performed over any number of frames. For example, interleaving over two frames produces a clip containing one frame of the first source clip, followed by two frames of the second source clip, followed by one frame of the first source clip, and so on. The following example illustrates interleaving over two frames.



Action: The first source clip



Action: The second source clip



Result: The first six frames of the interleaved clip

To interleave two clips:

1. In the Format menu, click Interleave.

The Spacing field appears.



2. Enter the number of frames across which the interleave is to be performed in the Spacing field.
3. Select the first source clip.
4. Select the second source clip.
5. Select the destination reel.

The interleaved clip appears on the destination reel.

Merging Fields

Use the Field Merge command to merge the odd and even fields in each frame of a source clip. This feature allows you to remove effects caused by field jitter.

To merge fields in a clip:

1. In the Format menu, click Field Merge.

The Level field appears.



2. Enter the percentage of blending between fields in the Level field.

If you enter a value of 100%, the average pixel value will be replicated on both the even and the odd line, resulting in total field merging. If you enter a value of 0%, the image remains unchanged. If you enter a value of 50%, each pixel will be replaced by 50% of its original value and 50% of the average value of the even and odd pixels.

3. Select the source clip.
4. Select the destination reel.

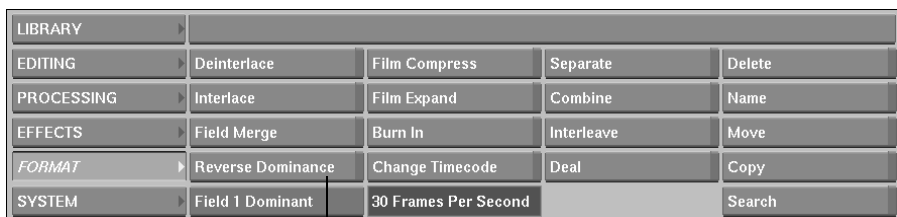
The clip is processed, and the result clip appears on the destination reel.

Reversing Field Dominance

Use the Reverse Dominance command to reverse the field dominance for a selected clip. For example, if the footage that you are using is field 2 dominant, you may want to change the dominance to field 1. If you reverse the order of frames in a clip using the Reverse command in the Editing menu, then you should also reverse the field dominance for the clip. This function drops the first and last fields from the clip, resulting in a clip that is one frame shorter. So, you may want to add "padding frames" before reversing dominance.

To reverse the field dominance of a clip:

1. Click Reverse Dominance.



Reverse Dominance box

2. Select the source clip.

The field dominance for the selected clip is reversed.

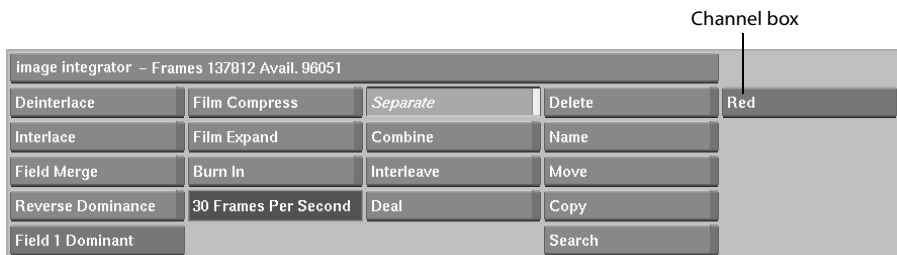
Separating RGB Channels in a Clip

Use the Separate command to separate the red, green, and blue channels of a clip. A monochrome clip containing the information for the selected channel only appears. For example, to generate a clip containing the red channel information of a source clip, select Red in the Channel box.

To separate colour channels:

1. In the Format menu, click Separate.

The Channel box appears.



2. In the Channel box, select the channel that you want to separate.

Select:	To generate:
Red	A monochrome clip containing only red channel information.
Green	A monochrome clip containing only green channel information.
Blue	A monochrome clip containing only blue channel information.
All	Three monochrome clips: one for each channel.

3. Select the source clip.
4. Select the destination reel.
The clip(s) are generated and placed on the destination reel.

Stamping Timecode or Frame Number

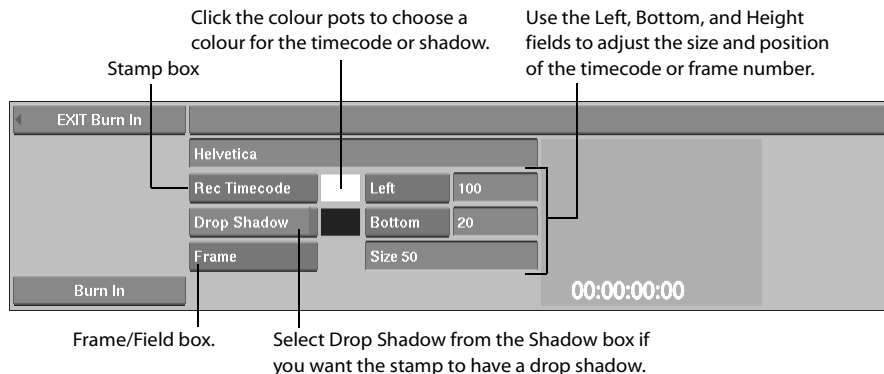
Use the Burn In command to stamp the current timecode or frame number on a clip (frame or field). You can adjust the size, position, colour, and shadow of the timecode or frame number.

This command uses the default font specified on the FontDefault line of the configuration file. To use a different font, change the font specified on this line. For more information, see the configuration file chapter in the *flame Installation Guide*.

To stamp the current timecode or frame number on a clip:

1. In the Format menu, click Burn In.

The Burn In menu appears.



2. In the Stamp box, select Rec. Timecode or Frame Number.
3. In the Frame/Field box select Frame or Field.
4. Select the clip to be stamped.
5. Select a destination reel for the stamped clip.



Section 5: Applying Effects

Tools for radical

keying, tracking

and stabilizing,

warping,

creating titles,

and adding

sparks.

Overview: The Effects Menu

Effectively so

*The Effects menu is the heart of **flame**. It is the gateway to the core special effects features of **flame** — keying, compositing, text, paint, tracking and more.*

Summary

In this chapter, you learn about:

- “The Effects Menu” on page 629
- “Accessing the Effects Menu” on page 630

The Effects Menu

Use the Effects menu to access the Effects modules of **flame**.

The following chapters provide complete details on the modules of the Effects menu.

Chapter(s)	Description
Chapter 33, “The Keyer.”	How to use the traditional Keyer.
Chapter 34, “The Modular Keyer.”	How to use the Modular Keyer.
Chapter 35, “The 3D Keyer.”	How to use the 3D Keyer.
Chapter 37, “Compositing and Optics.”	How to use the Compositor, Optics, and Quick Composite modules.
Chapter 38, “Tracking and Stabilizing.”	How to use Stabilizer for removing motion jitter from a clip, or for multiple-point image tracking.
Chapter 39, “The Warper.”	How to use the Warper for image warping or for creating A/B-type morphs.
Chapter 40, “Text.”	How to use the Text module.
Chapter 42, “Action: Overview and Setup Options,” to Chapter 47, “Action: The Particle System.”	How to use the Action module.
Chapter 48, “Paint: Overview,” to Chapter 56, “Paint: Creating, Loading, and Saving Setups.”	How to use the Paint module.

Accessing the Effects Menu

To access the Effects menu, click the Effects button in the Main menu. The following table

LIBRARY					
EDITING	Paint	Keyer	Sparks	L	Delete
PROCESSING	Warper	Compositor	Sparks	L	Name
EFFECTS	Text	Quick Composite	Sparks	L	Move
FORMAT	Stabilizer	Optics	Sparks	L	Copy
SYSTEM	Action	Modular Keyer	Sparks	L	Search

briefly explains the purpose of the modules and commands accessed from the Effects menu:

Module or Command:	Description:
---------------------------	---------------------

Paint	Use Paint to create images from scratch or to retouch existing images. The anti-aliased brushes are used to apply colour, special effects media, and filters to the image. The graphics animation tools allow you to create rotoscoped sequences.
Warper	Use the Warper to warp images and to create A/B-type morphs.
Text	Use the Text module to add text to a clip, and control font size, style, and format, as well as leading and kerning. Add text rolls for credits, text crawls, and bumpers.
Stabilizer	Use the Stabilizer to remove motion jitter from a clip, or to track images. Also, you can analyze motion data and copy it into Action or the Compositor.
Action	In Action, you create effects and animations by manipulating objects in the scene. Objects you work with in Action include surfaces, light sources, axes, particles, shadows, and the camera. Create optical effects by adding light sources, and add 3D objects from other products (such as 3d studio max®).
Keyer	Use the Keyer to create superior quality keys. You can build the key using the RGB, YUV, or HLS colour space. The colour correction tools can be used to remove colour spill from the keyed image.
Compositor	Use the Compositor to create a composite clip from a foreground clip, a background clip, and a matte clip. The animation feature allows you to create 2D keyframe animations of the object.
Quick Composite	Use the Quick Composite command to create a composite clip using a front clip, a back clip, and a matte clip. You can adjust the brightness of the front and back clips to change the brightness of the resulting clip.
Optics	Use the Optics module to apply glow to a clip.
Modular Keyer	The Modular Keyer provides all the tools found in the traditional Keyer, as well as other, more sophisticated tools that allow you to easily key much more challenging shots. Using a processing pipeline, you can specify the order in which you want processes to be applied to clips, giving you almost unlimited flexibility.
Sparks	Use the five Spark buttons to access third-party software plug-ins.

Keying is the process of isolating a region of an image by selecting pixels and associating a matte value with the selection. You can then use the matte to create a composite using a front and back image.

Summary

In this chapter, you learn about:

- “Accessing the Keyer” on page 632
- “Selecting a Keying Technique” on page 633
- “Creating a Key by Extracting a Single Colour” on page 633
- “Creating a Key by Extracting a Range of Colours” on page 635
- “Creating a Key by Setting the Luminance” on page 640
- “Adjusting the Luminance of the Key” on page 641
- “Adjusting the Key” on page 646
- “Removing Background Spill and Colour Correcting Your Clips” on page 649
- “Animating Your Key” on page 652
- “Processing Your Clips” on page 652
- “Keyer Hot Keys” on page 654

About the Keyer

To create a composite, a matte is created and applied to the two clips, and the front clip is superimposed on the back clip. Usually, a subject is shot in front of an evenly lit, pure blue or green screen, and the blue or green is replaced by another image that becomes the background.

The Keyer offers a variety of tools to replace a blue or green screen. To learn about using garbage masks when pulling a key, see Chapter 36, “Garbage Masks and the Tracer.”

NOTE: For some quick step-by-step methods to pull a key, see the *Precision Keying* lesson of the *flame Tutorial*.

Accessing the Keyer

You load three clips into the Keyer to create a composite: a front clip, a back clip, and a key-in clip. The key-in clip, which is usually the same clip as the front clip, is used to create the matte for the front clip.

To access the Keyer menu:

1. Click the Keyer button in the Effects menu.
2. Select the front, back, and key-in clips.
3. Select the destination.

The Keyer menu appears.



HINT: Use the S option on the Keyer button to automatically load the same clips used in the previous session. Click the S option, and then select the destination.

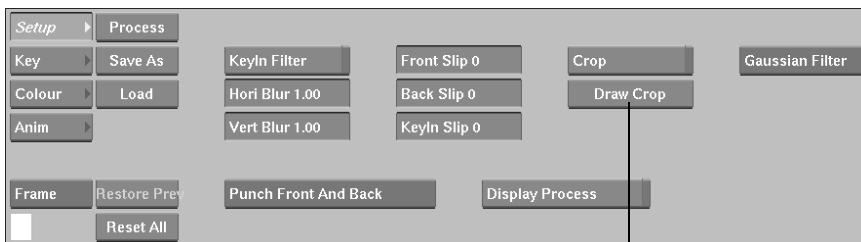
Cropping the Key Area

You can create a crop box to limit the area where the key setup is applied. This will save you processing time while setting up the key, especially at high resolutions.

To draw the crop box:

1. Click the Setup button.

The Setup menu appears.



Draw Crop button

2. Click the Draw Crop button.
3. Draw crop box on image by selecting a point on image and dragging cursor diagonally.
4. Click the Crop On button to enable the crop box.

Selecting a Keying Technique

There are three techniques you can use to create a key:

- Extract a single colour from the key-in clip. This is done using the functions in the CHAN menu. You can extract red, green, blue, or a custom colour that you select from the key-in clip. This method is most useful for clips containing transparencies such as glass or smoke. For more information, see “Creating a Key by Extracting a Single Colour” on page 633.
- Define a range of colours to be keyed out in the key-in clip. This is done using one of the colour models in the Range menu. Use this technique for clips where the colour you are extracting contains impurities. For more information, see “Creating a Key by Extracting a Range of Colours” on page 635.
- Set a key according to the clip’s luminance values. This is done using the functions in the LUM menu. This technique is most often used for clips with a high contrast or filmed against a black background, or when bringing in a matte (greyscale) as a key-in source. For more information, see “Creating a Key by Setting the Luminance” on page 640.

If you cannot create a good key using one technique, try a different one.

Creating a Key by Extracting a Single Colour

When you load clips, the CHAN (Channel) menu is displayed by default. You can extract one of the three primary colours or a custom colour from the key-in clip.

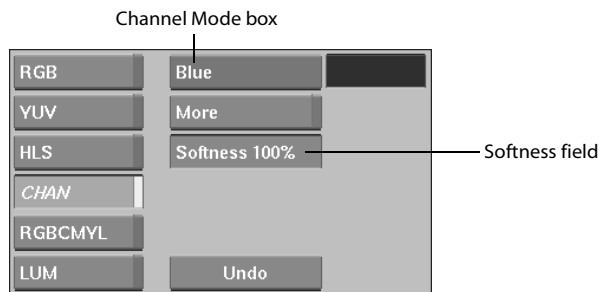
Extracting a Primary Colour

Extract one of the three primary colours in the key-in clip when you have a front clip shot in front of a blue, red, or green screen.

To pull a key by extracting a primary colour:

1. Click Key, Range, and then CHAN.

The Channel menu appears.



2. Select one of the primary colour options (Red, Green, Blue) from the Channel Mode box.
3. Enable More to enhance the keying effect.

The More option extends the range of the colour to be extracted from the key-in clip.

4. You can increase or decrease the softness for the key using the Softness field.

The softness value determines how much of key-in clip is partially transparent in matte.

Softness is used to create a smoother transition between front and back clips in composite clip.

5. Click Result to see the result of your key.

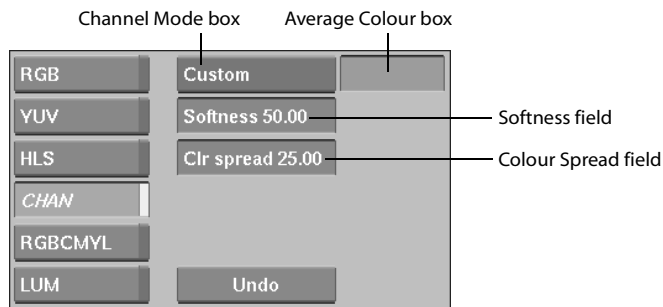
Extracting a Custom Colour

Use the Custom colour channel option to select the colour to be extracted from the key-in clip. This option can give good results when keying transparencies, particularly by experimenting with the Softness and Colour Spread values.

To pull a key by extracting a custom colour:

1. Click Key, Range, and then CHAN.

The Channel menu appears.



2. Select the Custom colour option from the Channel Mode box.
3. Click the Average Colour box. The cursor changes to a colour picker.
4. Click on the image to select a single colour, or drag the colour picker across a region to obtain the average colour of those encountered by the colour picker. Try selecting different colours within the background area to get the best result.

HINT: To keep shadows, click just outside them. To remove shadows, click inside them.

5. Adjust the softness for the key using the Softness field.
Increasing softness raises level of grey in the matte. Lowering softness makes matte sharper.
6. Adjust the colour spread for the key using the Clr Spread field.
Increasing the colour spread value extends the range of colours extracted from the key-in clip.
7. Click Result to see the result of your key.

NOTE: To improve the key at this point, try adjusting it with the histogram. See “Adjusting the Luminance of the Key” on page 641.

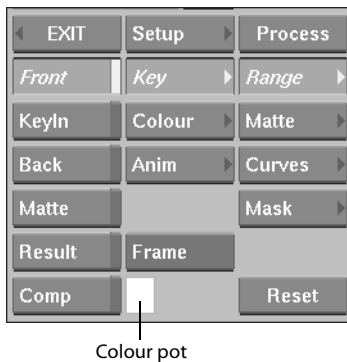
Viewing a Key with a Solid Colour Background

While creating the key, you can view the composite using a solid colour for the background in place of the back clip. This can help to see details in the image, such as colour spill, that you may otherwise miss. You can use the default colour (white), or select a colour using the colour picker.

NOTE: This view does not affect the final render.

To use a solid colour background:

1. Click the colour pot next to the Comp button.



The colour picker appears.

2. Select a colour for the background using the colour picker. For information on the colour picker, see “The Colour Picker” on page 57.
Click on the colour pot next to the Comp button to store the colour.
3. Click the Comp button to view the composite with the selected background colour.

Creating a Key by Extracting a Range of Colours

The second technique used to create a key is by extracting a range of colours in the key-in clip. When you build a key by defining a colour range, you can use one of four colour models, each of which interprets the key-in clip differently, and gives a slightly different result. You can choose the RGB, YUV, HLS or RGBCMYL colour models.

Once you choose a colour model, you set a range of colours to become partially transparent in the key-in clip to soften the transition between the front and back clips in the composite image. This is called the softness range. All pixels in the front clip within the softness range become grey in the matte. The key should have the greatest possible softness value.

After setting the softness range, you set a range of colours to be keyed out in the key-in clip. This is called the tolerance range. All pixels in the front clip within the tolerance range become black

in the matte. The tolerance range must not be too large or the edge of the composite will be too hard and the subject in the front clip will appear to be pasted into the back clip.

Selecting a Colour Model

There are four colour models you can use.

HLS

In the HLS menu, you set the softness and tolerance ranges using the hue, luminance, and saturation channels.

YUV

In the YUV menu, you set the softness and tolerance ranges using the luma and chroma signals of YUV component video.

RGB

In the RGB menu, you set the softness and tolerance ranges using the red, green, and blue channels.

RGBCMYL

In the RGBCMYL menu, you set the softness and tolerance ranges using the red, green, blue, cyan, magenta, yellow, and luminance channels. This colour model provides sub-pixel resolution.

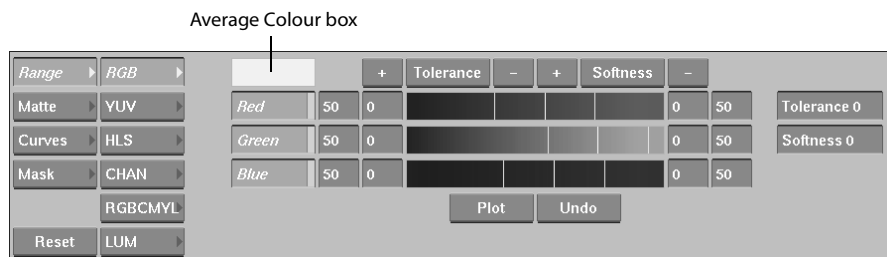
Trying Out the Different Colour Models

To determine which colour model will give you the best key, try creating a sample matte for each model. Each model operates independently of the others. When you switch from one model to another, the values you set are retained, so you can easily compare the results.

To create a sample matte for each colour:

1. Load your clips into the Keyer.
2. Click the Key button.
3. Click the Range button.

The Range menu appears.



- Click the KeyIn button to view the key-in clip.

- Select a colour model.

The softness for each model is automatically set to 50, which helps you to quickly gauge which model will create the best key for your clip.

- At frame 1, click the Average Colour box.

- Drag the colour picker around the area that you want to key out.

The average colour of the colours the colour picker samples appears in the Average Colour box. In each channel of the colour model, the channel value of the average colour appears as a white line. All the pixels in the key-in clip with the selected colour value are keyed out.

The yellow lines in the colour model fields indicate the limits of the softness range. The pixels in the front clip with colour values at the centre of the range are black (transparent) in the matte. As you move away from the centre, the pixels become more opaque.



Softness range

- Click the Matte button.



Matte button for viewing the matte

The matte for the selected colour model appears in the image window.

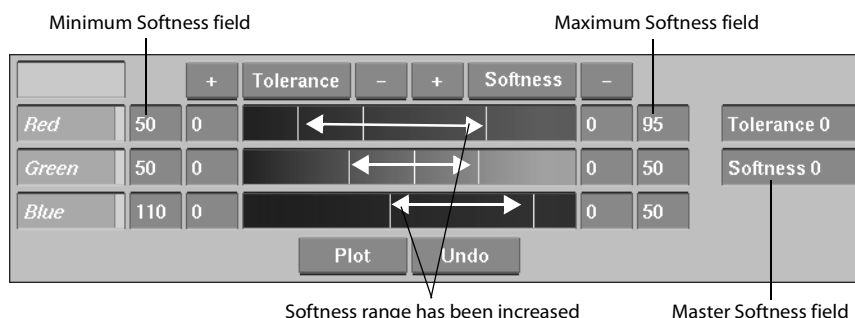
- Repeat steps 4-8 to plot the average colour using the other colour models, then view each of the resulting mattes to determine which model gives the best result. Once you decide on a colour model, use the tools described in the next sections to refine your matte.

Setting the Softness Range

After you choose a colour model and set the average colour, adjust the softness range. The softness at the edges of the matte increases as you increase the softness range.

To set the softness:

1. Zoom in to see the edges of the matte more clearly.
2. Click the Soft button in the selected colour model menu.
The cursor changes to a colour picker.
3. Position the colour picker at the edge of the matte. To increase the softness, click and slowly drag the colour picker toward the centre of the matte. The values within the area you selected are used to adjust the maximum and minimum values for the softness range. The positions of the yellow lines change as you drag the cursor on the image.



4. To increase or decrease the softness range, click the + or - button beside the Soft button and then click on an area of the image.

Setting the Softness Range Using the Numeric Fields

You can also set the maximum and minimum values for the softness range using the numeric fields on either side of the colour gradients.

To set the softness range using the numeric fields:

1. Set the minimum value for the softness range using the Minimum Softness field on the left side of the colour bar.
2. Set the maximum value for the softness range using the Maximum Softness field on the right side of the colour bar.

Using the Master Softness Field

You can adjust the maximum and minimum softness values for all channels using the Master Softness field. When you set a value in the Master Softness field, the value is added to the existing softness value of each channel. The Master Softness field is reset to 0 after you set the value.

Using the Plot Tool

You can display the colour value for any pixel in the key-in clip using the Plot tool. You can then adjust the tolerance or softness range so that the pixel falls within one of the ranges. For example, plot pixels at the edges of the matte to check for softness, or pixels in the background for tolerance.

To plot a pixel's colour values:

1. Click the Plot button (or press the **P** hot key).

The cursor becomes a colour picker.

2. Select a pixel in the image area.

A red bar appears in each of the colour gradients showing the colour value of the pixel.

Setting the Tolerance Range

Once you define the softness range, you can set the tolerance range to remove the greys outside the key shape. The maximum and minimum tolerance values define the range of colours to be keyed out in the key-in clip.

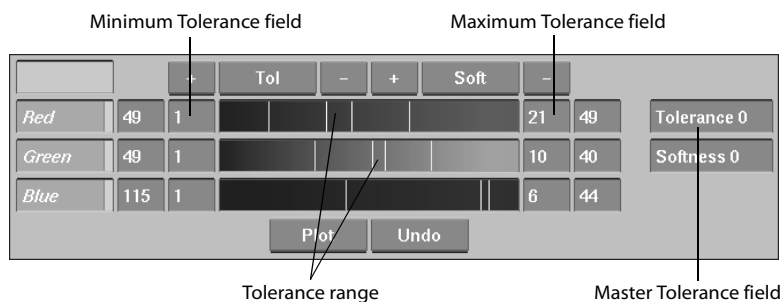
To set the tolerance range:

1. Click the Tol (Tolerance) button.

The cursor becomes a colour picker.

2. Drag the colour picker around the area of the image that you want to key out.

The colour values that the colour picker samples are used to set the maximum and minimum values for the tolerance range. The white lines define the limits of the tolerance range. All colour values between the white lines are extracted from the key-in clip.



3. To increase or decrease the tolerance range, click the + or - button beside the Tol button.

NOTE: An alternate method of setting the tolerance range is to press **CTRL** and draw a rectangle in the area of the image that you want to key out.

Setting the Tolerance Range Using the Numeric Fields

You can also set the maximum and minimum values for the tolerance range using the numeric fields on either side of the colour gradients.

To set the tolerance range using the numeric fields:

1. Set the minimum value for the tolerance range using the Minimum Tolerance field on the left side of the colour bar.
2. Set the maximum value for the tolerance range using the Maximum Tolerance field on the right side of the colour bar.

Using the Master Tolerance Field

You can adjust maximum and minimum tolerance values for all channels using Master Tolerance field. When you set a value in the Master Tolerance field, the value is added to the existing tolerance value of each channel. The Master Tolerance field is reset to 0 after you set the value.

Creating a Key by Setting the Luminance

You can create a key by using only the Luminance channel. The softness and tolerance values are expressed in percentage.

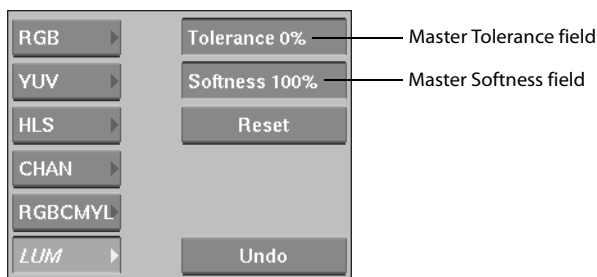
You can also use the Luminance channel to adjust a matte that has already been rendered. Load the matte as the key-in clip, and then adjust it in the LUM menu.

NOTE: When you load a matte as the key-in clip and open the LUM menu with default menu values, the resulting Keyer matte is identical to the original matte.

To create a key by setting the luminance:

1. Click Key, Range, and then LUM.

The Luminance menu appears.



2. Set the softness in the Master Softness field.
3. Set the tolerance in the Master Tolerance field.

A value of 100 for the tolerance creates a matte that is entirely opaque.

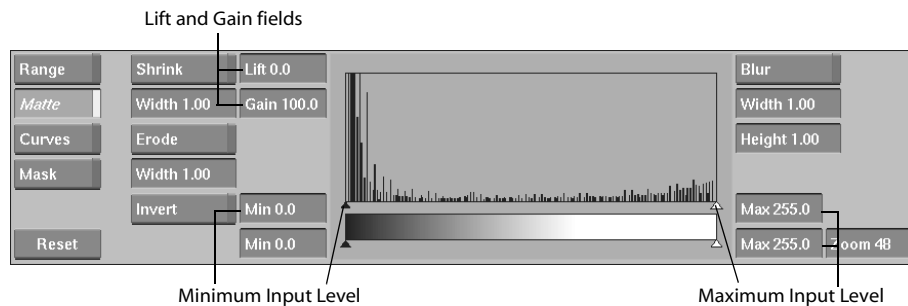
NOTE: You can reset values in the Master Softness and Tolerance fields by clicking the Reset button.

Adjusting the Luminance of the Key

After you create a key, you adjust its luminance. The histogram, in the Matte menu, displays the distribution of luminance values within the matte. The horizontal axis of the histogram represents the range of luminance values in the matte and spans from 0 (black) to 255 (white). The vertical axis shows the number of pixels at each luminance value.

Setting the Range of Luminance Values

Use the Input Level controls to set the range of luminance values in the matte. You can darken the black areas of the matte, or remove grey from the white areas of the matte.



Removing Grey from the Black Areas of the Matte

The Minimum Input Level sets the start of the range of luminance values. Pixels with luminance values below the Minimum Input Level are mapped to black (0).

You can set the Minimum Input level by dragging the black triangle, or by setting a value in the Minimum Input Level field.



The matte before adjusting the input levels



The matte after lowering the Minimum Input level

Removing Grey from the White Areas of the Matte

The Maximum Input Level sets the end of the range of luminance values. Pixels with luminance values greater than the Maximum Input Level are mapped to white (255).

You can set the Maximum Input level by dragging the white triangle, or by setting a value in the Maximum Input Level field.



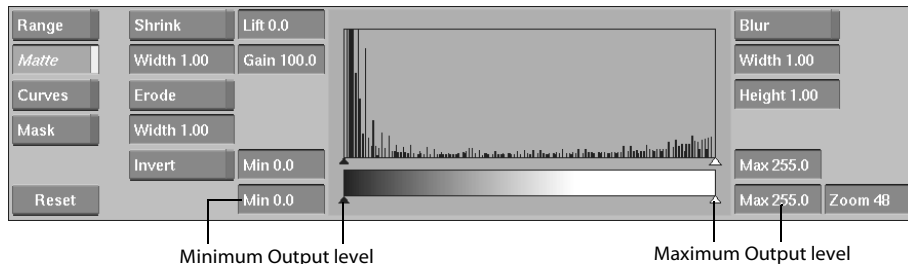
The matte before adjusting the Input Levels



The matte after lowering the Maximum Input Level

Remapping Black and White

You can use the Output Levels to remap the luminance values for black (0) and white (255). You can brighten the dark areas of the matte or darken the white areas.



Brightening the Matte

The Minimum Output level sets the luminance value of all black (0) pixels in the matte. All black pixels in the matte are mapped to the luminance value set by the Minimum Output level.

You can set the Minimum Output level by dragging the black triangle, or by setting the value in the Minimum Output Level field.



The matte before adjusting the Output levels



The matte after lowering the Minimum Output level

Darkening the Matte

The Maximum Output level sets the luminance value of white (255). All white pixels in the image are mapped to the luminance value set by the Maximum Output level.

You can set the Maximum Output level by dragging the white triangle, or by setting the value in the Maximum Output Level field.



The matte before adjusting the Output levels



The matte after lowering the Maximum Output level

Boosting the Key's Luminance Using Gain and Lift

You can remove the grey from the key by increasing the gain and lowering the lift values in the Matte menu. Increase the gain to eliminate the light greys that may be in the white area of the matte, and decrease the lift to eliminate dark greys in the black area of the matte.

NOTE: Adjusting these values increases the contrast and may harden the edges of the matte.

Using the Luminance Curves

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

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

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

$$\text{Result} = F * \text{FrontLUT} + B * \text{BackLUT}$$

where:

F = the R, G and B values of the front image.

B = the R, G and B values of the back image.

FrontLUT is the front matte pixel value, remapped according to any luminance curve change made in this menu. The value is expressed as a decimal, where, for example,

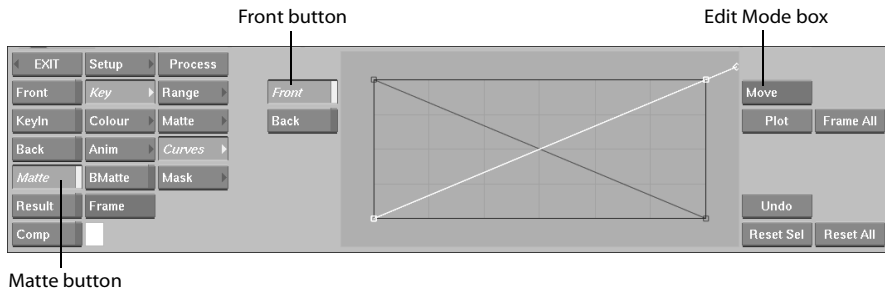
in 8-bit mode, $0 = 0$, $127.5 = 0.5$ and $255 = 1$

in 12-bit mode, $0 = 0$, $2047.5 = 0.5$ and $4095 = 1$

BackLUT is the back matte pixel value, remapped according to any luminance curve change made in this menu. The value is expressed as a decimal, as for the FrontLUT.

To adjust the luminance curve:

1. Click the Curves button in the Key menu.



2. Select the image you want to view as you adjust the curves.

Select: To:

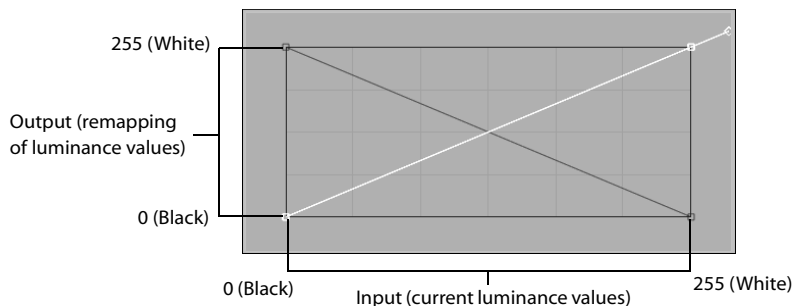
Result View the composite clip.

Matte View the front matte. You cannot see changes made to the back matte when this view is selected.

Bmatte View the back matte. You cannot see changes made to the front matte when this view is selected.

Comp View the composite with a coloured background. The default colour is white. To select a different colour, click the colour pot to the right of the Comp button. The colour picker appears. For information on using the colour picker, see "The Colour Picker" on page 57.

3. To adjust the luminance curve for the front matte, click the Front button. To adjust the back matte curve, click the Back button. Alternatively, click on a curve to select it. The selected curve changes to white.



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

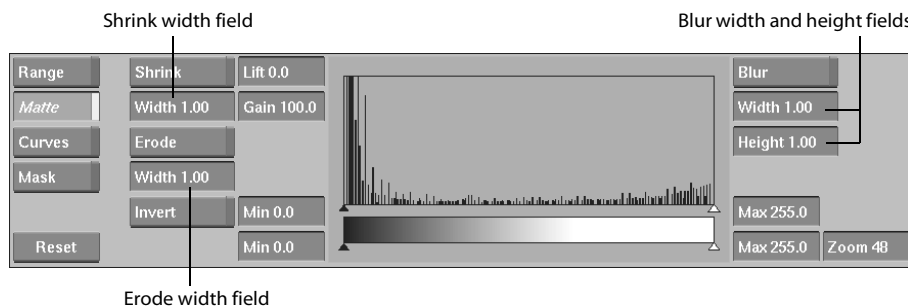
For complete information on using edit modes, see “Editing Animation Curves and Keyframes” on page 134.

Adjusting the Key

After creating the key and adjusting the luminance, you can modify the edges of the key by blurring the key-in clip or the edge of the matte, applying filters to the edge of the matte, or inverting and then adjusting the matte.

Modifying the Edges of the Key

There are three controls in the Matte menu you can use to enhance the edge of the keyed image.



Use:	To:
Shrink	Remove pixels from the edge of the matte. It should not be used when the object in the front clip has soft edges such as hair.
Erode	Blend the light and dark edges of the matte.
Blur	Apply a softening filter to the edge of the matte.

NOTE: For these options, you can select either a Gaussian filter or Box filter. For more information on selecting filters, see “Selecting a Filter for the Blur Options” on page 648.

To shrink the edge of the matte:

1. Click the Key button.
2. Click the Matte button.
3. Set a value in the Width field below the Shrink button.

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

4. Enable the Shrink button.



The matte before enabling the Shrink filter



The matte after setting the shrink width value to 1.00

To erode the edge of the matte:

1. Click the Key button.
2. Click the Matte button.
3. Set a value in the Erode width field.

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

4. Enable the Erode button.



The matte before enabling the Erode filter



The matte after setting the erode width value to 1.00

To blur the edge of the matte:

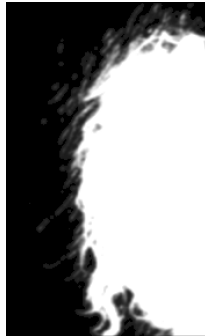
1. Click the Key button.
2. Click the Matte button.
3. Set values in the Width and Height fields below the Blur button.

These values specify the width and height of the Blur filter applied to the edge of the matte.

4. Enable the Blur button.



The matte before enabling the Blur filter



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

Blurring the Key-in Clip

You can apply a Blur filter to the key-in clip to create a softer matte.

To create a softer matte:

1. Click the Setup button.
2. Enable KeyIn Filter.
3. Set values in the Hor Blur and Vert Blur fields below the KeyIn Filter button.

These values specify the width and height of the Blur filter applied to the key-in clip.

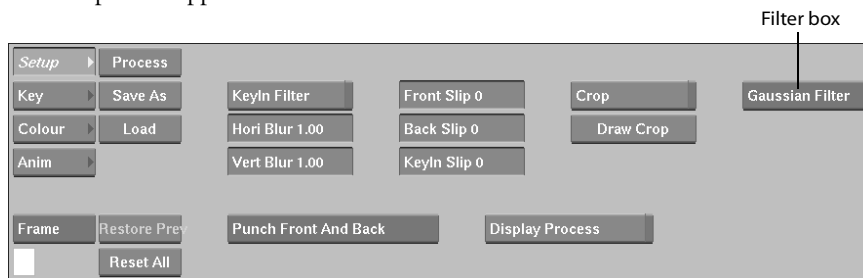
Selecting a Filter for the Blur Options

You can use either a Box filter or a Gaussian filter for the blur of the matte and key-in clip. The Gaussian filter gives sub-pixel resolution, allowing you to create more subtle effects.

To select a filter:

1. Click the Setup button.

The Setup menu appears.



2. Select either Gaussian filter or Box in the Filter box.

Inverting the Matte

You can invert the matte by enabling the Invert button in the Matte menu.

NOTE: Any Gain and Lift Level adjustments you make to the matte are not inverted when Invert is selected. You must therefore invert the matte before adjusting the Gain and Lift levels.

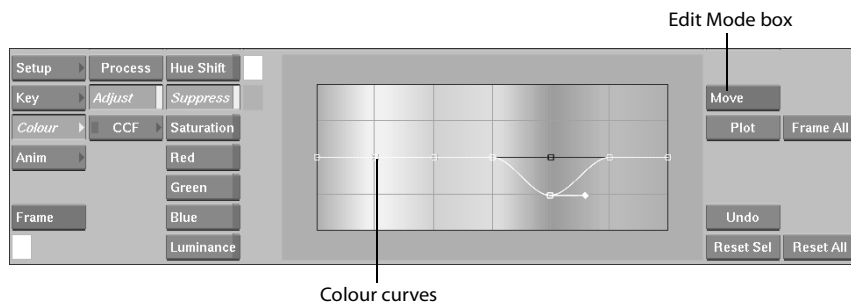
Removing Background Spill and Colour Correcting Your Clips

After you create a key and key out any trouble areas, you might find that some of the background colour has spilled over at the edge of the key. You can use the colour suppression tools in the Keyer to suppress the spill in the front clip. You can also access the Colour Corrector to colour correct the front clip.

To suppress colour spill:

1. Click the Colour button in the Keyer menu.

The Colour menu appears.



The hue spectrum and colour curves appear. There are colour curves for the primary colours, luminance, and saturation in the image.

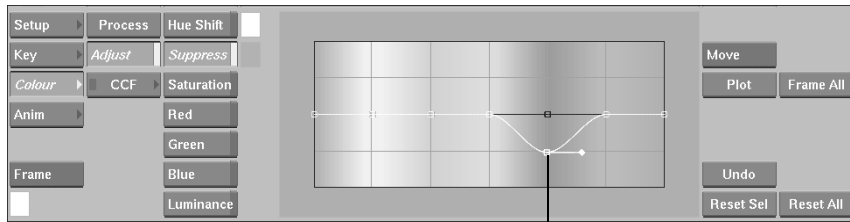
2. Enable Adjust to apply the colour curves of the hue spectrum to the front clip.
3. Click Result to view the result.
4. Click the Plot button.

The cursor turns into a colour picker.

5. Select a pixel within the spill.

A red vertical bar appears in the hue spectrum denoting the exact colour value of the selected pixel.

6. Edit the colour curves using the options in the Edit Mode box.



Suppressed colour curve

7. Click the Supp button to edit all the colour curves at the same time.

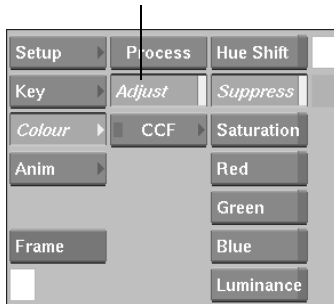
The colour box next to the Supp button displays the original colour extracted from the key-in clip.

8. Click the Red, Green, or Blue button to edit the individual colour curves.
9. Click the Sat button to adjust the saturation of the spill.

NOTE: Once you remove the saturation from a spill, you may want to increase the values for the other curves (i.e., red and green if you removed a blue spill) to reconstruct some of the natural colours at the edge of the keyed image.

10. Click the Lum button to adjust the luminance of the spill.

NOTE: To apply changes made in the colour menu, the Adjust button must be enabled.

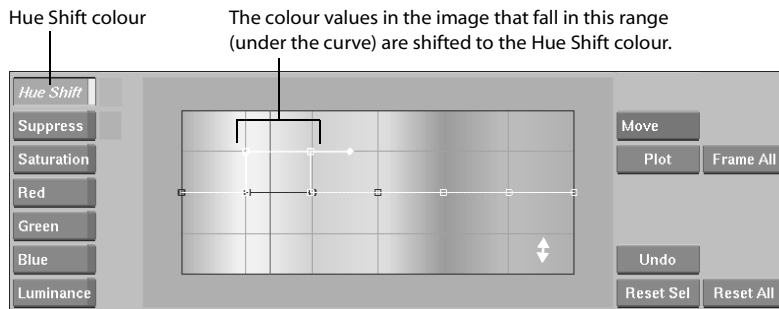


Hue Shift

Use the Hue Shift option in the Colour menu to select a colour from the front or back clip and create a hue shift between the selected colour and a range of colours. The hue shift is applied to the front clip that you define by editing the Hue Shift curve. This can be used to make a colour spill blend into a background colour that is not in the front clip.

To create a hue shift:

1. Click Result so that you can see the changes as you make them.
2. Click the Hue Shift button in the Colour menu.
3. Click the colour pot next to the Hue Shift button.
The colour picker appears.
4. Select or pick a colour.
5. Click again inside the colour pot next to the Hue Shift button to set the Hue Shift colour.
6. Use the Plot button to plot a pixel in the colour spill.
7. Specify the range of colours in the image that you want to shift to the selected Hue Shift colour by raising the Hue Shift curve to 75.00 for that range.

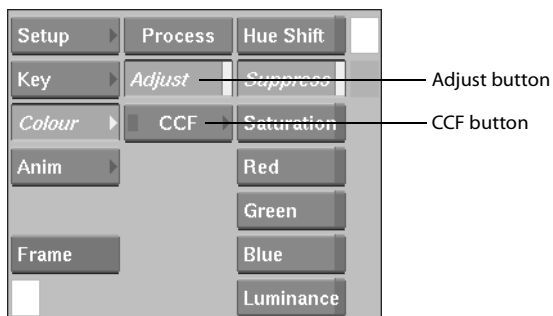


NOTE: To apply changes made in the Colour menu, the Adjust button must be enabled.

Colour Correcting the Front Clip

You can also use the Colour Corrector to correct the front clip. Click the CCF button in the Colour menu to bring the front clip into the Colour Corrector. The blue indicator on the CCF button is lit when a colour correction has been made to the front clip.

NOTE: To apply the colour correction setup to the front clip, the Front button must be enabled. If the Front and Adjust buttons are both enabled, the colour curves are applied first, and then the colour correction setup.



Animating Your Key

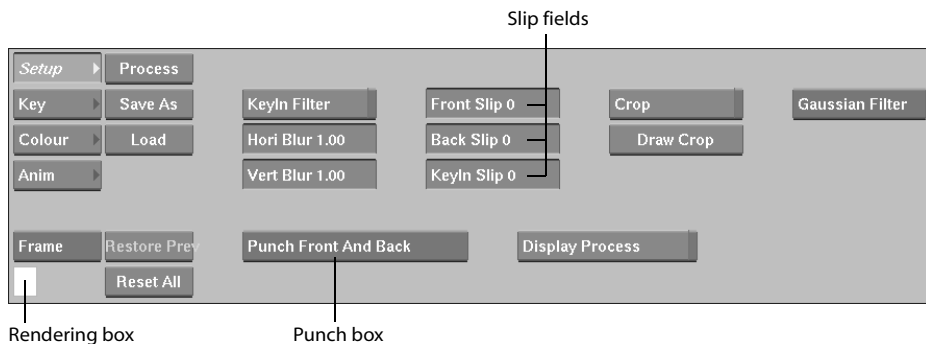
The following parameters in the key setup can be animated:

- The average colour selected using the Average Colour box in the Range menu
- The minimum and maximum Tolerance and Softness values
- The Lift, Gain, Shrink, and Erode values
- The blur factors for the matte and key-in clip
- The maximum and minimum input level and output level values for the histogram

To display the Animation menu in the Keyer, click the Anim button.

Processing Your Clips

Once you finish creating your composite clips, you can use the options in the Setup menu to improve the output and reduce processing time.



Field and Frame Rendering

You can render by field or by frame. If you select the Frame option in the Rendering box, the Keyer combines the two fields and renders each frame. If you select the Field option, the Keyer renders both fields separately. Use the Field option if the two fields in the image are not properly aligned and the filters you are using are not being applied to the proper pixels.

NOTE: The Field rendering option does not work if you select the Box filter in the Setup menu.

Punch Mattes Option

When you pull a key, a matte is created for both the front and back images. You can modify the luminance curves for both mattes using the Curves menu (see “Using the Luminance Curves” on page 644 for complete details). Using the Punch Mattes option, you can specify whether both mattes are used in the generation of the composite clip, or only the back matte.

- Punch Front and Back uses both the front and back mattes. This is appropriate for clips that do not contain a pre-multiplied alpha channel, such as clips originating from CG applications.

- **Punch Back Only** uses the back matte only. The luminance curve for the front matte is ignored, resulting in an additive key. Use **Punch Back Only** when you are using a CG clip with a pre-multiplied alpha channel in which the area to be keyed out is black. On these clips the front matte has been previously applied to the front image. If the front matte is applied a second time, an unwanted black fringe may result along the edges of the matte subject.

When the front matte is not used, the entire front image is used, and the curves calculation maps all the black areas of the front clip as transparent.

Displaying the Key while Processing

Enable the **Display Process** button to view the composite clip as it is being processed. When this button is disabled, the image window remains at frame 1 during processing. You can reduce processing time by disabling **Display Process**.

Slip Synching

You can offset the front, back, and key-in clips using the **Slip** fields in the **Setup** menu. You can delay the start of a clip or start anywhere in a clip; this way you do not have to exit the Keyer to edit the length or sequence of a clip.

Processing Options

You can generate either a composite clip or a matte clip.

To process the composite clip:

1. Click the **Result** button to display the composite clip.
2. Click the **Process** button.

To process the front matte:

1. Click the **matte** button to display the matte.
2. Click the **Process** button.

To process the back matte:

1. Click the **Bmatte** button in the **Curves** menu to display the back matte.
2. Click the **Process** button.

Keyer Hot Keys

NOTE: Press and hold the hot key as you carry out the action, unless otherwise indicated.

Press:	To:
T	Add to tolerance sample.
S	Add to softness sample.
P	Plot a pixel in the image.
F1	View the Front clip in the image window.
F2	View the Back clip in the image window.
F3	View the Matte clip in the image window.
F4	View the Result clip in the image window
ALT+ENTER	Play the clip.
ALT+HYPHEN (-)	Play the clip backwards.
SPACEBAR	Stop playing the clip.
ALT+END	Go to the end of the clip.
ALT+HOME	Go to the beginning of the clip.
ALT+LEFT ARROW	Go back one frame.
ALT+RIGHT ARROW	Go forward one frame.

The Modular Keyer provides all the tools found in the traditional Keyer, as well as other, more sophisticated tools that allow you to easily key much more challenging shots. You can specify the order in which you want processes to be applied to clips, giving you almost unlimited flexibility.

Summary

In this chapter, you learn about:

- “Accessing the Modular Keyer” on page 656
- “Animating the Shape Channel” on page 698
- “Setting Up the Processing Pipeline” on page 657
- “Nodes Placement Table” on page 699
- “Using Nodes” on page 676
- “Hot Keys” on page 700
- “Blend Nodes” on page 690

About the Modular Keyer

Use the Modular Keyer to pull a key and/or for selective colour correction of a clip. The processing pipeline provides an environment in which you can select and arrange the keying tools. You can select from an array of tools to create your key:

- The 3D Keyer, which provides automatic noise reduction, a 3D histogram to view the colours in a clip, and many methods to refine the key. See Chapter 35, “The 3D Keyer.”
- The 2D histogram to adjust the luminance of the matte.
- Garbage masks, including the Tracer, a tool you can use to key very difficult images. See Chapter 36, “Garbage Masks and the Tracer.”

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

Accessing the Modular Keyer

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

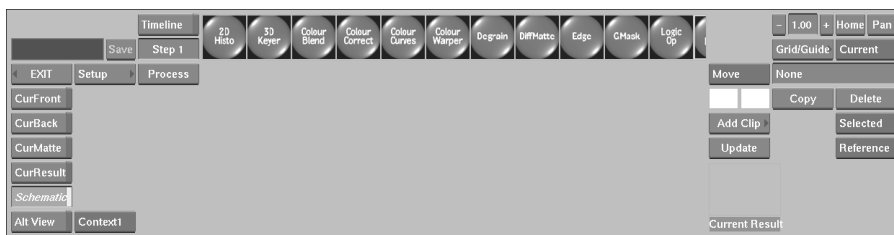
NOTE: For information on accessing the Modular Keyer from Action, see “Using Colour Correction and the Keyer in Action” on page 915.

To access the Modular Keyer:

1. Click the Modular Keyer button in the Effects menu.
2. Select the front, back, and key-in clips.
3. Select a destination.

The Modular Keyer menu appears in the lower part of the screen and the processing pipeline appears in the upper area.

Modular Keyer menu

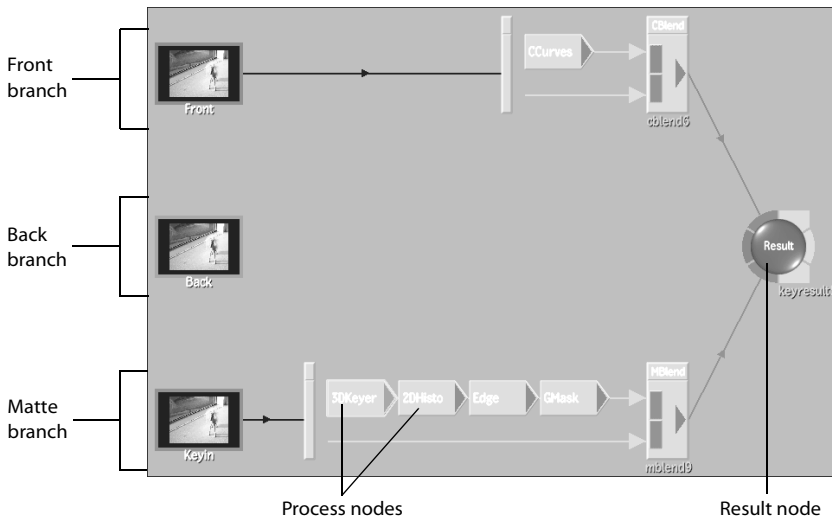


Setting Up the Processing Pipeline

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

The pipeline is made up of three processing branches—one each for the front, back, and matte branches.

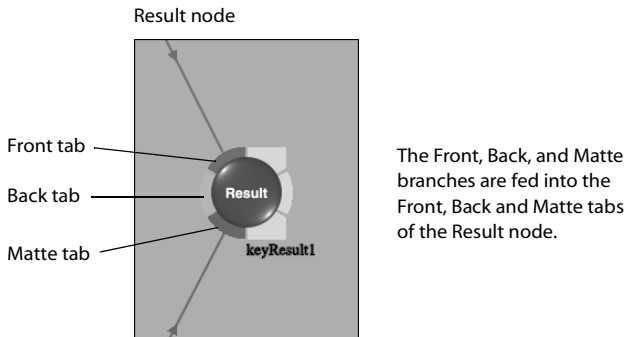
Default Processing Pipeline



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

The processing pipeline is similar to the process tree of the Batch module, except that its main purpose is to create a key. Unlike Batch, most of the nodes that you use in the processing pipeline have one input and one output. This allows for a more streamlined schematic. For more information on the Batch module, see Chapter 29, “Batch Processing.”

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



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

As you have observed, the default pipeline that you see when you first access the Modular Keyer contains certain nodes. These nodes correspond roughly to the functions of the traditional Keyer:

- Front branch:
 - The CCurves node opens the Colour menu, where you can remove colour spill from the front clip, perform a hue shift, and so on.
- Matte branch:
 - The 3D Keyer node opens the 3D Keyer, where you pull the key.
 - The 2D Histogram node opens the histogram, where you can adjust the luminance of the matte.
 - The Edge node opens the Edge menu, where you can adjust the edges of the key using shrink, erode, blur, and so on.
 - The GMask node opens the garbage mask menu, for creating garbage masks.
- Result node: Opens the Matte Curves menu so that you can adjust the front and back mattes.

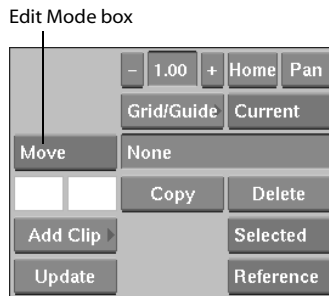
If the node for the operation you want to perform on a clip is not already on the pipeline, you add it to the appropriate branch. For example, if you want to blur the front clip, you add an RGB Blur node to the Front branch.

Displaying Menus

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

To access a menu for a node:

1. Select Move from the Edit Mode box (or press **M**).



2. Click on the node.

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

To access the Gmask and Colour Correct menus:

1. Select Move from the Edit Mode box (or press **M**).
2. Click on the node.
3. Click the Edit button.

The module opens.

NOTE: You can also double-click the GMask or CC node to access the module.

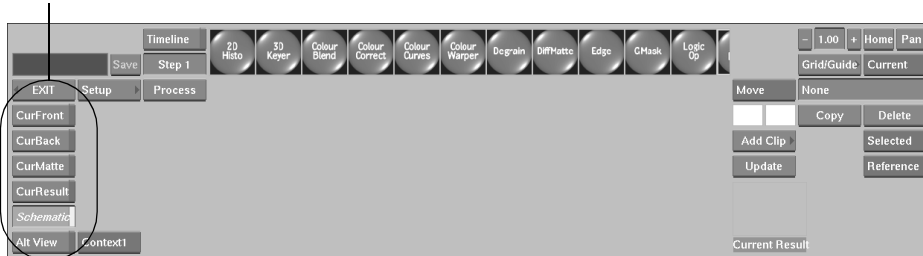
Modular Keyer Menu

The Modular Keyer menu provides the commands you need to:

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

Viewing Clips in the Image Window

Use tilde (~) to switch from Schematic view (pipeline) to the image window, and vice versa. You can use these buttons, or the equivalent hot keys, to change the image that appears in the image window.

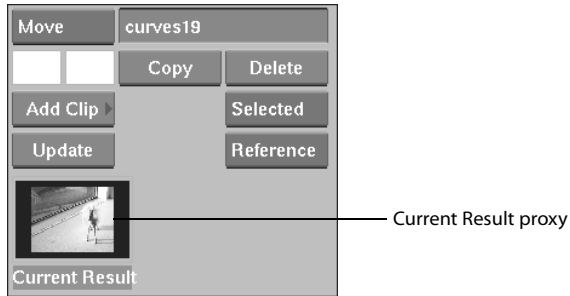


Select:	To View:	Hot Key
CurFront	The front clip.	F1
CurBack	The back clip.	F2
CurMatte	The matte (the input to the Matte tab of the Result node).	F3
CurResult	The result (the output of the Result node).	F4
Alt View	The view selected in the Alt View option box (see table that follows).	
Schematic	The processing pipeline. The ~ hot key toggles between the schematic (the pipeline) and the last selected view.	~

Using the Alt View option box, you can display the following alternate views in the image window:

Select:	To View:	Hot Key
Result	The result of the changes made in the selected node.	2
Context1	Context Point 1 (see “Setting a Context Point” on page 672 for more information).	3
Context2	Context Point 2 (see “Setting a Context Point” on page 672 for more information).	4
Action	Action context point (this is only available when accessing the Modular Keyer from Action).	5

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



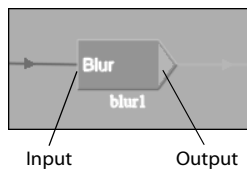
About Nodes

There are three types of processing nodes you can use on the pipeline:

- Single input nodes
- Multiple input nodes
- Blend nodes

Single Input Nodes

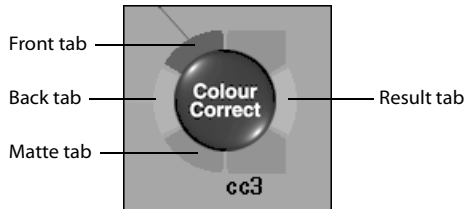
The simplest type of node has a single input and output:



Examples of this type of node are the 3DKeyer, Edge and RGB Blur nodes.

Multiple Input Nodes

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



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

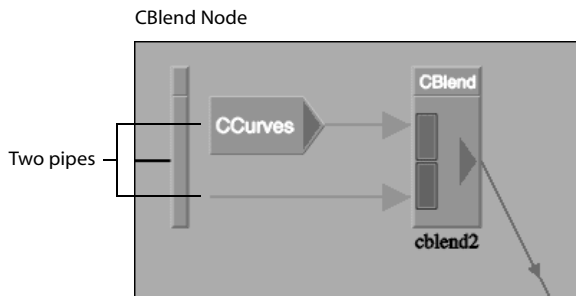
The tabs that are available depend on the node. If the node accepts a front, back, and matte clip, all source tabs are available. If the node only accepts a front and back clip, the red and green source tabs are available and the blue source tab is grey. The result tab is always available, except on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

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

Examples of multiple input nodes are the Result, Colour Correct and Regrain nodes.

Blend Nodes

The Front and Matte branches each contain a special node that includes two lines containing other nodes—the CBlend and MBlend nodes. The nodes in these two lines are processed sequentially, as in a branch. Because these lines function somewhat differently from a branch, they are referred to as *pipes*. The two pipes in the following example constitute a layer within the CBlend node.



Use the nodes on the CBlend pipes to apply colour corrections and other processes to clips. Use the nodes on the MBlend pipes to create a matte for the composite.

The two pipes of blend nodes provide extra functionality compared to branches in the pipeline:

- You can apply the processes to a selected region of the image. You use the lower of the two pipes to define the area on the image where you want to apply the processes used in the upper pipe. For example, you could add a garbage mask node to the lower branch and create a matte. The processes in the upper pipe will only be applied to the foreground subject of the matte. If there are no nodes in the lower pipe, the processes in the upper pipe are applied to the entire image.
- You can blend the two pipes in various ways.
- You can use multiple layers. The two pipes together constitute a layer. You can add one to three additional layers to blend nodes.

For more information, see “Blend Nodes” on page 690.

Working with Nodes

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

For information on individual nodes and where on the pipeline they may be used, see “Using Nodes” on page 676 and “Nodes Placement Table” on page 699.

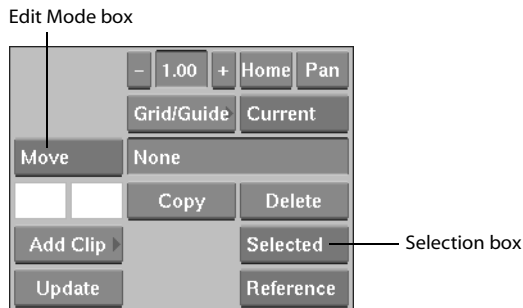
For examples of useful pipeline configurations for solving common difficulties when keying, see the Modular Keyer exercises in the *flame Tutorial*.

Selecting Nodes on the Pipeline

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

To select nodes in the schematic:

1. Select Move mode from the Edit mode box (or press **M**).



2. Select the node(s) as follows.

To Select:	Do this:
One node	Choose Selected from the Selection box and click on the node.
A branch	Choose Branch from the Selection box and click anywhere on the branch.
All nodes on pipeline	Choose Graph from the Selection box and click anywhere on the pipeline.
All nodes in schematic	Choose All from the Selection box and click on any node.
A combination of nodes	Press CTRL and drag a rectangle around the nodes. The current option in the Selection box does not affect this type of selection.

Selected nodes have a white border. Note that this is different from the yellow border, which indicates both that a node is selected and that its menu is the active menu.

To deselect a selection, click anywhere in the grey area of the schematic.

Adding Nodes to the Pipeline

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

Most nodes can be placed just about anywhere on the pipeline. There are a few exceptions when it comes to the pipes of blend nodes. Certain nodes do not function on some pipes, because of the way the pipes operate. For more information, see “Blend Nodes” on page 690.

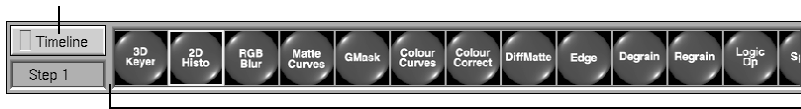
The method for adding nodes to blend node pipes differs from method for adding them to branches:

- When you add a node to the CBlend and MBlend pipes, you can simply drag and drop them onto the pipe and they are “attached”.
- When you add a node to branches of the pipeline (for example, to the Back branch), you need to cut the branch at the appropriate location, connect it to the new node, and reconnect the branch to the rest of the pipeline.

To add a node to the CBlend and MBlend pipes:

1. Display the Node bar by disabling the Timeline button (or swiping the cursor across the Swipe bar at the left of the menu).

Timeline button



Node bar

2. If needed, scroll the Node bar to display the node you want, by clicking on any node and dragging to the left or right.

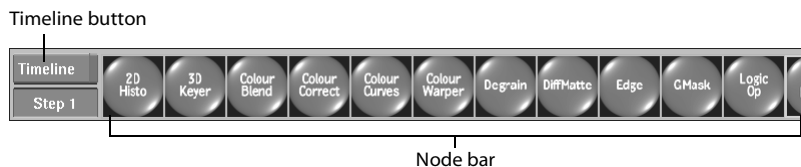
3. Click on the node and drag the cursor to the schematic.
The cursor changes to an arrow-headed cross.
4. Release the cursor.
The node appears in the schematic.
5. To use the node, it must be connected to the pipeline. Drag the node over one of the blend pipes and release the cursor.
To place the node between two nodes, position it between the two nodes such that it is touching both nodes. To place the node at the beginning of a pipe, overlap it with the first node and move it back a bit. Similarly, to place a node at the end of a pipe, overlap it with the last node, move it ahead a bit, and release the cursor.

The node is added to the pipeline.

NOTE: If the node is not added to the pipeline when you release the cursor, it is possible that the node cannot be added there. For example, you cannot add a Colour Curves node to the lower CBlend pipe. To learn more about the types of nodes you can add to the CBlend and MBlend pipes, see “Blend Nodes” on page 690 and “Nodes Placement Table” on page 699.

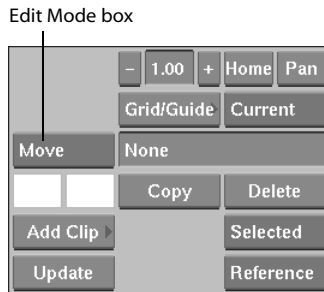
To add a node to branches:

1. Display the Node bar by disabling the Timeline button.



2. If needed, scroll the Node bar to display the node you want, by clicking on any node and dragging to the left or right.
3. Click on the node and drag the cursor to the schematic.
The cursor changes to an arrow-headed cross.
4. Release the cursor.
The node appears in the schematic.
5. Drag the node closer to the location on the branch where you want to add it.

6. Change to Parent mode by selecting Parent from the Edit Mode box (or press **P**).



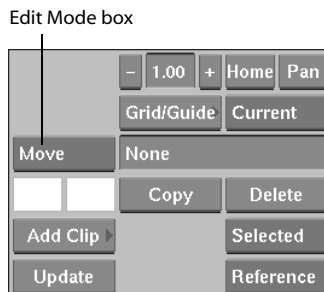
7. Drag the cursor across the branch where you want to add the node.
The portion of the branch between the previous and next items disappears.
8. Drag the cursor from the front of the previous clip or node (or its Result tab) to the back of the new node.
If the node has source tabs, attach the branch to the front, back, or matte tab, depending on the type of node. For complete information on using particular types of nodes, see “Using Nodes” on page 676.
NOTE: You can also use the output (result image) of any node on the pipeline as an input source for the new node. This includes the output of nodes along the pipes of blend nodes.
The branch is connected to the new node.
9. Drag the cursor from the result tab of the new node to the back of the next node (or to the appropriate source tab—see note, above).
10. Return to Move mode by selecting Move from the Edit Mode box (or press **M**).

Moving Nodes

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

To move a node:

1. Select Move mode from the Edit Mode box (or press **M**).



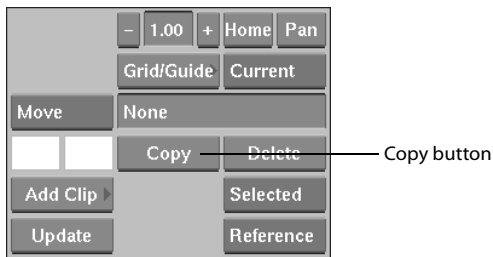
2. Detach the node from the pipeline:
 - If it is on a blend node pipe, **CTRL+ALT**-click on the node. It is detached from the pipe, and is selected (shown by its yellow border).
 - If it is on a branch, select Parent from the Edit Mode box and drag the cursor across the branches that connect it to the pipeline. Reconnect the branch without the node, then return to Move mode.
3. Drag the node over the pipeline where you want to add it and release the cursor. See “Adding Nodes to the Pipeline” on page 664 for details.

Copying Nodes

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

To copy one or more nodes:

1. Select the node(s). See “Selecting Nodes on the Pipeline” on page 663 for details.
2. Click the Copy button.



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

3. Drag the copies over the pipeline where you want to add them and release the cursor. See “Adding Nodes to the Pipeline” on page 664 for details.

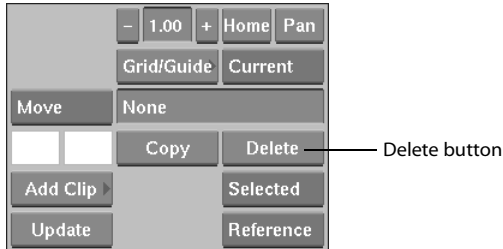
Deleting Nodes

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

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

To select and delete one or more nodes:

1. Select the node(s). See “Selecting Nodes on the Pipeline” on page 663 for details.
2. Click the Delete button.



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

To use Delete mode:

1. Select Delete from the Edit Mode box (or press **D**).
2. Click on the node you want to delete.
3. Click Confirm to delete the node, or click elsewhere to cancel.

HINT: To override Confirm, press **ALT** as you click on the node.

4. Repeat steps 2-3 for any other nodes you want to delete.
5. Return to Move edit mode.

To delete by dragging items off the desktop:

1. In the Edit Mode box, select Move.
2. In the Selection box, select: Current, Branch, Graph, or All.
3. In the schematic, click a node and drag either to the top or the bottom of the desktop.
The cursor changes to a green recycling icon.
4. Click Confirm to delete the node(s), or click elsewhere to cancel.

Updating Nodes

The nodes in the pipeline are not updated, or processed, whenever a change is made to the pipeline. This is to avoid using up time on processing them if it is not needed. For example, when you make a change in a node, that node and all previous nodes in the branch are updated immediately, but not the subsequent nodes.

In the pipeline, the status of a node is indicated differently depending on the type of node.

Node Type	Unprocessed	Processed
Single Input	The arrowhead at the right side of the node is red.	The arrowhead at the right side of the node is green.
Multiple Input	The label under the node is black.	The label under the node is white.
Blend	The arrowhead is red and the label is black.	The arrowhead is green and the label is white.

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

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

Use the Auto Update option in the Setup menu to set the update mode.

To set the update mode:

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

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

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

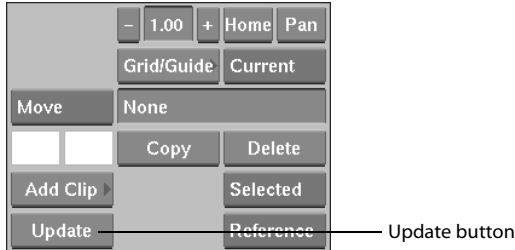
To update a node automatically:

1. Select Move edit mode.
2. Click on the node.

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

To update a node manually:

1. Select the node (see “Selecting Nodes on the Pipeline” on page 663).
2. Click the Update button (or press **U**).

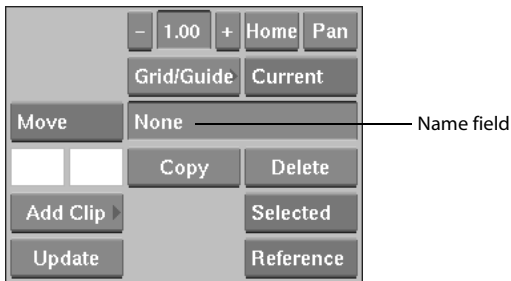


The selected node and all previous nodes are updated.

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

Renaming Nodes and Clips

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



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

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

To rename a node or clip:

1. Select the node or clip.
2. Click the Name field.
The on-screen keyboard appears.
3. Click Esc to delete the previous name and type a new name.
4. Click Enter to save the new name or click EXIT Keyboard to cancel.

Adding Clips to the Pipeline

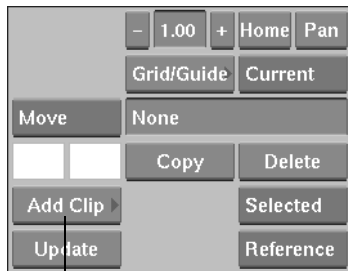
You can add clips to the pipeline to use with nodes that require extra clips. For example, when you use the Difference Matte node, you need two clips.

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

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

To add a clip to the schematic:

1. Click the Add Clip button.



Add Clip button

You are brought back to the desktop.

2. Select the clip you want to bring into the schematic by clicking its upper left corner.
You are returned to the Modular Keyer and the clip appears in the schematic.

To attach the new clip to a node:

1. Select Move from the Edit Mode box (or press **M**).
2. Move the clip to the left of the node you want to attach it to.
3. Select Parent from the Edit Mode box (or press **P**).
4. Drag the cursor from the right edge of the new clip to the appropriate source tab of the node.
If it is a single input node, drag the cursor to the left side of the node.
The clip is connected to the node.
5. Return to Move edit mode.

NOTE: For complete information on using particular types of nodes, see “Using Nodes” on page 676.

To replace an existing clip:

1. Select Move from the Edit Mode box (or press **M**).
2. Move the clip near the existing clip.
3. Select Parent from the Edit Mode box (or press **P**).
4. Detach the old clip from the pipeline by dragging the cursor across the part of the pipeline between the old clip and the rest of the pipeline.
5. Connect the new clip to the pipeline by dragging the cursor from its right edge to the appropriate source tab of the first node on the branch.
6. Return to Move edit mode.

To replace a clip that was added to the pipeline:

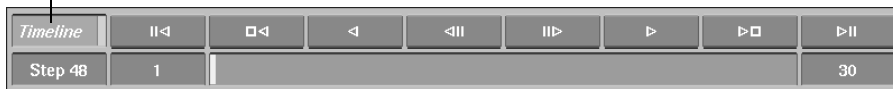
1. Double-click on the clip.
You are brought back to the desktop.
2. Select another clip.
It replaces the previous clip in the pipeline.

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

Playing Clips

To display the timeline and play controls, enable the Timeline button.

Timeline button



The node bar is replaced by the timeline.

Use the play controls and timeline to move to a different frame, scroll through the clip, and play the clip.

Setting a Context Point

As you work with different nodes to create your key, it will often be useful to see the effect of your changes on the image at a later part of the pipeline. You do this using context points. You set a context point on a particular node, then view that context point from the node you are working in. For example, you could place a Degrain node at the beginning of the Matte branch and set a context point on the 3D Keyer node. You could then degrain the clip while viewing the context point, in this case, the Current Result of the 3D Keyer.

NOTE: To view context point in the above example, you must have pulled a key with the 3D Keyer.

You can set two different context points in the pipeline.

To set context points:

1. Press and hold = and click on the node.

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



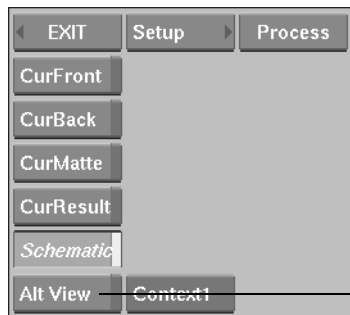
Context Point 1 set on GMask node

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

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

To view a context point:

1. Set a context point, as described above.
2. Select the node you want to work with.
3. Select Context1 or Context2 from the Alt View option box, according to the context point you want to see. Alternatively, press **3** for Context Point 1 or **4** for Context Point 2.



Alt View option box

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

Comparing Colours on Different Images

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

You can also apply a colour that you copied to one of the colour boxes to a colour box in one of the node menus. For example, apply the colour to the Suppress colour box in the Colour Curves menu.

To apply a colour to a colour box:

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

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

1. Open the node menu containing the colour box you want to copy a colour to.
2. Click in the colour box in the node menu.
The colour picker appears.
3. Click the Pick button, then click in the Modular Keyer colour box containing the sampled colour.
4. Click in the node menu colour box again.

For more information on using the colour picker, see “The Colour Picker” on page 57.

Panning the Schematic

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

To pan the schematic:

1. Click the Pan button, or press and hold the **SPACEBAR**.
2. Drag anywhere in the schematic.
Notice that the schematic area also extends “underneath” the menu area.

Cropping the Key Area

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

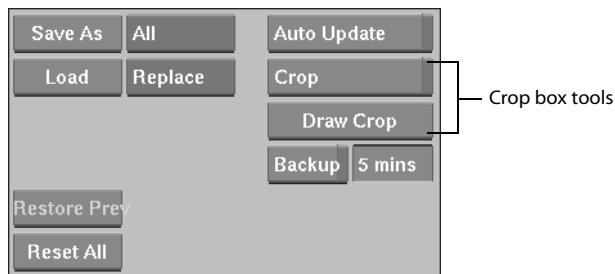
Using a crop box as you create the key speeds up interactivity, which is especially useful when working at high resolutions. You can also process the composite with a crop box. By setting the crop box to just surround the foreground subject, you can save processing time. Everything outside the crop box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

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

To use the crop box:

1. Press ~ to display the image window.
2. Click the Setup button.

The Setup menu appears.



3. Click the Crop button to enable the crop box.
4. Click the Draw Crop button.
The button turns white.
5. Draw the crop box on the image by selecting a point on the image and dragging the cursor diagonally.

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

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

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

Backing Up the Key Setup

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

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

Saving the Key Setup

In the Modular Keyer, you can save a key setup that includes all the settings in all nodes used to create the key. You can save part of a setup and reload it, load and save nodes separately, and save All and Selected (proxies are not saved when using All or Selected). You can load setups using Replace and Append.

To save the key setup:

1. Click the Setup button in the Modular Keyer menu.
The Setup menu appears.
2. Use the Save As button to save the setup, and the Load button to load an existing setup. For information, see Chapter 7, “Saving Setups and Preferences.”

Using Nodes

This section describes the nodes you can use to enhance your key, with three exceptions—the 3D Keyer, Gmask, and blend nodes.

To learn about:	See:
The 3D Keyer	Chapter 35, “The 3D Keyer.”
Garbage masks	Chapter 36, “Garbage Masks and the Tracer.”
Blend nodes	“Blend Nodes” on page 690.

Before reading this section, you should be familiar with the basic node operations described in “Working with Nodes” on page 663.

Node bar

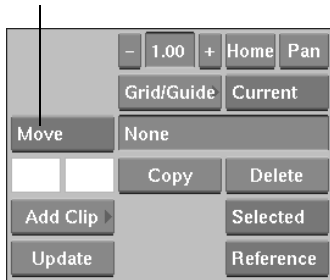


2D Histogram Node

Use the 2D histogram to adjust the luminance of the matte. You can add this node to any part of the pipeline except the Front pipe of the CBlend node.

To adjust the matte with the 2D histogram:

1. Select Move mode.



2. Select the 2D histogram node.

The histogram appears.

3. Press ~ to view the image.

The histogram is identical to the histogram in the traditional Keyer.

RGB Blur Node

Use the RGB Blur node to apply a Gaussian blur to colour images. You can add an RGB Blur node to any part of the pipeline except to the Front and Matte pipes of MBlend nodes and the Matte pipes of CBlend nodes. To blur a matte, use the Blur filter in the Edge node. Both the RGB and Edge node blurs are Gaussian.

NOTE: Using this node right before the MBlend node on the Matte branch is the equivalent of the KeyIn Filter in the traditional Keyer.

You can animate a blur using the Channel Editor.

For information on using the Frame/Field box, see “Using Field mode” on page 681.

To blur a colour image:

1. Place the RGB Blur node at the appropriate point in the pipeline (see “Adding Nodes to the Pipeline” on page 664).
2. In Move mode, click on the node to view the menu.



3. If you want the image to be equally blurred horizontally and vertically, click P to enable a proportional blur.
4. Set the width and height of the blur. When P is enabled, enter a value in either field. The higher the number that you enter, the greater the blur.

NOTE: Increasing the blur increases the processing time.

Matte Curves Node

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

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

You can add a Matte Curves node to any part of the pipeline except to the CBlend and MBlend pipes. This is because each layer of the blend nodes has its own Matte Curves menu built into it. To learn about adjusting the front and back mattes of blend node layers, see “Blending the Front and Back Images of a Layer” on page 693.

See “Result Node” on page 686 for instructions on using the Matte Curves menu.

Colour Curves Node

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

You can add Colour Curves nodes to any part of the pipeline except to the Front and Matte pipes of MBlend nodes and the Matte pipes of CBlend nodes. This is because these pipes are designed to produce a matte, and the result of the Colour Curves node is a colour image.

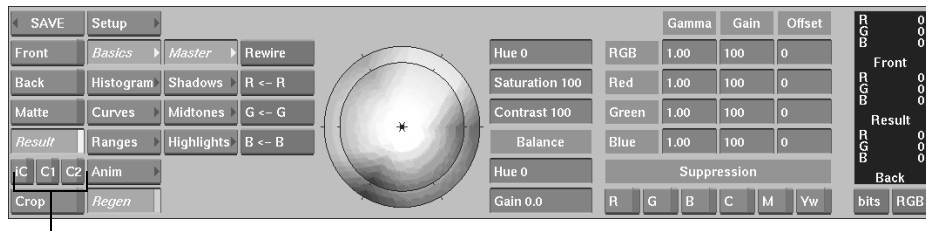
The Colour Curves menu is identical to the Colour menu in the traditional Keyer. See “Removing Background Spill and Colour Correcting Your Clips” on page 649 for complete information on using this menu.

Colour Correct Node

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

You can add Colour Correct nodes to any part of the pipeline except to the Front and Matte pipes of MBlend nodes and the Matte pipes of CBlend nodes. This is because these pipes are designed to produce a matte, and the result of the Colour Correct node is a colour image.

When using the Colour Corrector from the Modular Keyer, you will notice additional buttons in the menu: C1, C2 and iC.



Context Point buttons

The C1 and C2 buttons allow you to view Context Point 1 and Context Point 2 while colour correcting the clip. A third button, iC is visible when you access the Modular Keyer from Action. Pressing this button shows the Action result. For information on setting and using context points, see “Setting a Context Point” on page 672.

You can go directly to the Load Setup Browser by holding **ALT** when you drag a node to the desktop.

To use the Colour Correct node:

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

NOTE: When you add the Colour Correct node to the Front pipe of the CBlend node, the region to which the colour correction is applied is defined by the Matte pipe. See “Blend Nodes” on page 662 for more information.

2. Click Colour Correct node, then click the Edit button. Alternatively, double-click the node. The Colour Corrector opens. For complete information on using the Colour Corrector, see Chapter 26, “Colour Corrector.”

Colour Warper Node

The Colour Warper node accepts a front, back, and key-in clip as input. Use the Colour Warper node to access the Colour Warper module for colour correcting clips and refining mattes. For more information on using the Colour Warper, see Chapter 27, “Colour Warper.”

Difference Matte Node

Use the Difference Matte node to create a matte based on the differences between two clips. You can generate a matte from two clips with the same background but different foreground elements. For example, the first clip could be the blue or green screen shot you want to key, and the second clip could be a “clean plate”—a shot of the blue or green screen with no foreground subject.

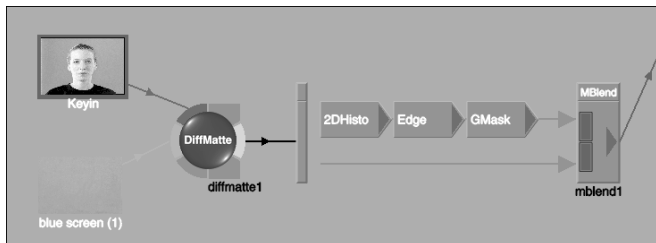
This node functions in the same way as the Difference command in the Processing menu, except that in the Modular Keyer you do not have the Front/Back/Comp option, only Front/Back. For more on how Difference Matte works, see “Using the Difference Command” on page 498.

You can add a Difference Matte node to any part of the pipeline except to the blend node pipes. This is because the nodes in the CBlend and MBlend pipes must have a single input only, and the Difference Matte node always requires two input sources. Typically, you would add it to the beginning of the Matte branch. You can then apply the 2D histogram, Edge tools, garbage mask, and so on, to the resulting matte.

NOTE: Since the result of this node is a matte, you would not use the 3D Keyer node in the Matte branch, as the 3D Keyer node uses a key-in clip, not a matte, as its input.

To use the Difference Matte node:

1. If needed, add the clip you want to use as the back clip to the schematic (see “Adding Clips to the Pipeline” on page 671).
2. Add a Difference Matte node to the schematic (see “Adding Nodes to the Pipeline” on page 664).
3. Attach a source to the Front tab and a source to the Back tab of the node.
4. Attach the output of the node to the pipeline at the appropriate position. For example, you could set up the node as follows.



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

For complete instructions on using this menu, see “Using the Difference Command” on page 498.

Edge Node

The Edge menu provides controls to:

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

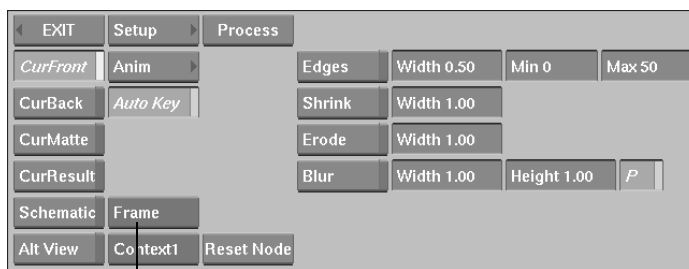
You can add the Edge node to any part of the pipeline except the Front pipe of the CBlend node. The Edge node is not appropriate for this pipe because its result is a matte.

Using Field mode

Video clips containing fast-moving objects can contain artifacts characterized by edge misalignments. These artifacts occur when the odd and even fields are combined into frames. Applying Erode, Blur, and Shrink to images containing misalignments does not produce a good result. To solve this problem, use Field mode. When you use Field mode, frames are separated into fields and the Edge effect(s) being used are applied to individual fields. The fields are then recombined into frames. This is all done “behind the scenes”—that is, fields are not displayed in the image window.

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

Edge menu



Field/Frame button

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

Use: To:

Edges	Create an image composed of the edges found in an image. The resulting greyscale image may be used as a matte or to produce special effects. You can apply the Edges tool to a colour or monochrome clip. This tool has many applications; among the most useful are: <ul style="list-style-type: none"> • To delimit a portion of the matte to perform colour corrections on the front clip. • Use directly on the front or back clip to create special effects.
Shrink	Remove pixels from the edge of the matte. It should not be used when the object in the front clip has soft edges such as hair. A negative value expands the matte.
Erode	Blend the light and dark edges of the matte.
Blur	Apply a softening Gaussian blur filter to the edge of the matte.

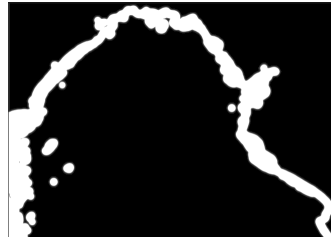
To transform an image into edges:

1. Select CurResult view from the Alt View option box.
2. Enable the Edges button.
3. Set a value for the edge-detection sensitivity in the Width field to the right of the Edge button. This control affects how the edge-detection algorithm determines whether or not each pixel forms part of the edge.
4. Set the Minimum Input Level in the Min field.
The Minimum Input Level sets the start of the range of luminance values in the image. Pixels with luminance values below the Minimum Input Level are mapped to black (0).
5. Set the Maximum Input Level in the Max field.
The Maximum Input Level sets the end of the range of luminance values in the image. Pixels with luminance values greater than the Maximum Input Level are mapped to white (255 in 8 bit mode and 4095 in 12 bit mode).
6. Enable the Shrink button.

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



The edge before adjusting the width using the Shrink Width field



The edge after adjusting the width using the Shrink Width field

To shrink the edge of the matte:

1. Select CurResult view from the Alt View option box.
2. Enable the Shrink button.
3. Set a value in the Width field next to the Shrink button.

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



The matte before applying the Shrink filter

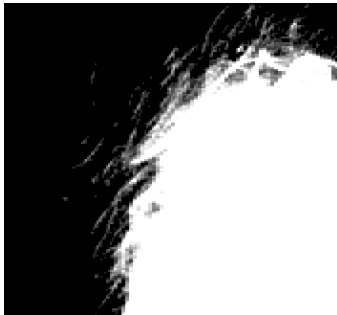


The matte after applying a shrink width value of 1.00

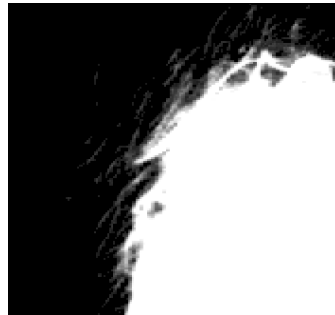
To erode the edge of the matte:

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

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



The matte before applying the erode filter



The matte after setting the erode width value to 1.00

To blur the edge of the matte:

1. Select CurResult view from the Alt View option box.
2. Enable the Blur button.
3. If you want the image to be equally blurred horizontally and vertically, click P to enable a proportional blur.
4. Set the width and height of the blur. When P is enabled, enter a value in either field. The higher the number, the greater the blur.



The matte before applying a blur



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

Degrain and Regrain Nodes

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

- You can add the Degrain node to any part of the processing pipeline.
- You can add Regrain to any part of the pipeline except to the Front and Matte pipes of MBlend nodes and the Matte pipes of CBlend nodes. This is because these pipes are designed to produce a matte, and the result of the Regrain node is a colour image.
- You can go directly to the Load Setup Browser by holding **ALT** when you drag a node to the desktop.

The Degrain and Regrain menus in the Modular Keyer contain exactly the same set of controls as the Degrain and Regrain menus accessed from the Processing menu on the desktop (only the menu layout is slightly different). For full instructions on using Degrain and Regrain, please see “Degraining and Regraining Clips” on page 504.

Logic Op Node

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

This node functions in exactly the same way as the Logic Ops command in the Processing menu. For more on how Logic Op works, see “Using Logical Operations” on page 495.

You can add a Logic Op node to any part of the pipeline except to the blend node pipes. This is because the nodes in the CBlend and MBlend pipes must have a single input only, and the Logic Op node always requires two input sources.

To use the Logic Op node:

1. If needed, add a second clip to the schematic (see “Adding Clips to the Pipeline” on page 671).
2. Add a Logic Op node to the schematic (see “Adding Nodes to the Pipeline” on page 664).
3. Attach a source to the Front tab and a source to the Back tab of the node.
4. Attach the output of the node to the pipeline at the appropriate position.
5. Click the node to view the Logic Op menu.

For complete instructions on using this menu, see “Using Logical Operations” on page 495.

LUT Editor Node

When you import or export Cineon or DPX files, a LUT Editor node is appended to the Import or Export node. A LUT converts a logarithmic image to a linear image, or linear images to logarithmic images, while maintaining colour accuracy.

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

For more information on the LUT Editor, see “Using a Lookup Table” on page 330.

Monochrome Node

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

Negative Node

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

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

To use the Negative node:

1. Add a Negative node to the schematic.
2. Switch to Parent edit mode.
3. Connect a source to its Front tab.
4. Connect the output to the input tab of the next node on the branch.

Result Node

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

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

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

Adjust the Front and Matte Luminance Curves

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

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

To learn how to set the matte curves to create an additive key, see “Creating an Additive Key for CG Images” on page 688.

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

$$\text{Result} = F * \text{FrontLUT} + B * \text{BackLUT}$$

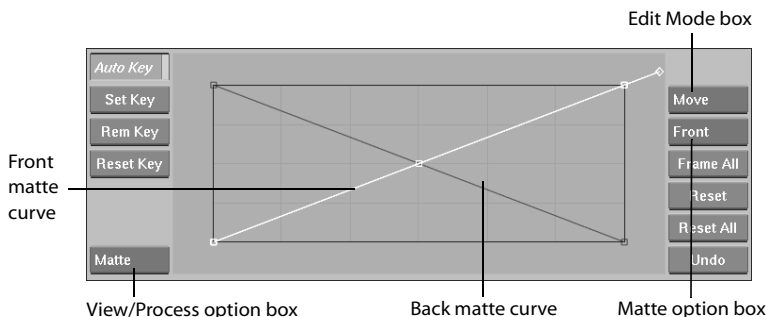
where:

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

To adjust the luminance curves:

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

The Matte Curves menu appears.

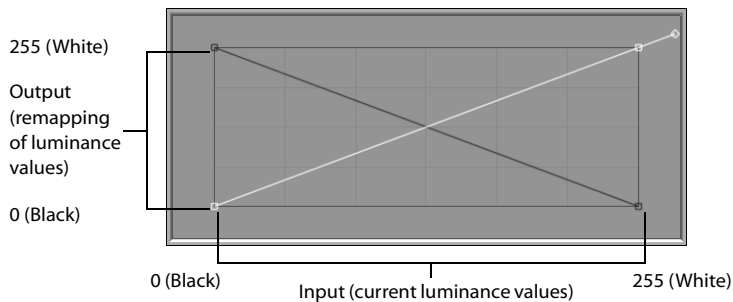


2. Select CurResult view from the Alt View option box (or press **2**). This allows you to use the View/Process option box to view a particular image as you adjust the curve.

3. From the View/Process option box, select the image you want to view as you adjust the curves.

Select:	To:
Result	View the composite clip.
Matte	View the front matte. You cannot see changes made to the back matte when this view is selected.
Bmatte	View the back matte. You cannot see changes made to the front matte when this view is selected.
Comp	View the composite with a coloured background. The default colour is white. To select a different colour, click on the colour swatch to the left of the Result option box. The colour picker appears. For information on using the colour picker, see "The Colour Picker" on page 57.

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



In Move edit mode, click on a point to display its tangent handle and drag the handle to adjust the curve. Use other modes in the Edit Mode box (Add, Delete, Break and so on) to further adjust the curve, adding or deleting points, or breaking tangent handles as needed. For complete information on using edit modes, see "Editing Animation Curves and Keyframes" on page 134.

Creating an Additive Key for CG Images

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

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

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

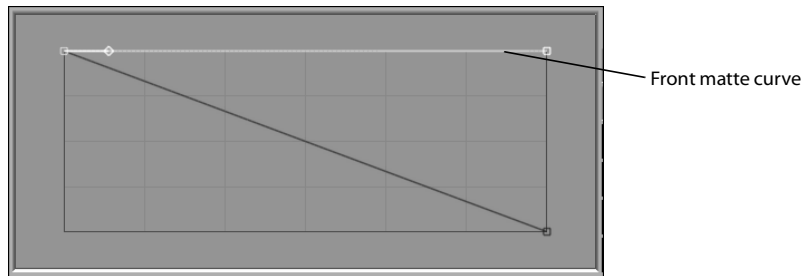
To create an additive key:

1. Pull the key and perfect it.

2. Select the Result node.

The Matte Curves menu appears.

3. Select the front matte curve from the Matte option box.
4. Drag the control point at the beginning of the curve from the lower left corner of the graph to the upper left corner of the graph.



This maps all pixels in the front matte to white (255 in 8-bit mode, 4095 in 12-bit mode), which has the result of using the entire front clip. The black pixels on the front clip are calculated as transparent.

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

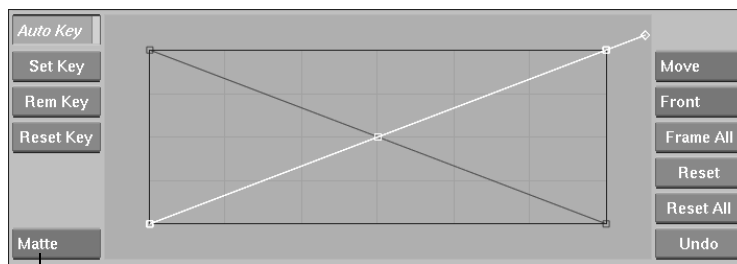
Processing Options

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

To process the composite:

1. Display the pipeline and select the Result node.

The Matte Curves menu appears.



View/Process option box

2. Select the type of clip you want to process in the Result option box.

Select:	To:
Result	Process the composite clip.
Matte	Process the front matte.
Bmatte	Process the back matte.
Comp	Process the composite with a coloured background. The default colour is white. To select a different colour, click on the colour swatch to the left of the Result option box. The colour picker appears. For information on using the colour picker, see “The Colour Picker” on page 57.

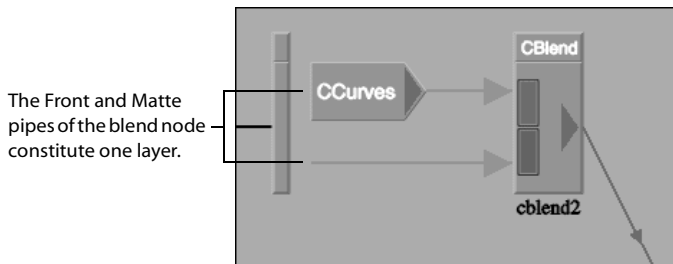
3. Click the Process button in the Modular Keyer menu.

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

Blend Nodes

Blend nodes include the CBlend (Colour Blend) and MBlend (Matte Blend) nodes.

A blend node is essentially a kind of container that you can use to create a “mini-composite” at different parts of the pipeline. CBlend and MBlend nodes each comprise one or more layers that include the components needed for a composite: a front, matte, and a back component. These components are referred to as *pipes*, and they may contain as many nodes as needed to create the desired image.



The back component is not represented visually as it is applied automatically to the composite.

Nodes on the pipes are processed sequentially, with a single input and single output to and from each node. Therefore, you cannot add multiple input nodes that require several clips (such as the Logic Op node) to pipes. However, some multiple input nodes can operate with only one input source. These are the Gmask, Colour Correct, Degrain, and Regrain nodes. If you add one of these node types to a blend node pipe, it is “converted” to a single-input node.

You can animate the curves in the MatteCurves, CBlend, MBlend, Colour Curves, or Result nodes. For more information about animation, see Chapter 9, “Animation.”

Use the CBlend node to create a composite that results in a colour image, and the MBlend node to create a composite that results in a matte.

CBlend Node

Use CBlend nodes to set up the colour operations you want to perform on a clip.

To do so, you can:

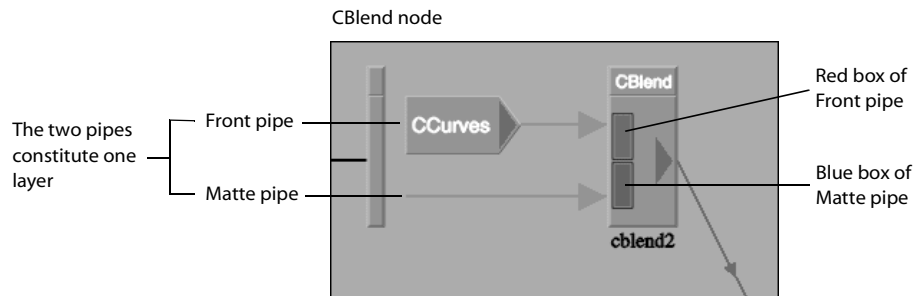
- Use the Front pipe of a layer to adjust the colour of a clip.
- Use the Matte pipe of a layer to isolate a region of the image to which the colour adjustments will be applied. With no matte, the colour correction is applied to the entire image.
- Blend the front and back images for each layer.
- Create layers of colour corrections for your front clip.

How the CBlend Node Works

The Front pipe of the CBlend node is designed to result in a colour image. This is represented visually by a red box at the right of the Front pipe. You can place any node on the Front pipe that results in a colour image. These include the Colour Curves, Colour Correct, RGB Blur, Degrain, and Regrain nodes.

The Matte pipe is designed to result in a matte; this is represented by a blue box at the right of the pipe. You can add any node that results in a matte to the Matte pipe, including the 3D Keyer, 2D Histogram, GMask, Edge and Degrain nodes.

NOTE: The Degrain node results in a colour image; however, it is often useful to place it on the Matte pipe before the 3D Keyer node (which uses a colour image for its input). For this purpose, it can be placed on the Matte pipe.



Components of the composite — The two pipes together, along with a back component, constitute one layer. For the first (bottom) layer, the CBlend node uses the source input image for the back component. For example, in the CBlend node on the Front branch of the default pipeline, the Front clip is used for the back component. To learn how subsequent layers are composited, see “Using Several Layers in a Blend Node” on page 696.

MBlend Node

Use the MBlend node to set up the matte operations you want to perform on a clip.

Use the Front pipe of a layer to define the matte, and use the Matte pipe to isolate an area where you want the matte to be applied. For example, you could use the Matte pipe to isolate an area of the key-in clip and apply a different matte to that area.

NOTE: When using the Matte pipe for this purpose, you would create a second layer and place the overall matte in the bottom layer and the second, localized matte in the upper layer. This is because layers are processed from the bottom up. See “Using Several Layers in a Blend Node” on page 696 for more information.

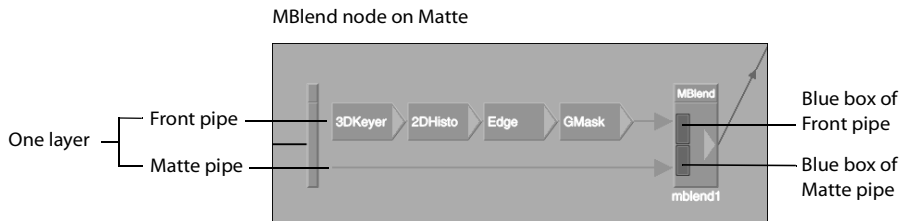
When there are no nodes on the Matte pipe, the matte is applied to the entire image.

As with the CBlend node, you can blend the front and back images for each layer and create several layers of mattes.

How the MBlend Node Works

The Front and Matte pipes of the MBlend node are both designed to result in a matte. This is represented visually by a blue box at the right of the Front and Matte pipes. You can place any node on the Front and Matte pipes that results in a matte. These include the 3D Keyer, 2D Histogram, GMask, Edge and Degrain nodes.

NOTE: The Degrain node results in a colour image; however, it is often useful to place it on the Matte pipe before the 3D Keyer node (which uses a colour image for its input). For this reason, it can be placed on the Matte pipe.



Components of the composite — The two pipes together, along with a back component, constitute one layer. For the first (bottom) layer, the MBlend node uses a pure black image for the back component. To learn how subsequent layers are composited, see “Using Several Layers in a Blend Node” on page 696.

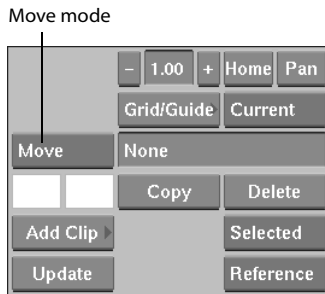
Adding a Matte to a Blend Node

Add a matte to the Matte pipe of the CBlend and MBlend nodes to limit the area on the image where the colour correction or matte will be applied.

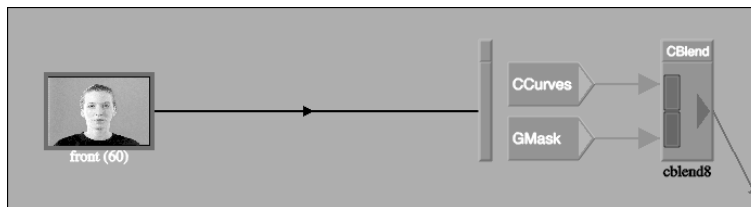
You can use any matte operation in the Matte pipe—add any combination of nodes that you would want to use to isolate an area of the front image. For example, you could add a garbage mask node to the Matte pipe of the CBlend node.

To add a matte to a blend node:

1. Select Move mode, and select the node in the bar.



2. Drag the node to the Matte pipe. Release the cursor.
The node is added to the Matte pipe.



3. Use the node menu to create the matte.
4. Add other nodes you want to use to refine the matte.

Blending the Front and Back Images of a Layer

There are two different methods you can use to blend the front and back images of a layer: Basic Blend and Curves Blend.

Basic Blend is appropriate for layers that do not have a matte component. It allows you to add the front and back images, and adjust the colour and opacity of the result image. In effect, since you are using the entire front image, the back image does not influence the result.

NOTE: Basic Blend is only available for MBlend nodes. It would not be useful to blend colour images in a composite in this way.

Curves Blend is suitable for layers that have a matte component. It allows you to use front and back matte curves to adjust the luminance of the front and back mattes, and also perform logical operations on the front and back images.

The two methods are mutually exclusive—you can set the blending in both the Basic Blend and Curves Blend menus, but it is the menu that is displayed that takes effect.

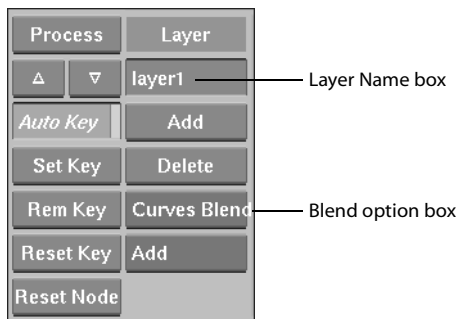
Basic Blend

Use Basic Blend to adjust the opacity and colour of an MBlend layer that has no matte component. It has the general effect of adding a second defined layer to the first one. You can control the opacity of the second layer.

NOTE: You can also use Basic Blend when you have defined a matte in the Matte pipe. In this case the blending is applied to the defined matte area only, rather than the entire front image.

To use Basic Blend:

1. Select the MBlend layer by clicking one of the blue boxes to the right of the layer.
The MBlend menu appears and the name of the layer you selected appears in the Layer Name box.



By default, if the layer does not have a matte component (that is, there are no nodes in the Matte pipe), the Basic Blend option is selected in the Blend option box, and the Basic Blend controls are displayed at the right of the menu.

NOTE: If Curves Blend was previously selected, then it is displayed. To display Basic Blend, select it from the Blend option box.

2. Display the Current Result view. This displays the result for the entire blend node.
3. Set the colour of the layer in the Colour field. A value of 100 sets the layer as white, a value of 0 sets it as black, and values in between give it a shade of grey.

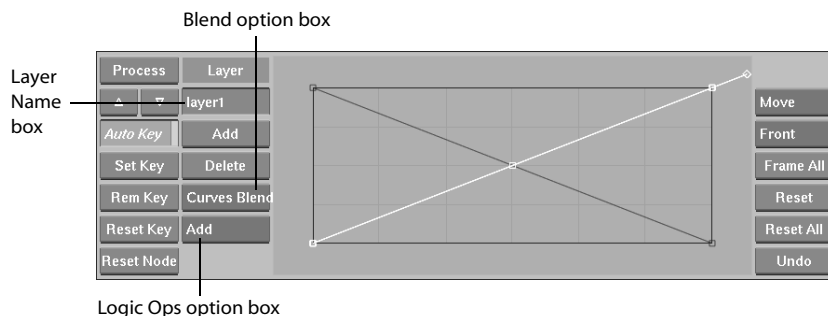
4. Set the opacity of the layer in the Opacity field. A value of 100 makes the layer completely opaque, and a value of 0 makes it completely transparent.

Curves Blend

Use Curves Blend to adjust the blending of the front and back images of a CBlend or MBlend node layer that has a matte component.

To use Curves Blend:

1. Select the CBlend or MBlend layer by clicking the red or blue box to the right of the layer. The CBlend or MBlend menu appears and the name of the layer you selected appears in the Layer Name box.



For MBlend nodes, if the layer has a matte component (that is, there are nodes in the Matte pipe), the Curves Blend option is selected in the Blend option box and the Matte Curves graph is displayed at the right of the menu.

NOTE: If Basic Blend was previously selected, then it is displayed. To display Curves Blend, select it from the Blend option box.

2. Display the Current Result view.
This displays the result for the entire blend node.
3. If you wish to perform a logical operation on the front and back images, select one from the Logic Ops option box.
4. Adjust the front and back matte curves as needed. For complete instructions, see “Adjust the Front and Matte Luminance Curves” on page 686.

Using Several Layers in a Blend Node

You can use up to four layers in blend nodes to create the composite. Using layers, you can avoid doing multiple passes on the same composite.

Layers are processed upwards from the bottom layer. In other words, the uppermost layer will appear on top of the next layer, and so on.

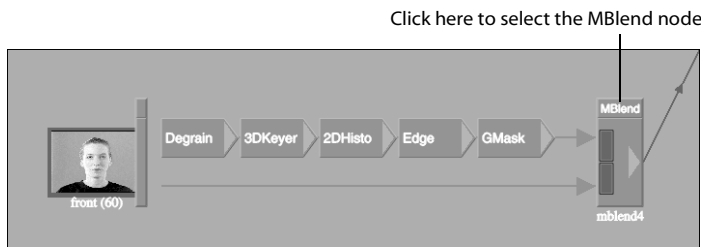
The back component of the lowest layer is a pure black image (for MBlend nodes) and the node input image (for CBlend nodes). For additional layers, however, the back component is different. On both CBlend and MBlend nodes, the result image of the lowest layer is fed into the next highest layer to serve as its back component. Likewise, the result image of this layer is fed into the next highest layer where it serves as that layer's back component, and so on.

NOTE: The Current Layer proxy does not display the current layer using the result of the previous layer, as described above. Instead, it displays the layer as if the back clip were a black image (for MBlend nodes) or the front image (for CBlend nodes).

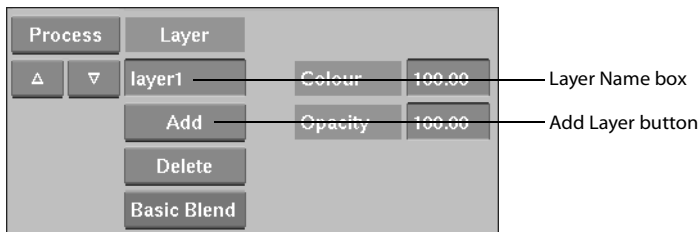
You can change the order of the layers using the Priority Editor.

To create a new layer:

1. Select the blend node by clicking anywhere in the rectangular box at its right end.

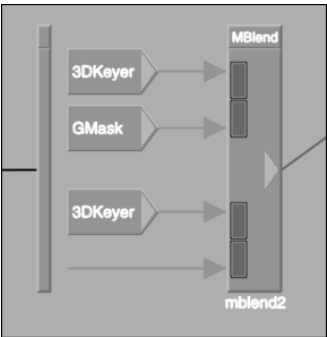


The node is highlighted and the blend node menu appears.



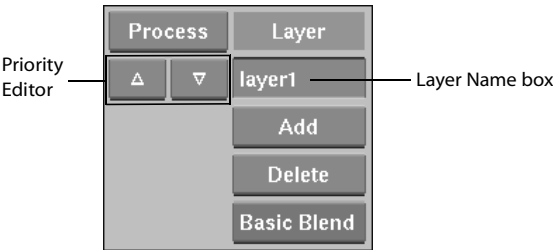
2. Click the Add button to create the new layer.
A new layer is added to the blend node, above the previous layer. The default name for the new layer is layer2.
3. Add nodes to the layer as needed, as described in “Adding Nodes to the Pipeline” on page 664.

A simple, two-layer MBlend node is shown as follows.



To change the order of the layers:

- 1. Select a layer by clicking either the red or blue box to the right of the pipes.
The selected layer's name appears in the Layer Name box.



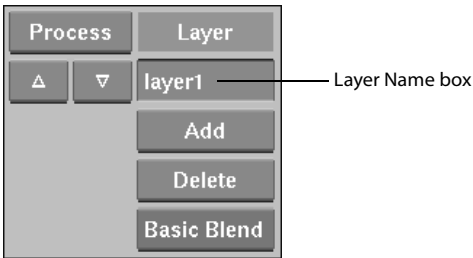
- 2. Click the ▲ button to move the selected layer up a layer, and the ▼ button to move it down a layer.

Renaming Layers

If you wish, rename layers to identify them more easily.

To rename a layer:

- 1. Select the layer by clicking on the red or blue box to the right of the layer.
- 2. Click on the Layer Name box in the blend node menu.



The on-screen keyboard appears.

3. Press **ESC** to remove the previous name and type a new name.
4. Press **ENTER** to save the name.

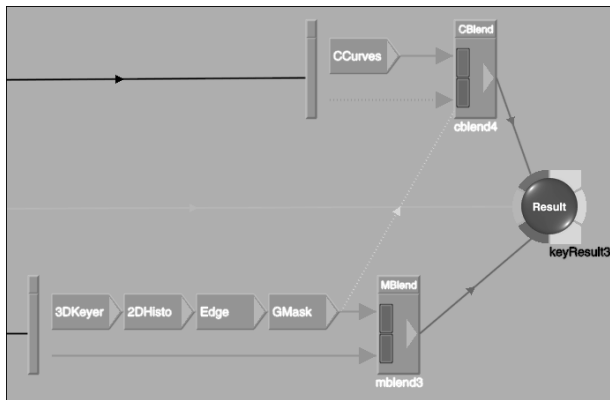
Bypassing Pipes

On blend nodes, you have the option of inputting a source image from another node on the pipeline, effectively bypassing the pipes. For example, suppose you want to use the key-in clip matte in a colour correction in the Matte pipe of the CBlend node. Instead of copying all the nodes in the Front pipe of the MBlend node, you could just add a branch from the end of the MBlend Front pipe to the blue box on the CBlend node, as shown below.

When attaching an alternate input source to a Matte pipe, you should use a matte image. If you use a colour image, it will be converted to greyscale.

For instructions on drawing the branch, see “Adding Nodes to the Pipeline” on page 664.

NOTE: To do this, you cannot have any nodes on the pipe receiving an alternate input source.



Animating the Shape Channel

The shape channel is a special type of channel that is unique to the Modular Keyer. Most channels can be edited in both the X and Y axes. However, some channels, such as an ellipsoid in the Modular Keyer, are composed of multiple parameters. For the shape channel, the Y value in the Channel Editor does not represent a single value; and each consecutive keyframe is assigned a sequential Y value. The first keyframe has a Y value of 1, the second keyframe has a Y value of 2, and so on, allowing you to adjust the curve.

Use the AutoKey, SetKey, RemKey, and Reset Key buttons to set and remove keyframes in the animation sequence.

For more information on animation and setting and removing keyframes, see Chapter 9, “Animation.”

Nodes Placement Table

Most nodes can be placed just about anywhere on the pipeline. There are a few exceptions when it comes to the pipes of blend nodes. The function of a particular node is not possible on some pipes because of the way the pipes operate. For more information, see “Blend Nodes” on page 690.

The following table lists the locations in the processing pipeline where you can place the various nodes of the Modular Keyer:

Node	CBlend Front Pipe	CBlend Matte Pipe	MBlend Front Pipe	MBlend Matte Pipe	Other Pipeline Parts
2D Histogram		3	3	3	3
3D Keyer		3	3	3	3
CBlend					3
Colour Correct	3				3
Colour Curves	3				3
Colour Warper	3	3	3	3	3
Degrain	3	3	3	3	3
Difference Matte					3
Edge		3	3	3	3
GMask		3	3	3	3
Logic Op					3
Matte Curves					3
Monochrome	3	3	3	3	3
LUT Editor	3	3	3	3	3
MBlend					3
Negative	3	3	3	3	3
Regrain	3				3
Result					
RGB Blur	3				3

Hot Keys

NOTE: Press and hold the hot key as you carry out the action, unless indicated otherwise.

Press:	To:
~	Toggle between the image window and schematic view.
M	Switch to Move mode.
P	Switch to Parent mode.
D	Switch to Delete mode.
SPACEBAR	Pan the image window or schematic view.
CTRL+ALT and click on node	Detach a node from the CBlend or MBlend pipes.
U	Update (process) selected node(s). Use when Auto Update (in Setup menu) is disabled. See "Updating Nodes" on page 668.
E	Edit the current node (for example, the Gmask or Colour Corrector nodes).
ESC	Return to the Modular Keyer from the Gmask and Colour Correct modules, keeping your settings.
= and click on node	Set a context point on a node (see "Setting a Context Point" on page 672).
F1	View the CurFront clip in the image window.
F2	View the CurBack clip in the image window.
F3	View the CurMatte clip in the image window.
F4	View the CurResult clip in the image window.
1	View the Context 1 clip in the image window.
2	View the Context 2 clip in the image window.
4	View the Result clip in the image window.
5	View the Action result in the image window (use when you have accessed the Modular Keyer from Action).
CTRL+SHIFT and drag in window	Draw a crop box in the image window. Note: The Crop button in the Setup menu must be enabled.

Use the 3D Keyer to pull a key from a key-in clip or to isolate a region of the front clip for colour correction. An RGB viewer displays the colours of your image in 3D RGB colour space to help you visualize and isolate specific colours.

Summary

In this chapter, you learn about:

- “Basic Keying Technique” on page 702
- “More Keying Techniques” on page 735
- “Mastering the RGB Viewer” on page 717
- “Setting 3D Keyer Preferences” on page 738
- “Animating Key Elements” on page 733
- “Hot Keys” on page 738

About the 3D Keyer

In the 3D Keyer you create a key using a key-in clip, which is typically a blue or green screen. You begin by defining a range of colours to be keyed out in the key-in clip, as well as the areas (such as the edges of the key and transparencies) you want to be semi-transparent. Areas that are keyed out or softened are represented visually in the RGB viewer. You can use the RGB viewer to analyze the colour areas in a clip, then apply other procedures to refine the key.

A matte is created according to the defined ranges, in which the keyed-out (transparent) areas are black, the semi-transparent areas are grey, and the remaining areas are white. You can then use the matte to composite a front and back clip. The front clip is usually the same as the key-in clip. The matte defines which areas of the front clip are used.

The 3D Keyer can be used for many purposes other than pulling a key from the key-in clip. For example, you can use it in the Matte pipe of the CBlend node to isolate a region of the front clip to which you want to apply a particular colour correction.

NOTE: It is recommended that you be familiar with the use and terminology associated with the Modular Keyer before you use the 3D Keyer. For more information, refer to Chapter 34, “The Modular Keyer.”

Basic Keying Technique

This section provides the recommended minimum steps for creating a precise key with the 3D Keyer. It also introduces the tools you can use in the 3D Keyer. Additional reference information on the available tools is given in the ensuing sections. To learn other techniques for perfecting your key, see “More Keying Techniques” on page 735 and the Modular Keyer exercises in the *flame* Tutorial.

The 3D Keyer node can be placed anywhere in the Modular Keyer processing pipeline except the Front pipe of the CBlend node. This is because the purpose of the CBlend Front pipe is to colour correct the Front clip, and the result of the 3D Keyer node is a matte.

Overview

The first step in pulling a key with the 3D Keyer is to set a range of colours to be keyed out in the key-in clip. This is called the tolerance range. All pixels in the key-in clip within the tolerance range are black in the matte. You set the initial tolerance range by selecting, or sampling, a range of colours in the key-in clip. The tolerance range must not be too large or the edge of the composite will be too hard and the subject in the front clip will appear to be pasted into the back clip.

The next step is to set a range of colours in the key-in clip to be partially transparent, to soften the transition between the front and back clips in the composite. This is called the softness range. You set the initial softness range by selecting a range of colours in the key-in clip. The pixels in the key-in clip within the softness range are grey in the matte. The key should have the greatest possible softness value.

Once the softness is set, you remove any graininess from the softened areas. Graininess, or noise, can occur at the edges of the key, or in the semi-transparent areas of transparencies such as water or glass. To remove the graininess, you analyze a softened area containing graininess, then scale the softness to smoothen the area.

Next, check to see if there are any grey areas within the key resulting from the softness setting. If so, there are several techniques to remove the grey areas:

- Performing negative sampling on the areas
- Using patches
- Using garbage masks
- Manipulating the softness element in the RGB viewer
- Using the Softness fields

At this point, you may want to adjust the luminance of the key using the 2D histogram. If needed, use the Edge tool to adjust the edges of the matte. For example, add a softening (blur) filter to the edges.

Next, remove any remaining colour spill, and if needed, colour correct the front clip. Lastly, you can adjust the luminance of the front and back mattes using the Matte Curves menu.

Summary of steps for pulling a key:

Step:	Refer to:
1. Select your clips and open the Modular Keyer.	"Accessing the 3D Keyer" on page 703.
2. Open the 3D Keyer.	"Accessing the 3D Keyer" on page 703.
3. Sample for tolerance.	"Sampling Tolerance" on page 706.
4. Sample for softness.	"Sampling Softness" on page 709.
5. Minimize the noise in softened areas.	"Removing Noise from Softened Areas" on page 711.
6. Remove grey areas from within the key.	"Removing Grey Areas within the Key" on page 714 for using negative sampling and patches. Chapter 36, "Garbage Masks and the Tracer," for using garbage masks. "Techniques for Adjusting Softness" on page 735 for using the RGB viewer and Softness fields.
7. Adjust the luminance of the key.	"2D Histogram Node" on page 677.
8. Adjust the edges of the key.	"Edge Node" on page 681 of Chapter 34, "The Modular Keyer."
9. Remove colour spill.	"Colour Curves Node" on page 678.
10. Adjust front and back matte curves.	"Result Node" on page 686.
11. Process the clip.	"Processing Options" on page 689.

Accessing the 3D Keyer

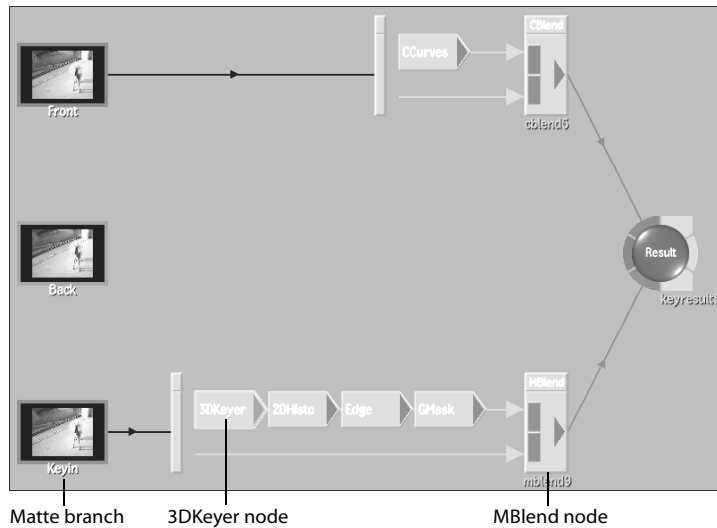
You access the 3D Keyer from the processing pipeline of the Modular Keyer.

NOTE: For information on accessing the Modular Keyer from Action, see "Using Colour Correction and the Keyer in Action" on page 915.

To access the 3D Keyer:

1. From the Effects menu, select the Modular Keyer.
2. Select a front clip, a back clip and a key-in clip.

The Modular Keyer schematic is displayed at the top of the screen.



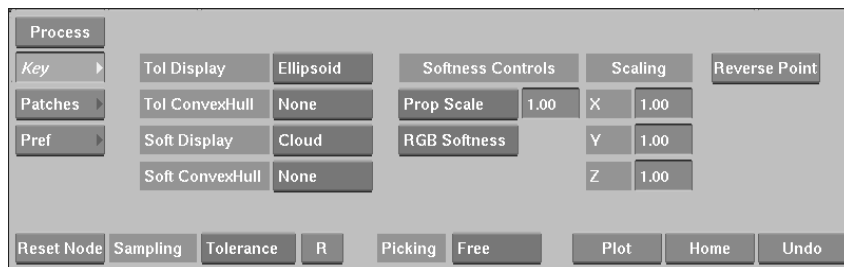
3. Select Move from the Edit Mode box (or press **M**).

Edit Mode box



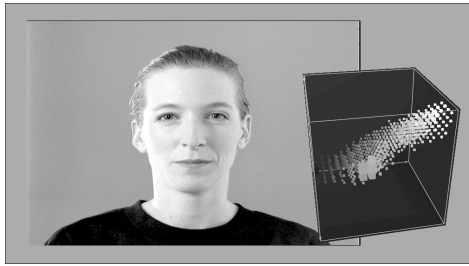
4. Click the 3DKeyer node.

The 3D Keyer menu appears at the bottom of the screen.



5. Click Pref to open the Preferences menu.
6. Click Reset Node and Confirm to reset any parameters set in the previous work session.
7. Press ~ to view the input image for the 3D Keyer. When using the default pipeline, this is the KeyIn clip.

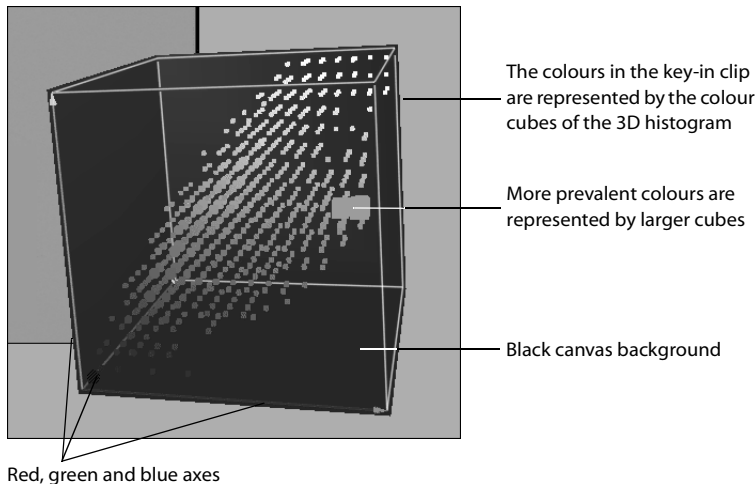
The image window showing the KeyIn clip and the RGB viewer appears:



Using the RGB Viewer

The RGB viewer is a 3D colour model for the key-in clip. Its X, Y, and Z axes represent the red, green, and blue components of the key-in clip's colour space. The cubes of colour show the distribution of all the colours of the key-in clip in RGB colour space. This 3D colour map is referred to as the 3D histogram.

RGB viewer



When you scroll through a clip, the 3D histogram cubes update to display the colour distribution of each frame.

Use the 3D histogram to analyze the colour mapping of the key-in clip and to locate the colours to be keyed out. When you sample for tolerance and softness, the sampled areas are represented as ellipsoids in the RGB viewer. You can manipulate sampled objects within the viewer to fine-tune the key. When you plot a colour in the image, its location is plotted in the RGB viewer.

You can turn the display of the histogram and the black canvas background on and off and set various options to control their appearance. Change the display any time as you create the key to help you best visualize the contents of your image and work with the RGB viewer. To learn more about controlling the display of the RGB viewer, see “Setting the 3D Histogram Display” on page 718.

Sampling Tolerance

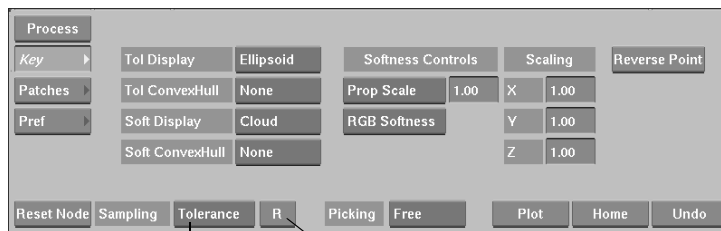
The first step in pulling the key is to define a range of colours in the key-in clip to be keyed out, called the tolerance range. To specify the tolerance range, sample the area that you want to key out in the image window. Take one or more samples until you are satisfied with the result. It is best to have a smaller tolerance, to avoid hard edges. You can always go back and add to the tolerance later on.

To sample for tolerance:

1. Click the Key button to display the Key menu.
2. Make sure you are at frame 1. If you move to a different frame while sampling, you will set a second tolerance keyframe (at the new frame), and animate the tolerance.

NOTE: If you want to avoid setting keyframes, disable the Auto Key button in the Preferences menu. For more on animating the key, see “Animating Key Elements” on page 733.

3. In the Sampling option box, set the sample type to Tolerance (or press **T**).



Sampling option box Sampling Reset button

4. Press **CTRL** and drag the cursor in the area to be keyed out.
A red rectangle appears. The pixel values within the rectangle are used to define the tolerance range. Notice that the rectangle has a maximum size. This ensures that the result is computed quickly.

NOTE: You can also take a sample by clicking on the image and dragging the colour picker that appears in the area to be keyed out. All pixel values that the colour picker samples are used to set the tolerance range.

5. Select CurResult from the Alt View option box (or press **2**) to see the matte.

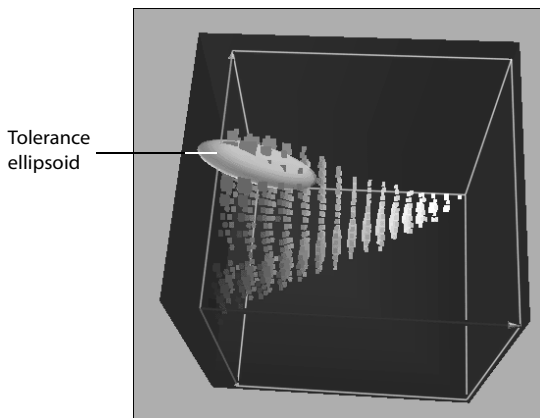
NOTE: For an explanation of the alternate views, see “Modular Keyer Menu” on page 659.

6. If needed, add to the tolerance range by sampling several more rectangles. The ellipsoid gets bigger as you add to the tolerance.

NOTE: If you are not satisfied with the result of the last sample, click Undo. The last sample you performed is cancelled. To reset the entire tolerance sample, click the R button. To remove the entire sample range at the current frame and start over with a new sample, press Ctrl-Alt and drag the cursor in the area to be keyed out.

Viewing the Tolerance Sample

The tolerance sample in the RGB viewer is represented as an ellipsoid by default. The ellipsoid shows you the location, in RGB colour space, of the range of colours included in the sample. All colours within the tolerance ellipsoid are black on the matte.



When the histogram is displayed, you can see the location of the sample in relation to the other colours in the image. When it is hidden, you can see the sample more clearly. As you are working, show or hide the histogram according to your needs.

1. To display or hide the histogram, press **H**. For more information on setting the histogram display options, see “Setting the 3D Histogram Display” on page 718.
2. To view the sample more closely, rotate the RGB viewer by pressing **CTRL** and dragging it. To zoom in on the RGB viewer, press **SHIFT** and drag inside it from left to right. To move the RGB viewer, press **ALT** and drag it.

Viewing the Convex Hull

The tolerance ellipsoid surrounds the exact sampled colours, which are connected in RGB space to form a 3D convex hull. You can see the convex hull within the ellipsoid by changing the tolerance display.

To view the convex hull:

1. Press **H** to remove the histogram display.
2. In the Tol Display option box, set the tolerance display to Cloud, which is semi-transparent.

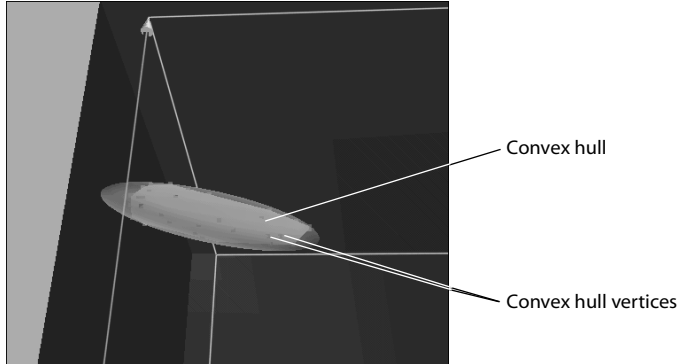
NOTE: To learn more about controlling the display of the tolerance sample, see “Changing the Display of Key Elements” on page 719.

3. In the Soft Display option box, set the softness display to None.

NOTE: When you set the tolerance range, the 3D Keyer automatically creates a minimum softness range that is equivalent to the tolerance range. This creates a softness ellipsoid of equal size to the tolerance ellipsoid in the RGB viewer. Sometimes the softness ellipsoid is visible at this point, depending on the way the ellipsoids are displayed.

4. Set the Tol ConvexHull display to Vert&Surf.

You can now see the convex hull, with vertices indicating the exact sampled colour areas:



The convex hull contains the exact colour values of the tolerance sample, while the ellipsoid represents the actual area in which corresponding pixels will be black in the matte. The 3D Keyer uses the ellipsoid rather than the convex hull to determine the keyed out area because it is much faster to calculate a mathematical shape such as an ellipsoid from frame to frame compared with an irregular shape.

You can change the shape of the convex hull directly in the RGB viewer to adjust the tolerance. The ellipsoid re-orientes itself according to the new shape. This is described in “Reshaping the Convex Hull” on page 731.

5. For the next step, return the softness display to Cloud.

Sampling Softness

After setting the tolerance range, you set a range of colours to become partially transparent in the key-in clip—the softness range. To specify the softness range, sample the areas to be softened in the image window.

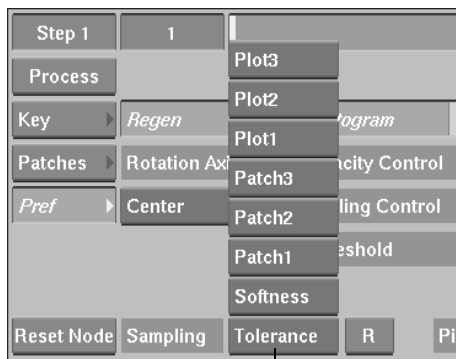
As shown in the following steps, the softness range can be set using the RGB or YUV colour models. RGB Softness is the default setting.

To sample for softness:

1. Make sure you are at frame 1. If you move to a different frame while sampling, you will set a second softness keyframe (at the new frame), and animate the softness.

NOTE: If you want to avoid setting keyframes, disable the Auto Key button in the Preferences menu. For more on animating the key, see “Animating Key Elements” on page 733.

2. In the Sampling option box, set the sample type to Softness (or press **S**).



Sampling option box

3. Select CurInput from the Alt View option box to view the input image for the 3D Keyer (or press **1**).
4. Zoom in to see the image more closely.
5. Identify an area where softness is needed. For example, if the clip consists of a talent in front of a blue or green screen, softness is needed at the edge of the talent to soften the transition between the front and back clips. If the clip contains a transparency, softness is needed in the semi-transparent area.

At this point you might want to switch to CurResult view, to watch the softness being applied to the matte as you are sampling.

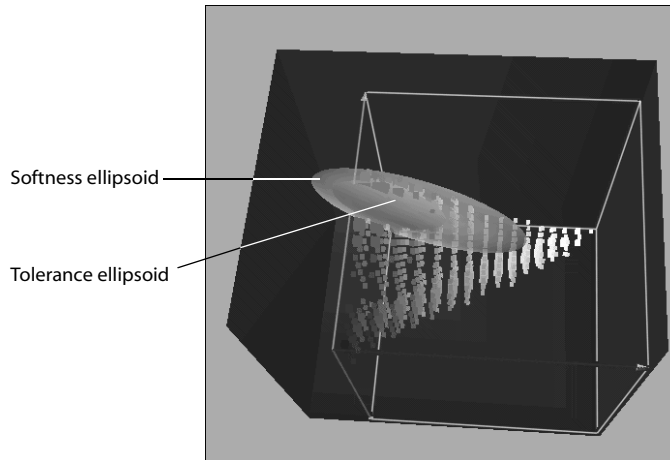
HINT: You can use any view as you sample the image, including CurResult view, which shows the final result of the processing pipeline. Use the **1** and **2** keys to easily switch between Current Input and Current Result as you sample.

6. Click in the area you identified.

The cursor changes to a colour picker.

7. Drag the colour picker in the area that needs softening. Avoid sampling any area that should not be softened, such as the key itself. (There is no need to avoid areas that have been keyed out). For transparencies, drag the colour picker in the area that you want to be semi-transparent.

A softness ellipsoid appears in the RGB viewer. Notice the tolerance ellipsoid within the softness ellipsoid.

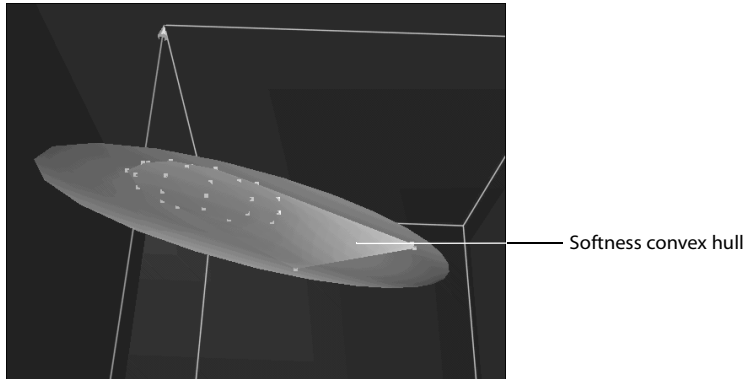


All colours within the softness ellipsoid (except those inside the tolerance ellipsoid) will be grey on the matte. Within the softness ellipsoid, the closer a point is to the tolerance ellipsoid, the darker the grey will be. For example, the mid-point between the edge of the tolerance ellipsoid and the edge of the softness ellipsoid is always the same mid-grey.

Notice how the softness ellipsoid increases in size as you sample the edges.

NOTE: The softness is rendered interactively on the matte – you see the results immediately. For information on rendering the softness only after releasing the cursor, see “Regen” on page 738.

8. Like the tolerance ellipsoid, the softness ellipsoid has a convex hull. To see it clearly, set the Tol Display and Tol ConvexHull display to None, and set the Soft ConvexHull display to Vert&Surf.



9. If needed, take another sample elsewhere on the edges or transparency. Each time you sample, the pixels touched by the colour picker contribute to the previously calculated softness range.

NOTE: If you are not satisfied with the result of a sample, click Undo. To reset the entire softness sample, click the R button. To remove the entire sample range and start over with a new sample, press Ctrl-Alt and drag the cursor in the area to be softened.

10. View the matte in the CurResult view.

The edges are softened but may be grainy.

Removing Noise from Softened Areas

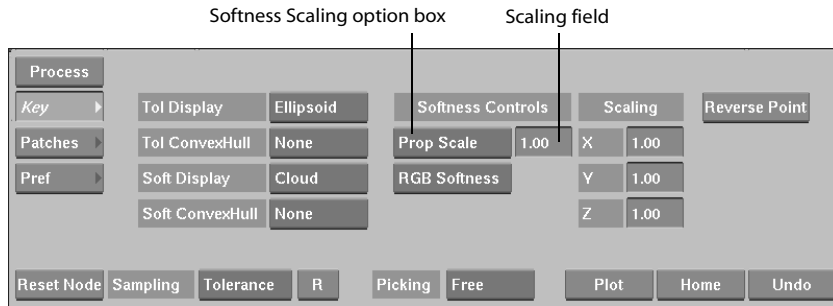
Once you sample for softness, use the Minimize Noise option to remove grain, or noise, from the softened areas. First, sample an area containing noise. The 3D Keyer analyzes the sampled area. Using Minimize Noise, you can then scale the softness in such a way as to minimize the noise in the softened areas.

Minimize Noise is especially useful for semi-transparent areas and edges requiring a lot of softness, such as smoke, reflection, and shadows.

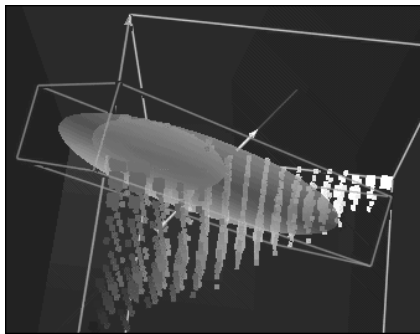
HINT: If the entire clip is grainy, or if you are not satisfied with the results after scaling the softness, consider using another approach. For example, degrain the clip with Degrain before keying with the 3D Keyer. You can add a Degrain node to the beginning of the Matte branch anytime. Additionally, use Minimize Noise on a degrained clip if needed.

To remove noise:

1. Sample for softness (see “Sampling Softness” on page 709).
2. Zoom in and choose the area you want to analyze. Look for graininess in the softened areas. For transparencies, look for graininess in the semi-transparent area.
3. Press **N** (Noise) and drag a rectangle in the selected area.
The 3D Keyer analyzes the pixels within the rectangle.
4. Select Minimize Noise from the Softness Scaling option box.



An arrow appears in the softness ellipsoid, showing the direction in which the softness needs to be increased to reduce graininess in the sampled area.



5. Increase the softness by dragging the cursor over the Scaling field to the right. The softness is increased based on the results of the analysis from the grainy region. Observe that the softness ellipsoid is scaled in the direction of the arrow.

NOTE: Each time you change the Scaling value, it is returned to 1.00.

6. If some edges or areas are still not softened, analyze again in that area and repeat the procedure.
7. Return the Softness Scaling to Prop Scale (proportional scaling).

NOTE: When Minimize Noise scaling is selected, it controls the arrow in the RGB viewer. To manually control the arrow, you must return to proportional scaling.

Alternatively, you can use **ALT+N** to scale softness based on the Minimize Noise analysis. This hot key is the equivalent of selecting Minimize Noise in the Softness Scaling option box, then scaling the softness using the Scaling field.

To scale softness using ALT+N:

1. Perform a noise analysis.

NOTE: **ALT+N** only works if an analysis for the noise has been done.

2. Press **ALT+N** and click and hold the cursor anywhere in the image window. Drag the cursor to the left to decrease the noise or to the right to increase it.

The softness is scaled according to the noise analysis. Notice that Minimize Noise appears in the Softness Scaling option box as you use the hot key, and when you release the cursor, the Softness Scaling option returns to Prop Scale.

HINT: For transparencies, use **V** as described above to remove unwanted grey areas within the matte, then use **ALT+N** to reduce noise in areas you chose to soften (the transparency). Alternately perform the two procedures until you achieve the best result.

Using YUV Softness

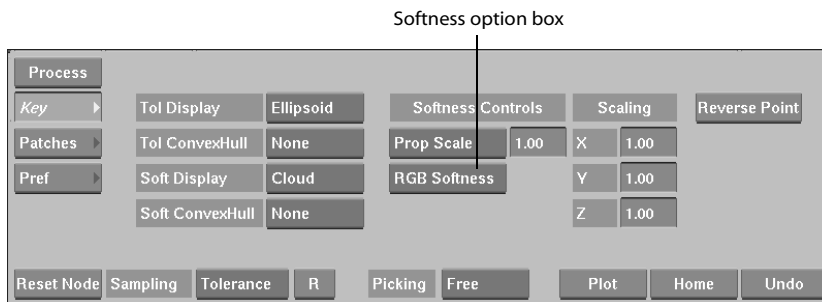
If you are not satisfied with the softness result—for example, if graininess still remains in the softened areas, try using YUV Softness. This option calculates the softness according to the YUV colour space.

YUV Softness often creates softened areas that are smoother, and typically it is useful for shadows and transparencies. However, since it removes more noise, resulting softness range is less precise.

NOTE: You can switch between RGB and YUV Softness and retain their settings.

To use YUV Softness:

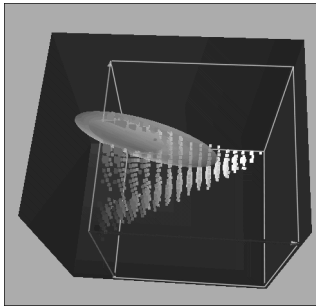
1. Select YUV Softness from the Softness option box.



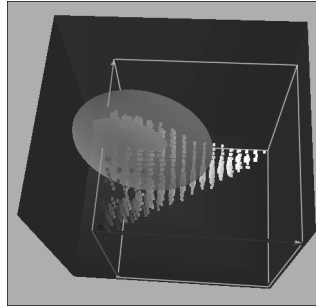
2. Sample for softness as described in “Sampling Softness” on page 709.
3. Minimize the noise as described in “Removing Noise from Softened Areas” on page 711.

Compare YUV with RGB Softness

Return to RGB Softness. The previous RGB Softness range is still in effect. Compare the results with the YUV Softness. Generally, the softness ellipsoid is larger with YUV Softness, as shown in the following example.



With RGB Softness, the softness ellipsoid is bound by the convex hull



With YUV Softness, the softness ellipsoid is scaled according to the Y component of YUV colour space.

If you are satisfied with the result, use YUV Softness. If not, return to RGB softness.

Removing Grey Areas within the Key

Increasing the softness may result in unwanted grey areas within the white part of the matte. There are many techniques you can use to remove these grey areas. In this section, two methods are shown—using Negative Sampling to refine the softness range and using the Patch tool to create “selective keys” within the white areas of the matte. To learn other ways to remove unwanted grey areas, see “Techniques for Adjusting Softness” on page 735.

Negative Sampling

This is probably the simplest method of removing greys from the matte. You sample in the unwanted grey area to subtract those colour values from the softness range.

Try this method first to see if it solves the problem. However, if the colour values within the foreground subject are too similar to those at the edges, you may remove too much of the softness.

To perform negative sampling:

1. Display the Current Result in the image window by pressing **2**. Identify the unwanted grey areas within the matte.
2. Select Sample Softness from the Sampling option box.
3. Press **ALT** and click on a grey area. Gently drag the cursor over the area.
If the negative sampling removed too much of the desired softness or brought back graininess, click the Undo button and try another method.



WARNING: There is only one level of Undo, so if you want to retain your original softness range, you must press Undo each time you perform a negative sampling.

Adding a Patch

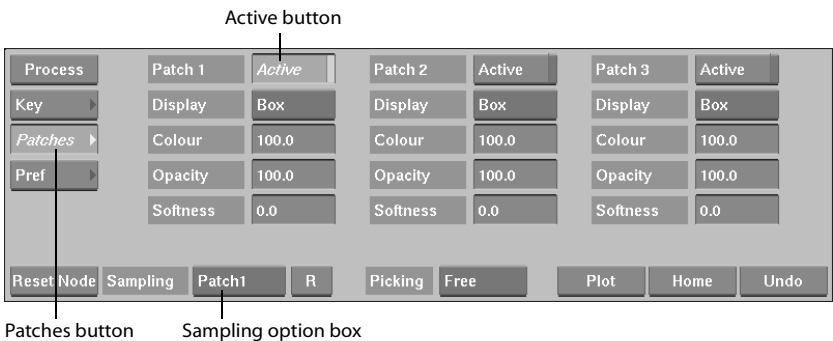
Patches are another type of sample you can take in the image window. Unlike tolerance and softness, you can specify the colour that a patch sample renders on the matte. Patch samples can be rendered as white, black, or any shade of grey on the matte. You can also set opacity of a patch.

One use for patches is to remove unwanted grey areas from the matte. Use a patch to isolate a range of colours from those that have been included in the softness range, then set the colour you want the pixels to have on the matte. Sampling a patch to remove grey areas within the key is similar to creating a garbage mask by keying instead of drawing.

You can use up to three different patches in a key.

To sample a patch:

1. Display the Current Result in the image window by pressing **2**. Identify the unwanted grey areas within the matte.
2. In the Sampling option box, set the sample type to Patch1 (or press **ALT+1**).
3. Click the Patches button to open the Patches menu.



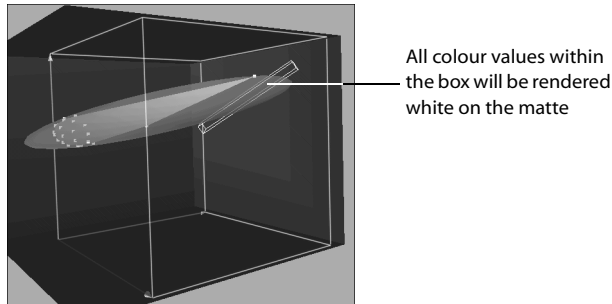
4. Enable Patch 1 by clicking its Active button.

5. Specify the colour in which you want to render the patch. In this example, make the patch white by setting the P1 (Patch 1) Colour to 100 (this is the default value). To render the sampled values as black, enter a value of 0. Use a value between 0 and 100 for any shade of grey.

6. Sample the unwanted grey area by dragging the cursor over it.

The grey areas of the matte corresponding to the colour values sampled are changed to white.

All the colour values sampled are included in the patch. Notice the patch in the RGB viewer, represented by a white wireframe box.



7. If necessary, sample again to add to the patch. Watch the edges to make sure they are not affected by the sample.

HINT: To remove colour values from the sample, press Alt as you sample. To undo a sample, click Undo.

8. If needed, soften the edges of the patch by dragging the cursor to the right or left over the P1 Softness button.

A negative softness value softens the edges of the patch inwards from the edge. A positive value softens the edges outwards beyond the edge, adding softened pixel values to the patch.

When the patch is displayed as a white wireframe box, you can see the softness, represented by a red wireframe box. For information on setting the patch display, see “Changing the Patches Display” on page 721.

9. If needed, adjust the opacity of the patch using the P1 Opacity field. A value of 100% renders the patch as fully opaque. A value of 0% renders the patch as fully transparent.

The RGB viewer now contains three key elements:

- The tolerance sample (as an ellipsoid)
- The softness sample (as an ellipsoid)
- The patch (as a white wireframe box)

For instructions on completing the key using the basic keying technique, see the sections referred to in steps 7-11 of “Summary of steps for pulling a key:” on page 703.

Mastering the RGB Viewer

The RGB viewer gives you a full graphic representation of what you are doing in the key. In this section, learn how to:

- Move, rotate, and zoom in on the RGB viewer.
- Change the display of the 3D histogram and canvas.
- Change the appearance of elements in the RGB viewer.
- Select key elements in the RGB viewer.
- Plot pixels in the image and in the RGB viewer.
- Translate, rotate, and scale key elements in the RGB viewer.
- Reshape the convex hull by manipulating its vertices.

Moving, Rotating and Zooming in on the RGB Viewer

The RGB viewer represents the 3D colour space of a key-in clip. As you work on the precision of a key, you will want to change the position, orientation, and size of the RGB viewer. Use **ALT**, **SHIFT**, and **CTRL** to move, rotate, and zoom in on the RGB viewer:

To:	Do this:
Move the viewer	ALT+CLICK on the viewer and drag it.
Zoom in on the viewer	SHIFT+CLICK on the viewer and move the cursor to the left or right.
Rotate the viewer	CTRL+CLICK on the viewer and move the cursor in any direction. By default this rotates the viewer around its centre. Using Rotation Axis options in the Preferences menu, you can also rotate the viewer around a plotted point, or around the centre of the tolerance or softness ellipsoid. This makes it easier to zoom in on specific elements in the viewer.

Resetting the RGB Viewer

To reset the viewer's position, orientation, and size, click Home in the lower left corner of the Key, Patches, and Preferences menus.

NOTE: Other settings, such as resolution or canvas light, are not reset.

Setting the Display of the RGB Viewer

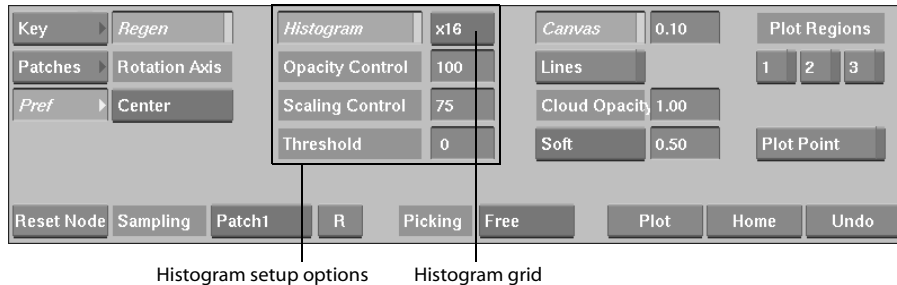
As you are working with sampled elements in the RGB viewer, set the display of the 3D histogram, canvas background, and border lines to suit the operation you are performing. For example, if you are modifying the softness range, you might want to hide the 3D histogram in order to see the softness ellipsoid more clearly.

Setting the 3D Histogram Display

The 3D histogram provides a visual representation of the colours in the key-in clip. It shows you how the colours in your clip are distributed in RGB colour space. The most prevalent colours are represented by larger cubes; colours that are less prevalent are represented by smaller cubes.

NOTE: The size of the cubes approximates the colour usage in the image, but is not intended as an exact representation. You can increase the resolution of the cubes using the histogram grid.

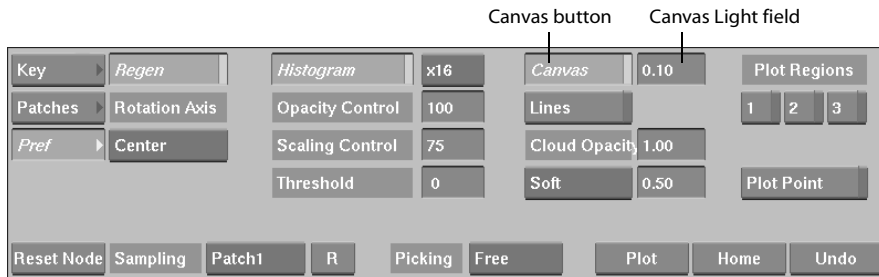
Change the display of the histogram using the options in the Preferences menu.



Use:	To:
Histogram	Show or hide the histogram. Alternatively, use the H hot key.
Histogram Grid	Select a resolution for the display of the cubes: <ul style="list-style-type: none"> • x16 (the default) • x8 to display fewer and larger cubes • x32 to display smaller cubes
Opacity Control	Adjust opacity of the cubes. As you adjust opacity, smaller cubes become transparent at a faster rate than larger ones. The default setting is 100%.
Scaling Control	Resize the cubes. Cubes representing less prevalent colours (the smaller cubes) are scaled down more than those representing more prevalent colours. The cube of the most prevalent colour remains the same size (as set in the Histogram Grid option box). By increasing the size difference between the colours of greater and lesser prevalence, you can see the main components of the colour space more clearly.
Threshold	Remove the display of colours that are less prevalent in the image. As you increase the threshold, less prevalent colours are increasingly removed from the histogram.

Setting the Canvas Display

The canvas is the black background of the RGB viewer. Control the display of the canvas using the options in the Preferences menu.



Use:

Canvas button

To:

Show or hide the canvas. Alternatively, use the C hot key.

Canvas Light field

Set the lighting in the RGB viewer. The light source emanates from behind the viewer.

Setting the Line Display

You can show or hide a white outline of the borders of the RGB viewer, as well as green wireframe boxes around selected ellipsoids. Show the boxes to quickly identify the selected element, or hide them to reduce “clutter” in the viewer.

- To show or hide lines in the RGB viewer, enable or disable the Lines button in the Preferences menu (or press L).

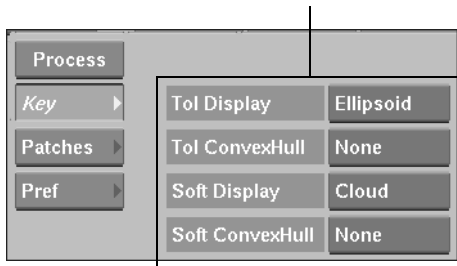
Changing the Display of Key Elements

You can change the display of the tolerance ellipsoid, softness ellipsoid, and patches as you are working on the key to suit the operation you are performing. For example, to adjust the tolerance range after setting the softness, you could set the softness display to wireframe, or remove its display altogether, for a better view of the tolerance. For more information on the tolerance and softness ellipsoids, see “Sampling Tolerance” on page 706 and “Sampling Softness” on page 709.

Setting the Tolerance and Softness Ellipsoids

As you sample the tolerance and softness in a key, corresponding key elements appear in the RGB viewer. Set the appearance of the tolerance and softness ellipsoids to make it easier to work with them while refining the precision of the key.

Use these fields in the Key menu to set the display of the tolerance and softness in the RGB viewer.



Use the Tol Display and Soft Display option boxes to set the appearance of the tolerance and softness ellipsoids.

Select:	To:
Ellipsoid	Display an opaque ellipsoid of the exact colours sampled.
Cloud	Display a semi-transparent grey ellipsoid. You can set the transparency of the ellipsoid using the Cloud Opacity field in the Preferences menu. See "Setting the Cloud Opacity" on page 721 for details.
WireFrame	Display the ellipsoid as a wireframe outline.
None	Remove the display of the ellipsoid. Note that when an ellipsoid is not displayed, it still has an effect on the image.

Use the Tol ConvexHull and Soft ConvexHull option boxes to set the appearance of the tolerance and softness convex hulls.

Select:	To:
Vertices	Display only the vertices of the convex hull.
Surface	Display only the surface of the convex hull.
Vert&Surf	Display both the surface and vertices of the convex hull.
None	Remove the display of the convex hull.

NOTE: Vertices do not appear on keyed elements when Tol Display or Soft Display is set to Ellipsoid (because an ellipsoid is opaque).

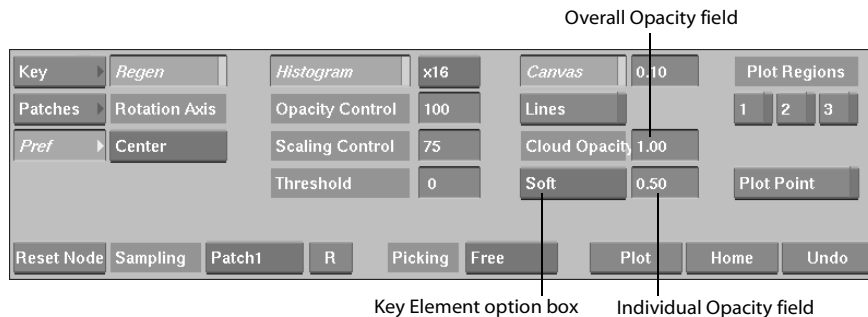
Changing the Patches Display

You set the display of each of the three patches individually. In the Patches menu, use the Display option boxes to set the display of patches 1, 2 and 3.

Select:	To:
Box	Display the patch as a wireframe box.
Surface	Display only the convex hull of the patch.
Box&Surf	Display both the wireframe box and convex hull of the patch.
Cloud	Display the patch as a semi-transparent grey box. You can set the transparency of the box using the Cloud Opacity field in the Preferences menu. See “Setting the Cloud Opacity” on page 721 for details.
None	Remove the display of the patch. Note that when a patch is not displayed, it still has an effect on the image.

Setting the Cloud Opacity

You can control the opacity of the Cloud setting of ellipsoids and patches as they appear in the RGB viewer using the Cloud Opacity fields in the Preferences menu.



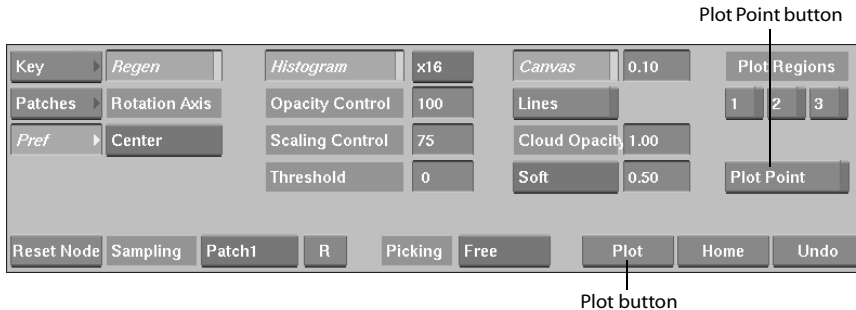
You can adjust the opacity for all key elements, or for individual key elements:

- To set the cloud opacity for all ellipses and patches, use the Overall Opacity field.
- To set the cloud opacity for an individual key element, select the key element from the Key Element option box, then set the opacity in the Individual Opacity field.

Displaying Plotted Points and Regions

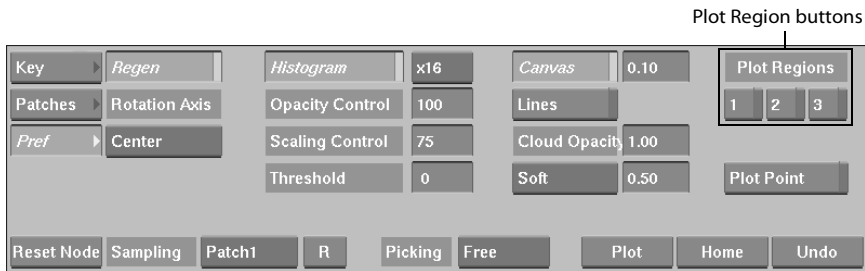
When you plot colour values and ranges of colour values in the key image, you can display or hide the plotted points and regions in the RGB viewer:

- To show or hide plotted points, enable or disable the Plot Point button in the Preferences menu (or press **B**).



NOTE: When you plot a colour value, the Plot Point button is automatically enabled to display the colour value in the viewer.

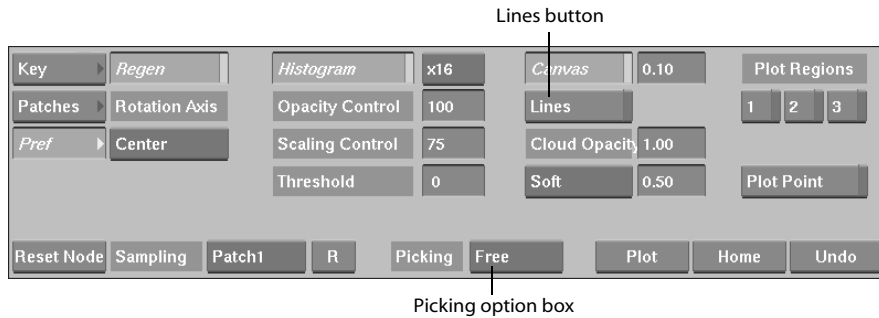
- To show or hide Plot Regions 1, 2, or 3, enable or disable the corresponding buttons in the Preferences menu.



Selecting Key Elements in the RGB Viewer

As you translate, scale, and rotate key elements in the RGB viewer, you must be able to select key elements individually. The 3D Keyer provides two methods for selecting key elements—clicking on them in the viewer or using the Picking option box. The Picking option box enables precise selection especially when tolerance, softness, and patch elements are overlapping each other in the RGB viewer.

The Preferences menu contains the Lines button, which allows you to display lines around key elements when they are selected within the viewer. This can make it easier to identify the selected key element.



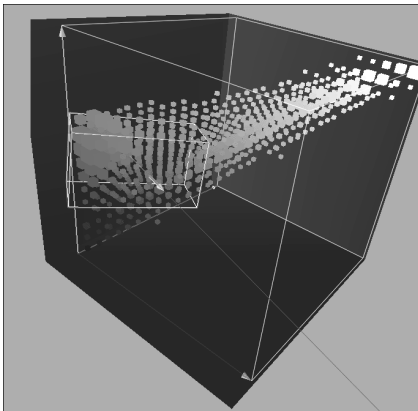
Clicking on Key Elements

When you click on key elements in the RGB viewer, a directional arrow appears through the selected element. The arrow points in a particular direction according to where you click on the element. For more information about the directional arrow, see “Controlling the Arrow” on page 725.

NOTE: The Picking option box must be set to Free in order to select an element by clicking on it. Also, the Softness Scaling must be set to Prop Scaling, not Minimize Noise, because Minimize Noise selects softness and controls the arrow direction.

To select key elements by clicking on them:

1. Verify that the Picking option box is set to Free and that the Softness Scaling is set to Prop Scaling.
2. Select a key element in the RGB viewer by clicking on it.



A directional arrow appears through the selected key element and, if the Lines button is enabled, a green wireframe box appears around the selected key element.

3. At this point, you are ready to proceed with modifying the selected key element (see “Translating, Scaling, and Rotating Key Elements” on page 729 and “Reshaping the Convex Hull” on page 731).

If you are unable to select the element, the element may be behind or inside another element. You can hide the element that is in the way by setting its display to None. See “Changing the Display of Key Elements” on page 719 for details. Alternatively, use the Picking option box to select the element, as described in the next procedure.

Using the Picking Option Box

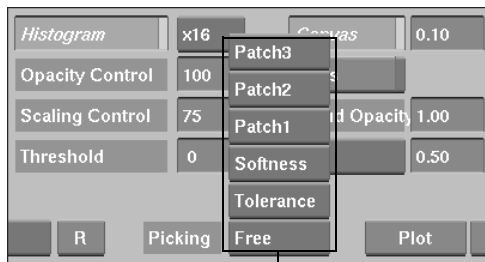
When an element is selected with the Picking option box, you cannot select other elements in the viewer by clicking on them. This is helpful in preventing other key elements from being selected by accident while you are working with a particular key element.

The Picking option box is also useful because it allows you to select a key element without changing its current arrow position. For example, if you accidentally click elsewhere in the viewer and lose the selection, and want to re-select the element without changing the arrow position, use the Picking option box.

NOTE: When you use the Picking option box to select a key element that has not been previously selected, the arrow is not displayed. You must actually click in the element to set the arrow direction.

To select key elements using the Picking option box:

1. Verify that the Softness Scaling is set to Prop Scaling, not Minimize Noise, which forces the softness to be selected.
2. Select the key element from the Picking option box.



Key elements to pick

If lines are displayed in the viewer, a selected ellipsoid has a green wireframe box around it and patch boxes change from white to green. If the element’s arrow direction has been previously set, the arrow is displayed.

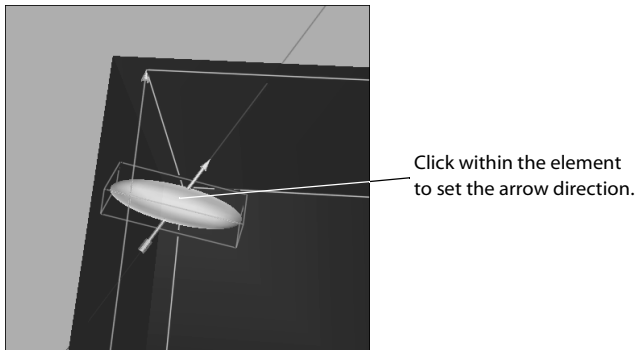
3. To display the element's arrow and set its direction, click in the element.

At this point, you are ready to proceed with adjusting the range of the selected key element. See “Translating, Scaling, and Rotating Key Elements” on page 729 and “Reshaping the Convex Hull” on page 731 for complete details.

Controlling the Arrow

Each of the key elements (tolerance, softness and patches) has a directional arrow that passes through its centre point. Use this arrow to indicate the direction in which you want to perform a particular action, such as scaling or translating the element.

- To display an element's arrow, select the element by clicking on it.
- To set the arrow direction manually, click inside the key element anywhere between its centre and the area towards which you want the arrow to point.



NOTE: You can only click on a part of the element's surface that is visible. If necessary, rotate the RGB viewer so that the correct area of the surface is visible.

The arrow changes direction to pass through the element's centre and the point where you clicked.

- To point the arrow towards a plotted point, simply select the element (either by clicking on it or using the Picking option box) before plotting. The arrow will automatically point towards the plotted point.
- To reverse the direction of the arrow, click the Reverse Point button in the Key menu.

NOTE: When Minimize Noise is enabled, this tool controls the arrow direction. See “Removing Noise from Softened Areas” on page 711 for details.

Plotting Colour Values

You can plot pixels in the key image and view the location of the pixel colour value in the RGB viewer. You can plot a single colour value or a range of colour values. These plotted areas do not affect the key at all—they simply identify colour values in the viewer.

It can be useful to plot a colour value to view its location in relation to key elements in the viewer. Additionally, you can employ a number of techniques to modify a key element in relation to a plotted colour value or range of values. For example, you could plot a colour value corresponding to an unwanted grey area within the foreground subject, then translate or scale the softness ellipsoid such that the colour value is outside the ellipsoid.

Plotting Single Colour Values

Plotting a pixel in the key image sets the exact location of the colour value in the RGB viewer. The colour value is identified by a small, white wireframe cube in the viewer. As an alternative, you can use the Plot tool to select a colour value directly in the viewer by clicking on a 3D histogram cube.

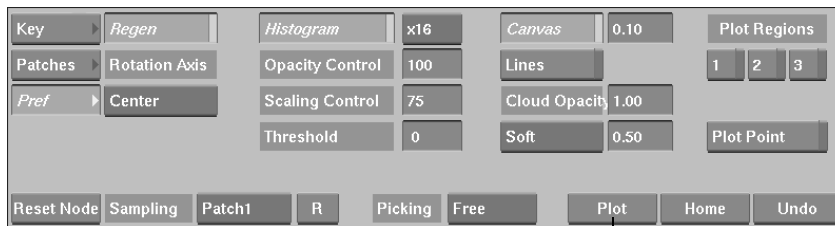
If a key element in the viewer is selected before you plot the colour value, the element's directional arrow moves to point towards that colour. Plotting pixels in this way allows you to quickly refine the matte by translating or scaling a key element with respect to the plotted colour.

To plot a pixel on the key image:

1. In the RGB viewer, select the key element that you want to modify in relation to a plotted colour value. For example, click on a patch element.

If you have problems selecting the element, refer to “Selecting Key Elements in the RGB Viewer” on page 722.

2. If needed, zoom in on the key image so that you can select a pixel more easily.
3. Click the Plot button (or press **O**).



Plot button

Plot mode is activated and the cursor changes to a colour picker.

4. Click on a pixel in the key image. You can also drag the cursor in the image, then release the cursor when the colour picker is over the desired pixel.

In the RGB viewer, the colour value of the selected pixel is plotted and the arrow of the selected key element moves to point towards the plotted colour value.

If needed, rotate the viewer by pressing Ctrl and dragging it to get a better view of the key element with respect to the plotted value.

You are now ready to proceed with modifying the selected key element. See “Translating, Scaling, and Rotating Key Elements” on page 729 and “Reshaping the Convex Hull” on page 731.

To select a colour value in the RGB viewer:

1. In the RGB viewer, select the key element that you want to modify in relation to a plotted colour value. For example, click on a patch key element.
If you have problems selecting the element, refer to “Selecting Key Elements in the RGB Viewer” on page 722.
2. Press **H** to display the 3D histogram.
3. If needed, zoom in on the viewer or rotate it so that you can select a histogram cube more easily.
4. Click the Plot button (or press **O**).



Plot button

Plot mode is activated and the cursor changes to a colour picker.

5. Select a cube in the histogram of the RGB viewer. If needed, you can drag the colour picker around the viewer and release it when the colour picker is pointing to the right colour value.
The selected cube is surrounded by a white wireframe box and the direction of the key element arrow moves toward the selected cube.

You are now ready to proceed with modifying the selected key element. See “Translating, Scaling, and Rotating Key Elements” on page 729 and “Reshaping the Convex Hull” on page 731.

Plotting a Range of Colour Values

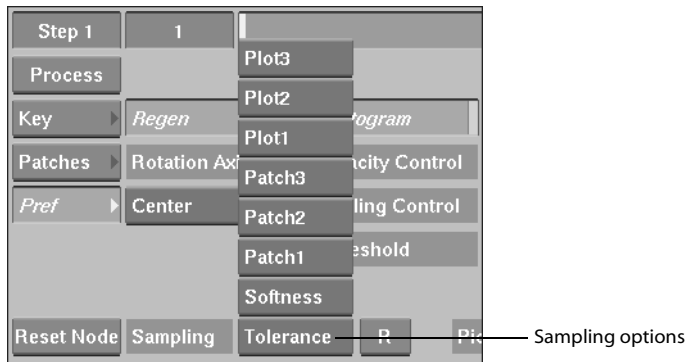
When you plot a range of colour values in the image, the resulting plotted region in the RGB viewer appears as a 3D convex hull. You can plot up to three different colour ranges in the image. The plotted regions do not in themselves affect the key image in any way.

A plotted region allows you to compare the intersection of a key element with a particular colour range in the image, providing you with yet another alternative for adding precision to your key.

For example, use this tool to plot a colour range that you do not want to be softened. Using the plotting region as the reference, you translate and scale the softness ellipsoid away from the plotted region to ensure the softness precision of the key. Plotted regions also help you to determine whether or not to use a garbage mask. If a color that you want to soften is the exact color that you want key out, then you should apply a garbage mask.

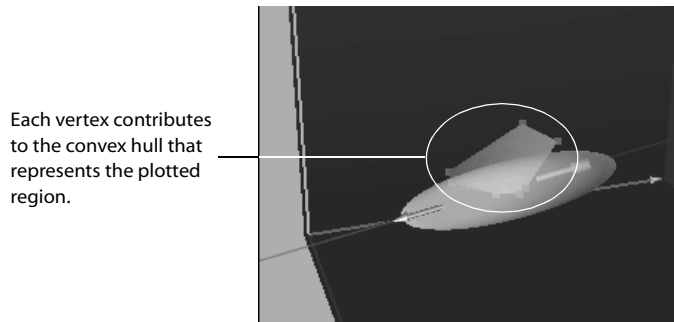
To plot a region:

1. In the Preferences menu, enable the Plot Regions 1 button. This enables the display of Plot region 1.
2. In the Sampling option box, select Plot1.



3. In the key image, sample one or more pixels using the same methods that you use for sampling tolerance or softness (simply drag in the image, or press **CTRL** or **CTRL+ALT**).

A 3D convex hull appears in the RGB viewer.



4. Continue sampling pixels in the key image until you are finished plotting the region.

Plotting Several Regions — You can plot a second and third region, and compare the positions of the convex hulls. To plot a second region, use the procedure described above, selecting Plot 2 from the Sampling box and enabling the Plot Regions 2 button.

For example, sample one plot region in an area of the matte containing unwanted grey areas, and another plot region where the grey area is intentional. You can then observe where the colours of the two regions are the same (where the plotted regions intersect), and adjust the softness accordingly. For instance, you could:

Do this:

Move the softness key element away from the plotted region intersection.

Use a patch.

Use a garbage mask.

To:

Remove the grey areas from the foreground subject.

Prevent the matte from becoming over softened or transparent.

Keep part of the key that has the exact color as the area being softened.

Translating, Scaling, and Rotating Key Elements

You can fine-tune the key by translating, scaling, or rotating the tolerance ellipsoid, softness ellipsoid, and patches in the RGB viewer. For example, scale the softness ellipsoid in a particular direction to increase the softness. You use the same methods for all three key element types.

To translate or scale an element in a particular direction, you use its arrow to specify the direction. When scaling an element along the axis of the arrow, you have the option of scaling it equally in both directions from its centre, or scaling it only in the forward direction of the arrow. You can also scale an element proportionally in all directions (X, Y and Z directions).

Before performing the following operations, set up the RGB viewer to best display the elements you are working with. Zoom in on the element, decide whether you want to show the histogram or not, and hide elements that are in the way. Refer to previous sections in this chapter, such as “Changing the Display of Key Elements” on page 719 for more information.

HINT: Display the 3D histogram as you make your modifications to see the areas of colour that you are working in. Try reducing the size and opacity of histogram cubes to see more clearly. See “Setting the 3D Histogram Display” on page 718 for more information.

To modify the tolerance or softness using these techniques, the ellipsoid must be displayed. The Tol Display or Soft Display must be set to Ellipsoid, WireFrame or Cloud. Likewise, when reshaping patches, the patch box must be displayed. The box is displayed when the Display is set to Cloud, Box, or Box&Surf.

See “More Keying Techniques” on page 735 to learn ways to apply these techniques in specific situations.

To translate a key element:

1. Select the element (see “Selecting Key Elements in the RGB Viewer” on page 722 if you have problems selecting the element).
2. Identify the direction in which you want to translate the element and set the arrow in this direction. You can do this manually (as explained in “Controlling the Arrow” on page 725) or by plotting a pixel on the image, or a histogram cube (see “Plotting Single Colour Values” on page 726). When you plot a pixel on the image or a histogram cube, the arrow of the selected key element automatically points towards the plotted pixel.
3. Press **6** and drag the cursor within the RGB viewer. Drag to the right to translate the element forward along the arrow axis, and to the left to translate it backward.

To rotate a key element:

1. Select the element (see “Selecting Key Elements in the RGB Viewer” on page 722).
2. Press **7** and drag the cursor within the key element.
The key element rotates in any direction around its centre point.

To scale a key element proportionally:

1. Select the element (see “Selecting Key Elements in the RGB Viewer” on page 722).
2. Press **8** on the keyboard and drag within the RGB viewer. Drag to the right to increase its size, and to the left to make it smaller.

NOTE: To scale the softness ellipsoid proportionally, you can also use the Prop Scale option in the Key menu.

To scale a key element from its centre:

1. Select the element (see “Selecting Key Elements in the RGB Viewer” on page 722).
2. Set the arrow in the desired direction. You can do this manually (as explained in “Controlling the Arrow” on page 725) or by plotting a point on either the image or a histogram cube (see “More Keying Techniques” on page 735). When you plot a point on the image or a histogram cube, the arrow of the selected key element automatically points towards the plotted pixel.
3. Press **9** and drag the cursor in the RGB viewer. Drag to the right to scale it larger along the axis of the arrow, and to the left to scale it smaller.

The element is scaled equally in the forward and backward directions of the arrow.

To scale a key element in one arrow direction:

1. Select the element (see “Selecting Key Elements in the RGB Viewer” on page 722).
2. Set the arrow in the desired direction. You can do this manually (as explained in “Controlling the Arrow” on page 725) or by plotting a point on either the image or a histogram cube (see “More Keying Techniques” on page 735). When you plot a point on the image or a histogram cube, the arrow of the selected key element automatically points towards the plotted pixel.
3. Press **0** and drag the cursor in the RGB viewer. Drag to the right to scale it larger in the forward direction of the arrow, and to the left to scale it smaller (towards the centre of the element).

The element is scaled in the forward direction of the arrow only.

Reshaping the Convex Hull

You can exclude or add specific ranges of colour to a sample by modifying the shape of the convex hull of the softness ellipsoid, tolerance ellipsoid, and patches. You do this by manipulating the vertices on the hull. The ellipsoid or patch box changes shape according to the new shape of the convex hull.

This can be a very accurate way of modifying the sampled ranges since you can see the area of RGB colour space in which you are working and because you can plot specific parts of the image and add them to the hull. Also, scaling the ellipsoid using one vertex of the convex hull provides a more precise (localized) result than scaling using the arrow alone (using **0**).

Use these methods to change the shape of the convex hull:

- Add vertices to it and move these vertices.
- Delete vertices from the convex hull.

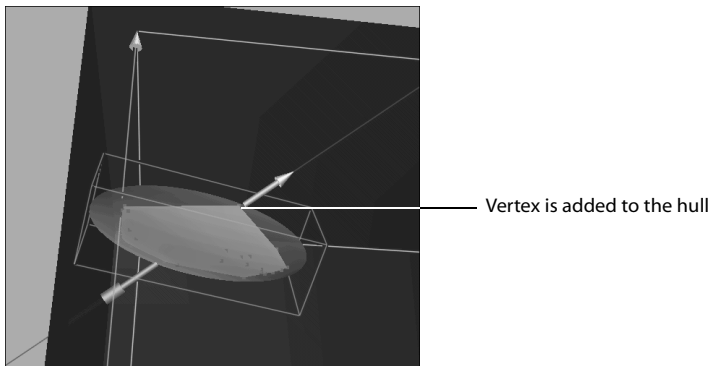
Before performing the following operations, set up the RGB viewer to best display the elements you are working with. Zoom in on the element, decide whether you want to show the histogram or not, and hide elements that are in the way.

To reshape the convex hull of the tolerance or softness, the ellipsoid must be displayed. It must be set to Ellipsoid, WireFrame or Cloud. Likewise, when reshaping patches, the patch box must be displayed. The box is displayed when the Display is set to Cloud, Box, or Box&Surf.

It is best to show the convex hull. Although it is not necessary to perform the operation (unless you are deleting vertices), you can see what you are doing more clearly. Refer to previous sections in this chapter, such as “Changing the Display of Key Elements” on page 719 for more information.

To add vertices to the hull and move them:

1. Select the key element by clicking on it.
2. Position the arrow of the selected element in the direction in which you want the ellipsoid or patch to grow or shrink. For example, point the arrow towards colours that you want to include in your sample. Alternatively, plot a point to position the arrow. See “Controlling the Arrow” on page 725 for details.
3. Press and hold **V** on the keyboard and click anywhere in the RGB viewer.
A vertex is added to the hull at the intersection of the arrow and the edge of the ellipsoid or patch.



NOTE: When using **V** in the RGB viewer, you can scale any convex hull (tolerance, softness or patches). If you use **V** in the image window, it will apply to softness only.

4. To add colours to the sample, drag the cursor to the right. This moves the vertex in the forward direction of the arrow.
To remove colours from the sample, drag the cursor to the left, moving the vertex backwards towards the centre of the element.

When you are satisfied with the result in the image window, release the cursor.

The hull is reshaped as you move the vertex. If you did not display the convex hull, it is temporarily displayed while you are moving the vertex. The direction of the arrow may change slightly as you move the vertex, but the direction it moves in is always the direction in which the arrow was initially pointing in.

NOTE: You can release the **V** key and the cursor, then press them again to scale the same vertex. You retain control over the new vertex until you change the position of the arrow.

To remove a vertex from the convex hull:

1. Display the convex hull of the key element, including the vertices. For information on displaying the convex hull, see “Changing the Display of Key Elements” on page 719.

NOTE: You do not have to select a key element to delete a vertex from its convex hull.

2. If necessary, zoom in on the element to distinguish the vertices more clearly.
3. Press and hold the - sign on the keypad (not the one on the keyboard) and click on the vertex that you want to remove.

See also “Techniques for Adjusting Softness” on page 735.

Animating Key Elements

You can animate the tolerance, softness, and patch ranges, as well as the colour, opacity and softness of patches.

Animating the Range of Key Elements

You animate the tolerance, softness, and patch ranges by changing the range at different frames. In the Channel Editor, range changes appear in the Shape channel. You can set a Shape keyframe by:

- Adding or subtracting pixel values from the range by sampling again in the image.
- Adjusting the key elements in the RGB viewer. All transformations made to a key element are applied to its shape keyframe, including translation, rotation, scaling, noise analysis, and so on.
- Changing the values in the Softness Scaling X, Y, and Z fields.

If you display the convex hull while animating a key element, the convex hull fills the ellipsoid in between keyframes. The sample is recreated to fill the ellipsoid on interpolated frames. This ensures the smoothest possible transition between keyframes.

To display the Channel Editor, click the Anim button. For animation basics, see Chapter 9, “Animation.”

NOTE: To animate key elements, the Auto Key button in the Preferences menu must be enabled. It is enabled by default.

Shape Keyframes

The shape of ellipsoids and patches is defined by their translation, rotation, and scaling values, so the shape keyframe is actually composed of multiple parameters. The Y value in the Channel Editor does not have the same significance as it does for a single parameter (for example, opacity). Instead, the Y value is composed of sequential numbers that represent keyframes that have been set. Each consecutive keyframe is assigned a sequential Y value: the first keyframe has a Y value of 1, the second has a Y value of 2, and so on. Although the Y value does not represent a single value, you can still adjust the curve to tweak the shape.

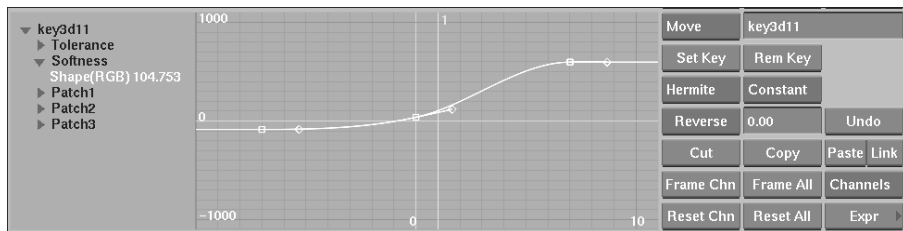
To animate the range of a key element:

1. Perfect the key at frame 1.
2. In the Preferences menu, make sure that the Auto Key button is enabled.
3. Scroll to other frames and, where necessary, adjust the range using any of the techniques provided in this chapter.

Keyframes are added at each frame where you change the tolerance, softness, or patch range.

NOTE: Whenever you set a tolerance keyframe, a softness keyframe is also set. This is because the softness range is always at least as large as the tolerance range: when you set the tolerance range, the 3D Keyer automatically creates a minimum softness range that is equivalent to the tolerance range. So when you adjust the tolerance, the softness range is also adjusted to accommodate the change.

4. Optionally, tweak the animation by adjusting the shape curve in the Channel Editor. The image updates as you make changes.



Changing the Interpolation

Parameter values in between keyframes are interpolated. Change the type of interpolation using the Channel Editor.

The default interpolation for the shape curve is Hermite, which creates a smooth transition between keyframes. If your clip has a very sudden change in the colour values (for example, if a light was switched on at a particular frame), use Constant interpolation between the two

keyframes where the change occurs. For more information on different interpolation types, see “Setting Interpolation” on page 152.

Animating the Patch Parameters

Animate the colour, softness, and opacity of patches by changing the values in the Patches menu. You will find channels for each of these parameters in the Patch folders of the Channel Editor.

To animate the patch parameters:

1. Go to the appropriate frame.
2. In the Patches menu, make sure that the Auto Key button is enabled.
3. Change the values in the Colour, Opacity or Softness fields as needed.

Resetting a channel

There are two ways to reset a channel:

- Using the R (Reset) button next to the Sampling box.
- Using the Reset Chn button in the Anim menu.

More Keying Techniques

In this section, learn how to adjust the key using various methods to manipulate the key elements (tolerance, softness and patches) in the RGB viewer, as well as the Softness Scaling Controls in the Key menu.

It is recommended to read “Basic Keying Technique” on page 702 and “Mastering the RGB Viewer” on page 717 before proceeding.

Techniques for Adjusting Softness

This section includes three additional methods for adjusting the softness of the key.

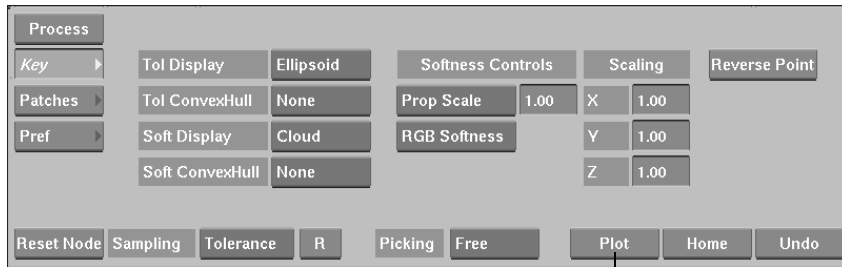
Scaling the Softness Ellipsoid to Remove Grey Areas

Increasing the softness may have created unwanted grey areas in the foreground subject. There are several ways to remove these grey areas. One method is to plot a pixel in a grey area, then translate, scale, or rotate the softness ellipsoid such that the plotted region is no longer within the ellipsoid. In effect, you will no longer be softening the plotted region.

If you translate, rotate, or proportionally scale the softness ellipsoid, all areas of the ellipsoid are affected to a degree. By contrast, when you use vertex scaling, you can affect a more localized area of the ellipsoid, leaving the rest of the ellipsoid unchanged. Vertex scaling is therefore often the most accurate method to use.

To scale the softness ellipsoid:

1. Show the Canvas (press **C**) and hide the Histogram (press **H**).
2. Use the Soft Display option box to change the display of the softness ellipsoid to Cloud.
3. Select the softness ellipsoid in the RGB viewer by clicking on it or selecting Softness from the Picking option box.
4. Display the CurResult clip to see the matte.
5. Zoom in to see the grey pixels up close.
6. Click Plot.



Plot button

7. Plot a grey pixel in the foreground subject by clicking it.
The colour value of the pixel that you clicked is plotted in the RGB viewer. The softness ellipsoid arrow passes through the plotted point.
8. To view the exact location of the plotted colour value in relation to the softness ellipsoid, rotate and scale the viewer.
The plotted point inside the softened region is represented by the softness ellipsoid.
9. Press **V**, click in the viewer and drag the cursor to the left to scale down the ellipsoid along the axis of the arrow. Watch the image update interactively. As soon as the grey area within the subject disappears, release the cursor to stop scaling the ellipsoid.
NOTE: If you are not satisfied with the result, click Undo to return the ellipsoid to its previous shape, then scale again.
10. If some grey areas remain, plot those areas and repeat the procedure.

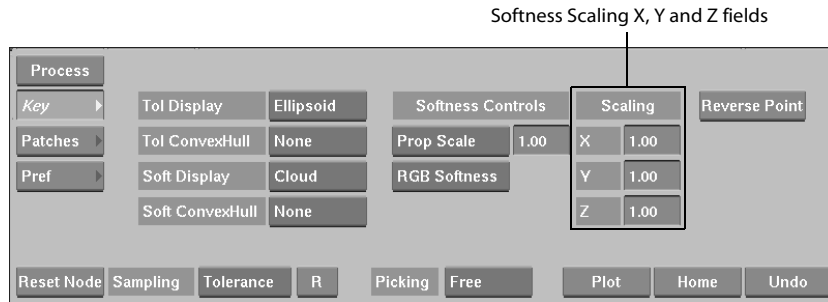
Scaling the Softness Ellipsoid to Add to Softness

You can also use the technique described in the previous section to increase the softness at the edges of the key. Start by identifying the region where you want to introduce more softness, and then plot a pixel in that area. You see the colour value in the viewer. Press the **V** hot key, then either enlarge or decrease the size of the softness ellipsoid minimally by dragging the cursor to the right or left. Include the colours you want and exclude the rest.

HINT: Display the 3D histogram to help you figure out the direction in which you want to scale the tolerance or softness.

Using the Softness X, Y, and Z fields

You can also adjust the softness in the key by changing the values in the Softness Scaling X, Y and Z fields. Drag the cursor over the fields, watching the result in the image window. You can often get a good result just by experimenting.



NOTE: The Sampling option box does not have to be set to Softness to use these fields.

Adjusting Transparencies with Patches

When keying transparencies, the transparency is sometimes either not visible enough or too prominent in the final composite. You can use patches to lighten or darken transparencies.

To adjust a transparency using a patch:

1. Use the basic keying technique to get the transparency as soft as possible.
2. Sample a patch within the transparency (see “Adding a Patch” on page 715).
3. View the final composite by displaying the CurResult view.
4. To lighten the transparency, set the Colour of the patch to 0 and the Opacity to 100, then lower the Opacity to your liking.
5. To darken the transparency, set the Colour of the patch to 100 and the Opacity to 0, then raise the Opacity until you are satisfied with the result.

Setting 3D Keyer Preferences

Auto Key

Enable this button to allow setting keyframes to animate the key. See “Animating Key Elements” on page 733 for details.

Regen

When you sample (softness, tolerance, and so on) using the colour picker or the numeric fields, you have the option of having the image update interactively as you move the cursor, or having the image update only when you release the cursor.

It is useful to update the image only after sampling if interaction during sampling is slow. This can happen if you are working with high resolution images, or if you are sampling while viewing a context from further along the pipeline. This can also occur in CurResult view, where a large amount of processing is required to display the result.

- To update the image interactively, enable Regen.
- To update the image after completing the sample, disable Regen.

Hot Keys

NOTE: Press and hold the hot key as you carry out the action, unless indicated otherwise.

3D Keyer Hot Keys

Press:

CTRL-click and drag on image

ALT-click and drag on image

CTRL+ALT and drag on image

N-click and drag on image

ALT+N, click and hold

O and release. Click on image or on histogram cube

T

S

ALT+1

ALT+2

ALT+3

To:

Sample with a rectangle.

Remove pixels from a sample.

Sample from scratch with a rectangle (removes previous sample range).

Sample for noise.

Scale softness based on the Minimize Noise analysis.

Plot a point on the image, or a histogram cube. A key element (the tolerance or softness ellipsoid, or a patch) must be selected to plot a histogram cube.

Select Sample Tolerance.

Select Sample Softness.

Select Sample Patch 1.

Select Sample Patch 2.

Select Sample Patch 3.

RGB Viewer Hot Keys

Press:	To:
H	Show or hide the histogram.
C	Show or hide the canvas.
L	Show or hide the RGB viewer outline and ellipsoid boxes.
B	Show or hide plotted points.
CTRL -drag viewer	Rotate the viewer.
ALT -drag viewer	Translate the viewer.
SHIFT -drag viewer	Zoom in on the viewer.
6 -drag item	Translate the selected key element in the direction of the arrow.
7 -drag item	Rotate the selected key element in any direction, around its centre point.
8 -drag item	Scale the selected key element proportionally (equally in X, Y and Z directions) from its center. Drag left to scale down and right to scale up.
9 -drag item	Scale the selected key element in the direction of the arrow (non-proportionally) from its center. Drag left to scale down and right to scale up.
0 - drag item	Scale the selected key element in the direction of the arrow (non-proportionally), starting from the point on the item's surface where the back end of the arrow protrudes.
O and release. Click on a histogram cube	Plot a point on a histogram cube. A key element must be selected.
- AND CLICK ON POINT	Remove a vertex from the tolerance, softness, or patch convex hull.
V	Add a vertex to a convex hull, scale the convex hull and perform vertex scaling of the softness directly on the image. Set the arrow direction, press V and click in the RGB viewer. Drag to the right to scale the hull larger in arrow direction; drag to the left to scale the hull smaller. (See "Reshaping the Convex Hull" on page 731 for details.)

Channel Editor Hot Keys

Press:	To:
A	Switch to Add mode.
M	Switch to Move mode.
P	Switch to Pan mode.
D	Switch to Delete mode.
SPACEBAR OR P	Switch to Pan mode. Click and drag in Channel Editor.
Z	Switch to Zoom mode. Click and drag left or right in Channel Editor.
CTRL	Switch to Rect Zoom mode. Click and drag a rectangle in Channel Editor.
B	Switch to Break mode.
X	Switch to XScale mode.
Y	Switch to YScale mode.

Garbage Masks and the Tracer

Time to take out the trash

When pulling a key, use garbage masks to isolate particular areas of an image for inclusion with, or exclusion from, the opaque area of the matte. You can animate garbage masks in many different ways.

Summary

In this chapter, you learn about:

- “Creating Garbage Masks” on page 742
- “Using Garbage Masks from the Modular Keyer and Batch” on page 756
- “Using Region of Interest (ROI)” on page 757
- “Pulling Keys with the Tracer” on page 758
- “Garbage Mask Hot Keys” on page 770

About Garbage Masks

A garbage mask is a special type of spline-based object you can draw directly on an image. You can use the mask to isolate regions of an image, to either key out or leave in a key. You do this by specifying whether you want the area inside or outside the mask to be opaque (to be included in the opaque part of the matte).

You can use garbage masks in many different ways to help create the best possible key. For example, use garbage masks on areas of the key-in clip that are difficult to key out, or to blend portions of the front and back clips.

Keying out unwanted white border with garbage mask



Matte before garbage mask is applied



Matte after garbage mask is applied

You create and animate garbage masks using the tools found in the Garbage Mask menu, which can be accessed from the following modules:

- The traditional Keyer
- The Modular Keyer
- The Batch module

When you access the Garbage Mask menu from Batch or the Modular Keyer, two additional features are provided — the Tracer and Region of Interest (ROI).

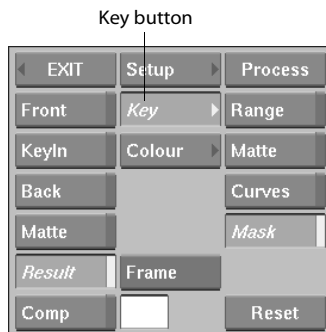
NOTE: It is recommended to have a basic understanding of these modules before learning about garbage masks.

Creating Garbage Masks

This section describes the basic procedure for creating garbage masks using the Garbage Mask menu in the traditional Keyer. To learn how to use the additional features found in Batch and the Modular Keyer, see “Using Garbage Masks from the Modular Keyer and Batch” on page 756, “Using Region of Interest (ROI)” on page 757, and “Pulling Keys with the Tracer” on page 758.

To open the Garbage Mask menu from the Keyer:

1. In the traditional Keyer, click Key.



2. Click Mask.

The Garbage Mask menu appears.



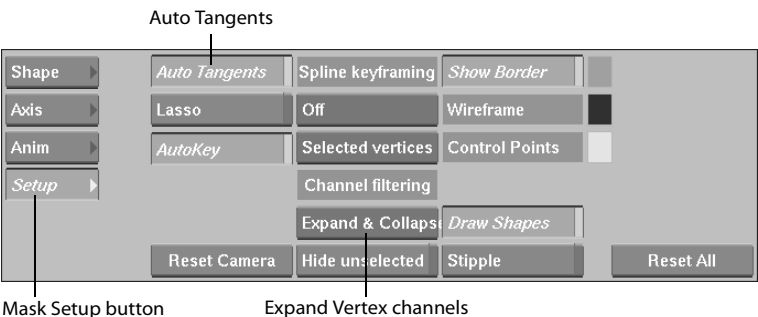
Setting Drawing Options

Before you create a mask, set options on how you want to draw mask, and how to display mask.

To set the drawing options:

1. Click Setup.

The Mask Setup menu appears.



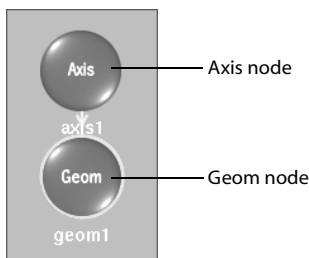
2. Set options as described in the following table.

Use:	To:
Auto Tangents	Position a tangent for each vertex set to create a smooth curve between vertices. When Auto Tangents is disabled, tangent is positioned under vertex, resulting in straight lines between vertices.
Lasso	Draw the mask freehand. With Lasso, you drag the cursor to draw the mask. A number of vertices are automatically added to the mask. When Lasso is disabled, you click at each location where you want to add a vertex to the mask.
AutoKey	Set keyframes automatically when you change animatable parameters. NOTE: Vertex parameters are not affected by Auto Key. For these parameters, use the Spline Keyframing controls. For details, see “Rotoscoping a Mask” on page 750.
Reset Camera	Reset camera parameters to their default values. For information on using the Camera, see Chapter 44, “Action: Camera and the Scene.”
Spline	See “Rotoscoping a Mask” on page 750.
All Vertices	See “Rotoscoping a Mask” on page 750.

Use:	To:
Expand Vertex Channels	<p>For each mask, control the display of vertices and their animatable parameters in the Channel Editor. These include all parameters in the vertex hierarchy.</p> <ul style="list-style-type: none"> • Expand Only expands and highlights vertices as they are selected. • Expand & Collapse expands and highlights vertices as they are selected and collapses all other channels. • No Expand does not expand selected vertices. You must expand them manually, and select an element in the hierarchy to highlight the element.
Hide Unselected	For each mask, hide vertices that are not selected (whether they are expanded or not). Takes effect when you make a selection.
Show Border	Display the Softness Offset wireframe border, defined in the Shape menu (this does not apply to the Tracer).
Show Border, Wireframe, Control Points and Picker Colour Pots	Set the colour of each of these mask elements.
Draw Shapes	Display the mask wireframe, vertices, and tangents.
Stipple	Stipple all wireframes (they appear as dashed lines).
Reset All	Remove all garbage masks.
Grab, Flush and Split On	See “The Reference Buffer” on page 81 for a description of this feature (available only when accessing the Garbage Mask menu from the Gmask node in the Modular Keyer and Batch).

Drawing a Mask

When you create a garbage mask, an Axis node and Geom node are added to the Keyer schematic.

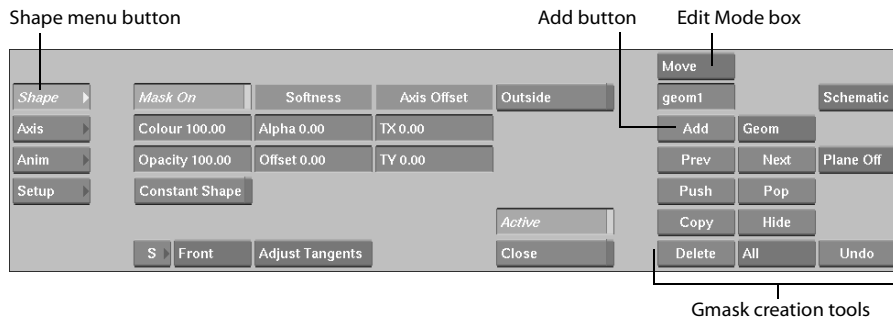


The Axis node contains all the rotation, scaling, and position data, and the Geom node contains all the information about how the mask will affect the image (softness, opacity, alpha, axis offset). For more information on Schematic view, see “Using the Schematic” on page 955.

To draw a garbage mask:

1. Open the Shape, Setup, or Axis menu by clicking the appropriate Garbage Mask menu button.

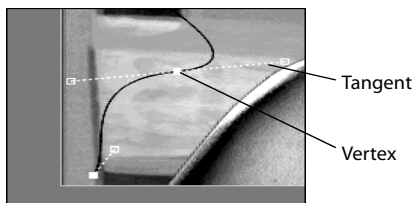
The Gmask creation tools appear to the right of the menu.



2. Click the Add button to add a new garbage mask. Alternatively, select Create from the Edit Mode box.
3. Display the Front, KeyIn or Matte view.
4. Click on the image to set the first vertex.

NOTE: If you are drawing a mask with the Lasso option, press down on the cursor and draw the mask freehand. When you have finished the mask, release the cursor and the mask closes automatically.

5. Click again to add the second vertex.



NOTE: If you are not in Auto Tangent mode, you can still set a tangent while adding a vertex, by dragging the cursor before releasing it.

6. To close the mask, click the Close button or click on the first vertex.

If you want to view the two new nodes, press ~ to toggle to Schematic view.

HINT: You can also close the mask by pressing the hot key for these Edit modes: **M** for Move, **S** for Select, or **B** for Break. Press the hot key for the mode that you want to use next, and the mask closes and you are ready to work in that mode.

Setting Mask Characteristics

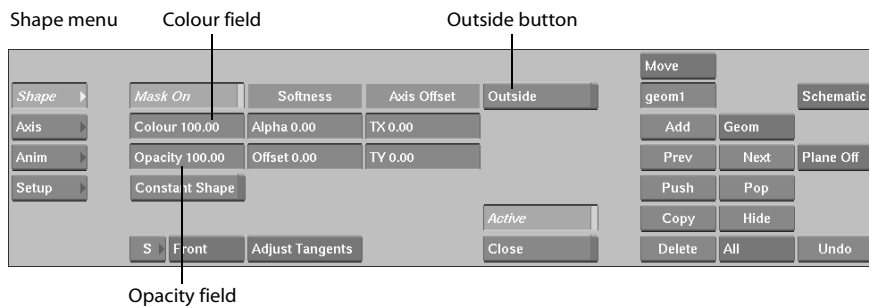
The Geom node contains all the information that controls how a mask affects the matte or composite clip, such as:

- Opacity, colour, edge softness of the mask.
- Whether the effect is applied to the inside or the outside of the mask.
- Offset of the mask from its axis.

You can change these settings in the Shape menu. All the mask characteristics are animatable.

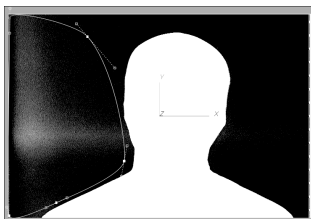
To set mask characteristics:

1. Display the Matte or Result view.
2. Select the mask in the image window (by clicking anywhere on the blue border) or in the Keyer schematic (by clicking on the Geom node).
3. Open the Shape menu.

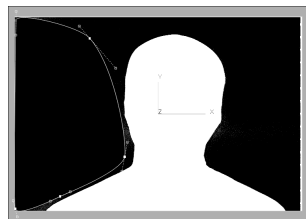


4. Enable the Outside button if you want to apply the effect to the part of the image that is outside the mask.
5. Set the mask opacity in the Opacity field.

The Opacity field defines the opacity of the matte in the area affected by the mask (that is, either inside or outside the mask). A value of 100% means the matte is completely opaque. A value of 50% means the matte is 50% transparent. A value of 0% has no effect on the image.



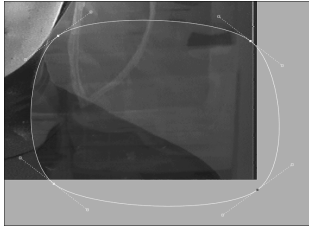
Mask with 0% opacity



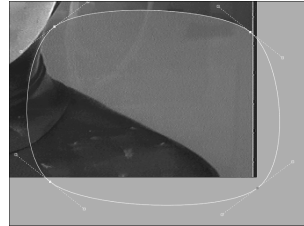
Mask with 100% opacity

6. Set the mask colour in the Colour field.

The Colour field defines the blend between the front and back image in the area that the mask is affecting (that is, either outside or inside the mask). A value of 50% is a 50/50 blend between the front and the back clip. A value of 100% is the front clip. A value of 0% is the back clip.



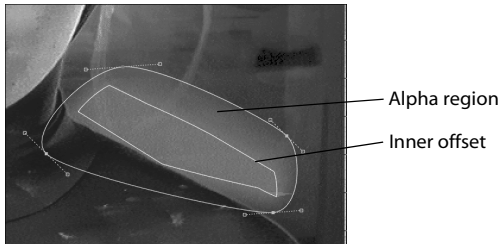
Mask with 25% colour



Mask with 75% colour

7. Set the softness of the mask edge using the Alpha and Offset fields.

The Alpha and Offset fields define the fade-out of the softness gradient from the edge of the mask. Use Offset to set the border position of the gradient and Alpha to set the gradient transparency.



8. To offset the mask relative to its axis, use the Axis Offset fields.
The Axis Offset fields define the mask's X and Y offset from its axis.
9. If you do not want to animate the mask, and want to be able to modify its shape without setting keyframes, enable the Constant Shape button. This forces all animatable parameters (except the Tracer parameter Sample On) to be set for the whole clip rather than only for the current frame. It also removes any existing keyframes and keeps the shape of the current frame only.

Turning a Mask On and Off

You can turn one or more masks on or off as you work. A mask that is turned off can be seen in the image window, but it has no effect on the image. To turn off a mask, select the mask then disable the Mask On button. Masks are turned on by default.

NOTE: This parameter is not animatable.

Changing the Priority Order of Masks

When you create several garbage masks on an image, you can change the order in which they are drawn, or layered in the scene. This affects the resulting image because a mask with a higher priority has precedence over one with a lower priority.

Use the Push and Pop buttons to control the drawing priority of garbage masks. Push moves a mask down a layer and Pop moves a mask up a layer.

NOTE: You cannot animate the priority by setting Push and Pop at different frames. When you set a mask's priority with Push or Pop, it is set for the entire clip.

To set a mask's priority:

1. Select the mask.
2. Click Push to move the mask down a layer, or Pop to move the mask up a layer.

Transforming the Mask as a Whole

You can apply transformations such translation, rotation, scaling, and shearing to a garbage mask. The transformation data for a mask is stored in its Axis node. You apply transformation changes to the mask using the Axis menu.

Axis menu

Shape ▶	PosX 0.00	RotX 0.00	ScaleX 100.00	ShearX 0.00	CenterX 0.00
Axis ▶	PosY 0.00	RotY 0.00	ScaleY 100.00	ShearY 0.00	CenterY 0.00
Anim ▶	PosZ 0.00	RotZ 0.00	ScaleZ 100.00	ShearZ 0.00	CenterZ 0.00
Setup ▶	MotionPath		Prop Scale		
<div> S ▶ Track Front Rot Off Scale Off Adjust Offset </div>					

NOTE: The Axis menu is similar to the Axis menu in Action.

Use the transformation fields to modify the position, shape, and size of the mask. Use the Center X, Y and Z values to change the position of the axis in relation to the mask. All these parameters are animatable. You can also use a motion path to animate the position of a mask. For more information, see “Understanding Motion Path Animation” on page 141.

Working with Tangents and Vertices

Use the options in the Edit Mode box to manipulate the vertices on the mask. You can edit the mask most easily in Front, Key-In or Matte view.

Edit Mode box



Selecting Vertices and Tangents

You can select vertices and tangent handles when the Select, Move, or Scale mode is active:

- To select an individual vertex or tangent handle, click on the vertex or tangent handle.
- To select multiple vertices, press the **CTRL** key and drag a selection box over a series of vertices.
- To add another vertex to your multiple selection, press **SHIFT** and click on the vertex.
- To add several more vertices, press **SHIFT** and **CTRL** and drag a selection box over the additional vertices.
- To de-select all vertices, click anywhere outside the mask.

Selecting Pickers and Softness Vertices (Tracer Only)

- To select one picker or softness vertex, click on the element.

To select multiple pickers or softness vertices:

1. Select the mask vertices that the pickers or softness vertices are associated with (see previous section).
2. Click on a picker or softness vertex corresponding to one of the selected mask vertices.
The pickers or softness vertices are selected for all the selected mask vertices.

Moving Vertices and Tangents

You can move vertices and tangents using the Move and Scale modes:

- To move a vertex or tangent in any direction, use Move mode. Click on the vertex or tangent and drag.
- To move selected vertices in any direction, use Move mode. Click on one of the selected vertices and drag.
- To move one or more selected vertices in the direction perpendicular to their tangents, use Scale mode. Click on one of the selected vertices and drag.

Breaking Tangents

You can separate two tangent handles (“break” the tangent) and move them separately using Break mode:

- To break and move a tangent handle, select Break mode and click on the tangent handle. The tangent is now displayed as a solid line, indicating it is “broken.”
- To re-connect two broken tangent handles, select Auto mode and click on either of the two tangent handles. The tangent is displayed as a dotted line.
- To change the position of an individual tangent handle after releasing cursor, use Move mode.

Removing and Adding Tangents

You can also use Break mode to “remove” tangents from vertices by clicking on the vertex while in Break mode. When you remove a tangent, the curve defined by the tangent is removed.

The shape of the curves changes towards a broken tangent depending on whether the adjacent vertices are broken or not:

- If both adjacent vertices are unbroken, their border lines will curve as they reach broken tangent.
- If both adjacent vertices are broken, the border lines will be straight as they reach the broken tangent.

You can create a garbage mask composed entirely of straight edges by removing the tangents from all the vertices.

- To remove the tangent of a vertex, select Break mode and click on the vertex.
- To add a tangent back to a vertex, select Auto and click on the vertex. Alternatively, in Break mode, click on the vertex and “drag” the tangent out again.

Automatic Adjusting Tangents of Adjacent Vertices

When you move vertices, the tangent handles of the two adjacent vertices remain fixed in their current position. You can make them automatically adjust to create the smoothest curves between the vertices, by pressing the **G** hot key as you move vertices around.

Rotoscoping a Mask

You can animate the shape of your mask by moving vertices at different frames throughout the clip. Each vertex has its own X, Y and Z position channels in the Channel Editor, as well as several other parameters, as shown in the following table.

Parameter	Channel Editor Folder & Name(s)	Channel Editor Values
The position of the mask vertices	position: X, Y, Z	
The position of left and right tangent handles relative to the mask vertex.	tangents: left, right	

Parameter	Channel Editor Folder & Name(s)	Channel Editor Values
The tangent continuity (handles broken or unbroken).	tangents: continuity	0 = broken handles 1 = unbroken handles
The active/inactive status of mask vertices. See "Adding and Removing Vertices" on page 752 for more details.	Vertex_x: active	0= inactive 1 = active

For information on animating with the Tracer, see "Animating a Tracer Mask" on page 765.

Setting Spline Keyframing Options

Before you start to animate the mask, set the Spline Keyframing options in the Setup menu. These options provide several different ways of working by letting you specify which elements will get a keyframe as you animate the mask.

Vertex Options

As you adjust your mask, you can opt to apply keyframes to selected vertices or to all vertices. To quickly animate a mask, it can be useful to keyframe all vertices every time you adjust a single vertex. This way, what you see at a given frame will be exactly what you get no matter how you animate the "surrounding" frames.

To have more control over the shape of the mask frame by frame, you may want to animate a single vertex at a time. This method is especially useful for tracking garbage masks and keeping control over tangent animation (see "Animating Selected Vertices on a Mask with Tracking" on page 755).

All Vertices — When you set a keyframe for a vertex parameter (or a parameter of a related element such as a picker), a keyframe for that parameter is set for all the vertices on the mask.

Selected Vertices — When you set a keyframe for a vertex parameter (or a parameter of a related element), a keyframe for that parameter is set for all selected vertices on the mask.

Spline Options

Spline & Tracer — When you set a keyframe for a parameter of one or more vertices (or related elements), keyframes are set for all the parameters of the vertex or vertices and related elements, with the exception of the Sample On parameter. (this option is only available when using the Tracer)

Spline — When you set a keyframe for a parameter of one or more vertices (or related elements), keyframes are set for the specified parameter plus for the mask vertex position and the tangent position channels.

Item — When you set a keyframe for a parameter of a vertex (or of a related element), only that parameter gets a keyframe.

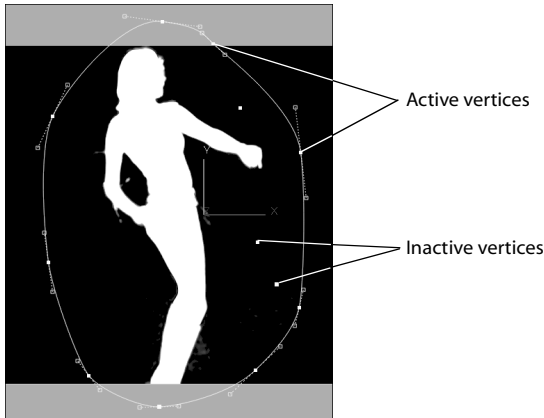
Off — No keyframes are set.

By combining the Vertex and Spline options in different ways, you can set an animation mode to suit the task at hand. For example, by choosing All Vertices and Item, you could animate just the tangent handles of all vertices on the mask.

Adding and Removing Vertices

To further control the shape, you can add and remove vertices from your mask at any frame of the clip. For example, if the shape you are masking changes radically or gets bigger part way through the clip, you can add the vertices you need to reshape the mask at that frame. When you add a vertex part way through the clip, it is added to the whole clip, but respects the existing shape of the mask in all previous frames and subsequent frames. Therefore, any careful reshaping that you have done to the mask is not affected by the new vertices.

If you delete a vertex, it becomes “inactive” from that frame to the next active/inactive keyframe set for that vertex, or, if there are no keyframes, to the end of the clip. An inactive vertex has no effect on the shape of the mask. On previous frames, the vertex is still active, so the mask shape is unchanged by deletion. To fully delete vertex (throughout the whole clip), click it a second time.



Use the tools in the Edit Mode box to add and delete vertices from the mask, and to control which vertices are active or inactive.

To add a vertex to the entire mask:

1. Switch to Add mode.
2. On any frame in the clip, click on the mask where you want to add the vertex.

The new vertex is added, and is active throughout the clip. It does not affect the existing mask animation, regardless of where in the clip you add the vertex.

To make an active vertex inactive:

1. Switch to Delete mode.
2. Click on the vertex.

The vertex is now inactive from the current frame to the next frame where an Active/Inactive keyframe has been set, or, if there are no keyframes, to the end of the clip. The status of the vertex on previous frames is unchanged.

NOTE: Alternatively, use the Active button in the Shape menu to make vertices inactive. Select the vertex you want to make inactive (you must be in Select, Move, or Scale mode), and then click the Active button.

To make an inactive vertex active:

1. Switch to Add mode.
2. Click on the inactive vertex.

The vertex is now active from the current frame to the next frame where an Active/Inactive keyframe has been set, or, if there are no keyframes, to the end of the clip. The status of the vertex on previous frames is unchanged.

NOTE: Alternatively, use the Active button in the Shape menu to make vertices active. Select the vertex you want to make active (you must be in Select, Move, or Scale mode), then click the Active button.

To delete a vertex from the mask:

1. At any frame, switch to Delete mode.
2. Click on the vertex. It becomes inactive.
3. Click on the vertex a second time.

It is removed from the mask for the duration of the clip.

Setting the Interpolation Mode for Animating the Mask

The way that the shape of the mask changes between frames is controlled by the interpolation mode assigned to the vertex position channels. For example, if you use a Linear interpolation, vertices will travel in a straight trajectory between keyframes, which would be suitable for an object with straight edges. With a Hermite interpolation, vertices travel on a curved path.

Use the Curve Interpolation box in the Channel Editor to set the mode.

Animating Masks with the Tracker

You can animate a mask by applying tracking data to it. To do so, you can either:

- Animate the entire mask by applying the data to the mask's axis.
- Animate selected vertices according to reference points that you set in the Tracker.

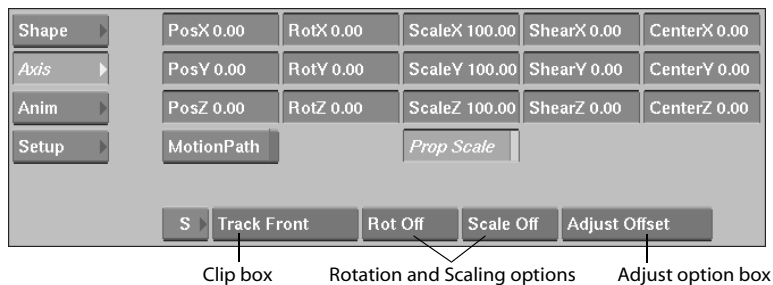
When you apply tracking data to a mask, only the mask axis, or position of selected vertices, is animated. Any other keyframes that have been set for the mask are disregarded. You can, however, animate individual vertices after you have applied tracking to your mask's axis or to individual vertices.

Animating Entire Masks with Tracking

You can track an object on your front or back clip and apply the tracking data to the axis of a mask, or to a hierarchy of masks.

To apply tracking data to the entire mask:

1. Select a mask.
2. Click the Axis button.
The Axis menu appears.
3. In the Clip box, select the clip you want to track. Select Track Front to track the front clip or Track Back to track the back clip.



4. Set the Rotation and Scaling options. To track rotation, set the option to Rot On; to track scaling, set Scale On.
5. Set the Adjust option box:
 - Select Adjust Offset if the selected mask is parented to one axis.
 - Select Adjust Axis if the selected mask is parented to a hierarchy of objects.
6. Go to the frame where you want to start tracking.
7. Click the S button.
The Stabilizer appears.

8. Position the Trackers.

NOTE: The first Tracker is for the position data and the second Tracker is for rotation and scaling data. If rotation and scaling were not selected, the second Tracker has no effect.

9. Click the Analyze button.

Fine-tune your analysis if necessary.

10. Click EXIT.

The Keyer reappears. The tracking data is applied to the mask.

11. Fine-tune your mask if necessary.

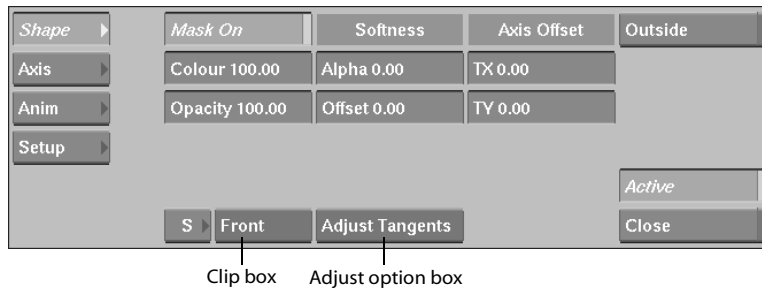
Animating Selected Vertices on a Mask with Tracking

You can apply tracking data to individual vertices on your mask so that the mask border follows a shape in the clip defined by reference points. Each vertex that you select is assigned a tracker box in the Stabilizer. The vertices are repositioned according to the reference points that you set in the Stabilizer.

When you have finished tracking the individual vertices, you can tweak their animation in the Garbage Mask menu. It is especially useful to manually adjust the tangent handles of the vertices at different frames where needed.

To apply tracking data to individual vertices on a mask:

1. Select a vertex or a group of vertices on a mask.
2. Click the Shape button.



3. In the Clip box, select the clip you want to track. Select Front to track the front clip or Back to track the back clip.

NOTE: When tracking vertices on a mask using the Garbage Mask menu in the Modular Keyer or Batch, you can only track the front clip.

4. Set the Adjust option box. Select Adjust Tangents if you want the tangents for the selected vertices to be adjusted while the points are being tracked.

5. Go to the frame where you want to start tracking.
6. Click the S button.
The Stabilizer appears. The trackers are automatically placed in the position of the selected vertices. Reposition if needed.
7. Click the Analyze button.
Fine-tune your analysis if necessary.
8. Click the Save button.
The Garbage Mask menu reappears. The tracking data is applied to the selected vertices on the mask.
9. Fine-tune your mask if necessary.

Using Garbage Masks from the Modular Keyer and Batch

Two additional functions are available when you open the Garbage Mask menu from the Modular Keyer and Batch —ROI (Region of Interest) and the Tracer:

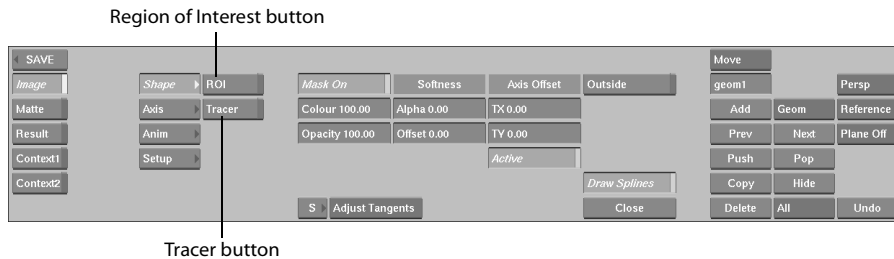
- Using ROI you can draw several masks that each reveal the portion of the matte that falls within the mask, and remove the portion of the matte that falls outside it, while respecting the area defined by the other masks.
- Using the Tracer you can pull mattes from images with fine edge detail that would be almost impossible in the traditional Keyer. The Tracer also provides a very effective method of adjusting the softness gradient at the edge of the mask.

To open the Garbage Mask menu from the Modular Keyer and Batch:

1. Add a Gmask node to the processing pipeline in the Modular Keyer or the processing tree in Batch.

NOTE: In the Modular Keyer, the default processing pipeline contains a Gmask node in the Matte branch.
2. Click the Gmask node then click the Edit button. Alternatively, double-click the Gmask node or press the **E** hot key.

The Garbage Mask menu appears.



For more information on using the Modular Keyer, see Chapter 34, “The Modular Keyer.”

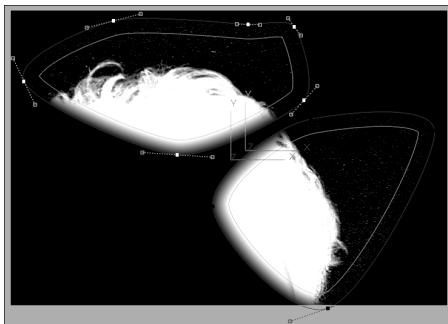
For more information on using Batch, see Chapter 29, “Batch Processing.”

Using Region of Interest (ROI)

Use Region of Interest when you want to have several masks that use the portion of the matte falling within them, and reveal the Back clip in the parts of the image that do not fall within any of the masks.

When ROI is not enabled, if you have several masks with Outside enabled and Colour set to 0.0 (black), the masks do not respect each others settings, so the entire image ends up being black (except for areas where all the mattes overlap). Using ROI, you can have any number of garbage masks—overlapping or not—with Outside enabled and Colour set to 0.0 and the revealed portion of the matte within each mask is respected by the other masks.

Two garbage masks with ROI enabled



The ROI button can be used either in Tracer mode or in regular Garbage Mask mode.

To draw more than one garbage mask revealing a matte:

1. Select Geom and click the Add button.
2. Draw the first garbage mask.

3. Switch to Result view.
4. Enable the ROI button.

flame automatically enables Outside and sets the Colour to 0.0 for the mask.

NOTE: Disabling ROI toggles the Outside and Colour settings back to what they were before.

5. Adjust offsets and softness as required.
6. Draw any other garbage masks as needed.

When ROI is enabled, it forces all masks in the image to have Outside enabled and Colour set to 0.0.

NOTE: Each time you add a new garbage mask with ROI enabled, Result view is temporarily cleared of all masks so that you see the entire matte. This way you have a better view of what you want to reveal and mask.

Pulling Keys with the Tracer

The Tracer is a powerful tool you can use to pull a key from scratch. It uses a system of localized keys called *pickers*, which analyses the colour information both inside and outside the mask. The Tracer then compares the values and uses a keying algorithm (the same one used in the 3D Keyer), to derive a key for the mask edge. This system allows you to key images with a lot of detail at the edges.

The Tracer is particularly useful for keying challenges such as:

- Very fine, wispy talent edges (hair, fur, or lace) where a blue- or green-screen background may show through.
- An object which was not shot on blue or green screen.

NOTE: The object must have sufficient (and relatively consistent) chroma and luma differences between its foreground and background.

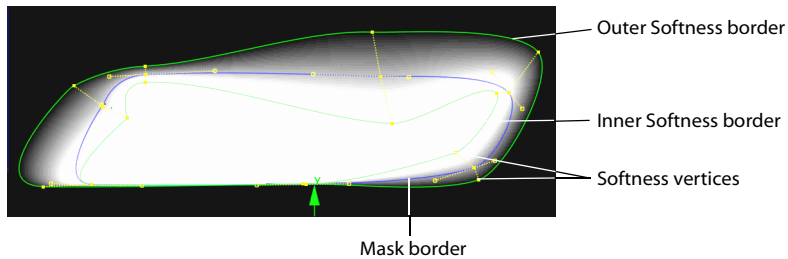
Advanced Gradient and Pickers

When using the Tracer, you can work in two different modes: advanced gradient and pickers. Individual vertices can be set to either of these modes. Advanced gradient is the default mode when working in the Tracer. That is, any vertex not specifically set for pickers uses advanced gradient.

Advanced Gradient

Advanced gradient provides a versatile method for setting the softness of the mask edge. It allows you to customize the softness gradient at different parts of the mask. This is made

possible by two softness borders, one inside and one outside of the mask border, with inner and outer softness vertices for each regular mask vertex.



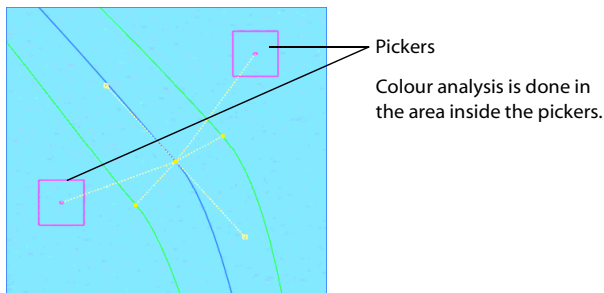
The softness gradient is bound by the two softness borders. You can customize the softness gradient at different parts of the mask by adjusting the distance of each softness vertex from the mask border.

Some potential uses for the Advanced Gradient tool are:

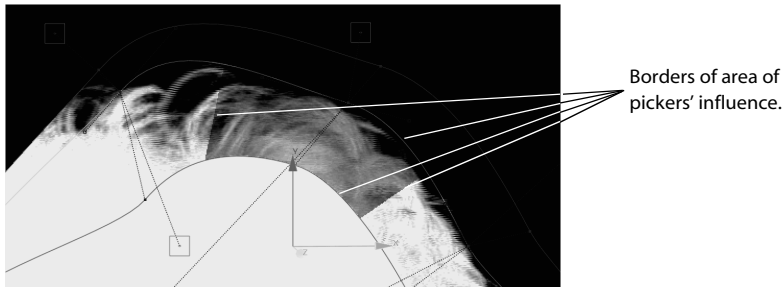
- Creating compositing effects when using layers in Action.
- Wire removal.
- When pulling a key with pickers (see next section), use on areas of the mask where pickers are not needed.

Pickers

When great control over the mask edge is needed, for example, for very fine edge detail, use pickers to effectively key out the background. The Tracer uses pairs of pickers to do luma and chroma analysis of the area inside and outside of the mask and derives localized edge keys from this information.



The area that each picker affects extends halfway towards the two adjacent mask vertices, and up to the two softness borders, as shown in the following example.



Using pickers, you can key objects that otherwise would be extremely difficult to key. Imagine a golden-beige horse with a very fine mane on an unfocused background of various colours. As long as you have some chroma/luma differences in the background (green vegetation, blue sky, black earth or rocks), you can 'force' the outside pickers to sample these colour values. The inside pickers can sample the golden-beige average values, and the Tracer can generate a soft-edged matte based on the difference between the two sets of values.

Mixing Advanced Gradient and Pickers

When the subject has a mixture of fine, wispy edges and hard, clean edges, use a mix of localized keys and advanced gradients. Pickers are better for the fine edge areas and advanced gradient is sometimes better for the hard edge areas. You can set the state of each vertex to advanced gradient (pickers off) or localized key (pickers on) mode.

- To convert a section of a mask from advanced gradient to pickers, select one or more vertices with no pickers and enable the Picker button. Adjust the pickers as needed.
- To convert a mask section from pickers to advanced gradient, select one or more vertices with pickers and disable the Picker button.
- Use hot key **0** (zero) to toggle selected vertices between the two modes.

To completely remove gradient (for hard edges):

1. Select the mask vertices in the area of the mask border where you want to remove the gradient.
2. Click on any one softness vertex corresponding to one of the selected mask vertices (either an inner or outer vertex).
This selects all the corresponding inner or outer softness vertices.
3. Press hot key **X** to switch to Scale edit mode.
4. Click on any one of the selected softness vertices and drag until the softness border is lined up with the mask border.

5. Repeat for the second softness border (inner or outer).

To separate the softness border from the mask border:

Press and hold hot key **Q** and then click on a mask vertex and drag the softness vertex out again.

Setting Mask Viewing Options

The Tracer menu contains several options that allow you to control the display of garbage masks.



Set the following options for each mask by selecting the mask then setting the option.

Use:	To:
Draw Pickers	Hide or display pickers.
Draw Borders	Hide or display softness borders.
Draw Splines	Hide or display softness borders, mask border and pickers.
Extra Tangents	Hide or display tangents on the softness vertices. You cannot adjust these tangents, but they provide a helpful visual reference by showing how adjustments to the shape affect the curves.

Using the Tracer on a Clip

Drawing a Tracer-aided matte entails four basic steps:

1. Analyzing the image to determine which areas are appropriate for pickers and which areas are better served by advanced gradients.
2. Drawing the garbage mask.
3. Assigning pickers to vertices where they are needed.
4. Making adjustments to softness borders, offsets, tangents, vertices, and pickers (where used).

To analyse the image:

Examine the edges of your talent to decide which portions would benefit from advanced gradient treatment and which would respond better to localized key treatment. If your clip has colours that change greatly throughout the clip, using pickers may be difficult. For more details see “Advanced Gradient and Pickers” on page 758.

To draw the garbage mask:

1. In the Modular Keyer or Batch, add a context point further down the processing pipeline or processing tree. You can then use the Context view to see the effect of the matte on the result image.

NOTE: When using the Tracer to pull a key from the Modular Keyer, you should remove the nodes before the Gmask node in the default pipeline.

2. Open the Garbage Mask menu (See “Using Garbage Masks from the Modular Keyer and Batch” on page 756.)
3. Select Image view. This view provides a reference image of the clip (it is not affected by your changes in the module).



Image view

Add button

4. Select Geom and click the Add button.
5. Draw a garbage mask around the subject and close it by clicking the first vertex drawn. Use as many vertices as required to adequately enclose the subject, but try to avoid excessive vertices. This will minimize unnecessary tweaking and manipulation later.

HINT: Place more vertices in areas with variation in the background.

To assign pickers to selected vertices:

1. Click the Tracer button.

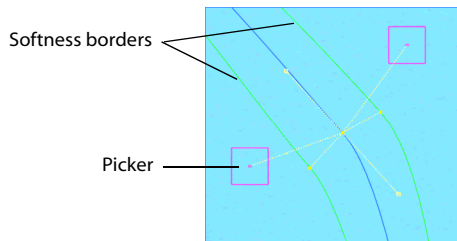
The Tracer menu appears, and two softness borders (the green lines) are added to the mask.



2. Select the vertices to which you want to apply pickers. For details, see “Selecting Vertices and Tangents” on page 749.
3. Enable the Pickers button.

NOTE: When one or more pickers are enabled, mask characteristics are automatically set to Outside disabled and a Colour value of 0 so that the mask will be white inside and black outside. Therefore do not set the Outside button unless you want to reverse the effect.

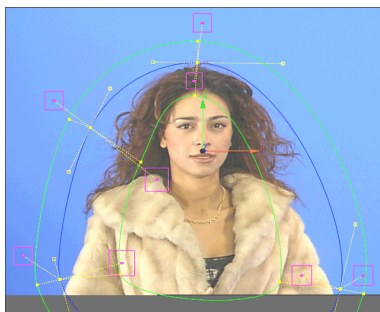
4. A pair of pickers are provided for each selected vertex.



- On vertices without pickers, the softness borders delineate the area where a softness gradient is applied.
- On vertices with pickers, the softness borders delineate the area to which softness is applied according to the picker values.

To fine-tune the matte:

1. Adjust the softness borders and mask border by moving the vertices and tangents:
 - The mask border should follow as closely as possible the general contour of the subject.
 - The outer softness border should completely surround all details that you want to include in the matte—all wispy details and edges must be within this line.
 - The inner softness border should be well within the area where softness control is needed.



When moving tangents, you can opt to have the pickers follow the tangent movement (the default behaviour), or be independent of tangent movement. To make pickers independent of tangents, switch to Break mode and click on either of the pickers. This is an animatable parameter (see “Animating a Tracer Mask” on page 765).

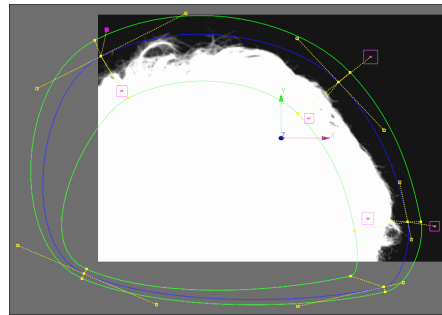
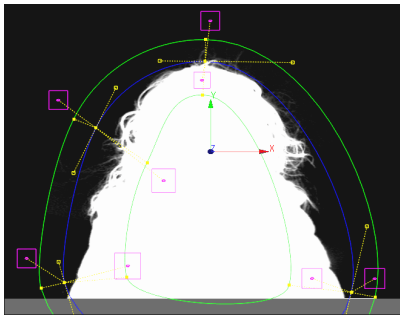
NOTE: If the tangents are broken, the pickers are automatically unlinked from the tangents.

2. For each pair of pickers, one requires a sample of the area outside the mask, while the other requires a sample of the mask interior.

Place one picker outside the subject to sample values you do not want to include in the matte (for example a blue screen), and place the other one within the subject for a colour value sample of an area you do want included in the matte. Try to select areas where the colour values do not change too much throughout the clip.

3. Click Matte or Result (Matte will provide a clearer view) to see your progress.
4. Scale the pickers by either:
 - Pressing the **TAB** key, clicking within the picker and dragging to the left to reduce the size or to the right to increase the size.
 - Selecting the picker and then dragging the cursor over the Sample Size field.
5. Move the pickers around and enlarge/reduce them in size to interactively see how you may obtain the best results. This will require toggling between the front and matte views.

The following figure illustrates the Matte view result of the pickers placed in the previous figures.



Animating a Tracer Mask

When using the Tracer, you can animate the vertex parameters detailed in the section “Rotoscoping a Mask” on page 750, and the following additional parameters.

Parameter	Channel Folder and Name(s)	Channel Values
The position of the two softness vertices relative to the mask vertex.	border: upper = outer vertex* bottom = inner vertex*	
Whether pickers are linked to tangents or not.	pickers: fixed	0 = unlinked 1 = linked
<ul style="list-style-type: none"> • Linked — pickers move with tangents when they are moved. • Unlinked — pickers remain in their current position when tangents are moved. 		
The on/off status of pickers.	pickers: active	0 = off, 1= on
The position of the pickers relative to the mask vertex.	pickers: upper = outer picker* bottom = inner picker*	
The picker size.	pickers: u-size = outer picker size* b-size = inner picker size*	
The Sample on/off status of pickers. See “Sample On” on page 766 for details.	sample: active	0 = Sample off 1 = Sample on
The interpolation mode for picker values when Sample is off. See “Interpolation Mode” on page 767 for details.	sample: interp.	0 = Constant 1 = Linear

*Channel Editor designations for “upper” and “bottom” assume mask was drawn in the clockwise direction.

Animating Picker Values

Two options are available to control how picker values (that is, the sampled colour values) are animated - Sample On and Interpolation Mode.

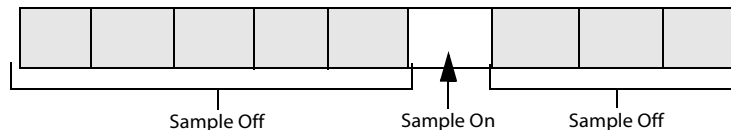


Sample On — Using Sample On, you can opt to have picker values resampled at every frame (the default), or turn off resampling for one or more frames. When Sample On is disabled at a particular frame, **flame** uses the picker values previously sampled at another frame. It is often a very useful strategy to adjust the size and position of pickers at one particular frame until the result is optimal, and then apply these values to other frames. In this way, if movement in the clip in other frames causes the area being sampled to move away from the pickers, you do not need to readjust the pickers to get a good reading.

You can enable or disable Sample On for one or more selected pickers.

Turning Sample off will work best when colour values remain fairly consistent throughout the clip. When this is the case, one picker sampling will often be enough for the entire clip.

Typical technique for using Sample On: one sample is used for the entire clip



When there is more colour variation in the clip, you may need to take samples at several frames.

To use one picker sampling throughout a clip:

1. Choose a frame in which the colour values are representative of the average colour values found in the clip.
2. Select the pickers you want to sample (see “To select multiple pickers or softness vertices:” on page 749)
3. At that frame, make sure Sample On is enabled (the default).
4. Set the picker values by setting their position and size until you get the optimal result.
5. Go to the next frame and disable Sample On.

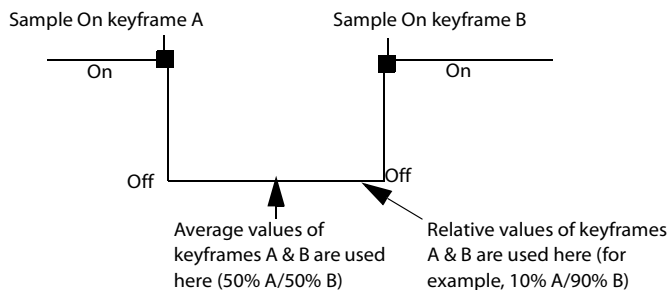
The sample values read at the frame you chose will be used to compute the softness for all other frames in the clip.

To take samples at several frames:

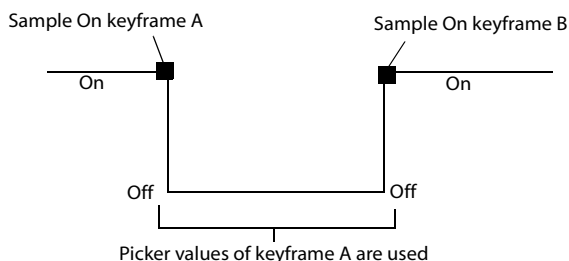
1. Follow the steps in the previous procedure to set a picker sampling for the clip.
2. Starting at the first frame sampled, examine your results frame by frame (either forward or backwards through the clip).
3. When you come to an unsatisfactory result, select the pickers you want to re-sample and click Sample On.
4. Adjust the picker position and size until you get a good result.
5. Go to the next frame and disable Sample On.
6. Continue verifying frames until you come to one that needs to be adjusted.
7. Repeat steps 3-5.
8. Repeat for the remainder of the clip.

Interpolation Mode — This option lets you specify how you want picker values to be interpolated between Sample On keyframes, that is, the interpolation at frames where Sample On is disabled. Choices are Linear or Constant.

- **Linear:** Picker values are interpolated between Sample On keyframes. The picker values for each frame are computed based on the values of the previous and next Sample On keyframes and the proximity of the frame to the previous and next Sample On keyframes.



- **Constant:** Picker values are fixed between Sample On keyframes. This means that the picker values at the first Sample On frame will be used for all subsequent frames up until Sample On is again enabled.



Tracking with the Tracer

Tracking with the Tracer is done using exactly the same method as with regular garbage masks. See “Animating Masks with the Tracker” on page 754.

NOTE: When animating selected vertices with the Tracker, it is the vertices that are doing the tracking, and not the pickers. Once you obtain the tracking data, you will need to go back through your clip from frame 1 onwards, making adjustments to the pickers where necessary.

Troubleshooting the Tracer

Every clip has its own challenges and requires a certain amount of tweaking and fine-tuning to achieve the best results.

Problem	One of the Tracer segments (localized keys) on my matte shows noisy black and white artefacts and nothing else. What’s wrong?
Possible cause	The two pickers are sampling luma and/or chroma values that are too similar.
Solution	Try moving one of the pickers to a differently coloured area, or enlarge/ reduce the size of one or both of the pickers to include more varied colour information. If there is simply no area in the clip that differs enough, you will need to use an advanced gradient for that segment.
Problem	One of the Tracer segments (localized keys) on my matte appears to be showing the black/white matte information reversed, or as a negative. What’s wrong?
Possible cause	You may have reversed the position of the two pickers.
Solution	Try exchanging the inside picker for the outside one and vice-versa.

Problem	I can't get a good colour sample from the area outside the mask edge in a certain portion of the image. What can I do?
Possible cause	There is simply none (or not enough) of the needed colour in the immediate area of that particular picker.
Solution	You can drag the picker somewhere further away from the vertex to get a better colour sample. The picker can be located anywhere on the image in order to facilitate the best possible colour sample. Do not, however, place a picker outside the image.
Problem	I'm getting really nice edge detail on my matte, but I'm also getting black/white holes in my matte. Is there anything I can do?
Possible cause	This may be caused by the generation of random noise and/or artefacts.
Solution 1	Try the Clean algorithm by clicking the box labelled Clean. You may want to try several different percentages to find the optimal level of cleaning that doesn't effect your edges (the default percentage is 70%). The Clean algorithm works only with the Tracer and pickers on. If you have advanced gradient segments, they will be unaffected. The Clean command works on all vertices with pickers, whether or not they are selected. The area affected by the Clean command is the same area that the pickers affect (see "Pickers" on page 759)
Solution 2	Use the inner border to isolate the problem areas.
Solution 3	Add a second garbage mask.
Problem	I'm repositioning and rescaling the pickers and the matte goes from having too much softness (overly transparent) to having too little (overly opaque). What can I do?
Possible cause	The sampled colour range is either too broad or too narrow.
Solution	Locate the areas that have a tendency to become too softened and the picker that is 'responsible' for this by moving the pickers until the matte results change for the worse. Include a larger portion of these softer areas in a picker box. By sampling a fuller range of chroma/luma in this area, a more uniform, averaged effect is created.

Garbage Mask Hot Keys

The following table lists some of the hot keys that are useful when creating garbage masks. Press and hold the hot key as you carry out the action, unless otherwise indicated.

HINT: To determine the hot key currently assigned to a particular button, press **CTRL+ALT+SHIFT** and click on the button. The hot key is shown in the message bar.

Press:	To:
Esc	Save the garbage mask and exit the Garbage Mask menu.
N	Switch to Create mode (cursor changes to green Add Mask cursor) and create a new garbage mask axis.
G and move a vertex	Force the tangents of adjacent vertices to adjust their positions (and therefore the curve of the spline) according to the new position of the vertex.
B	Switch to Break mode.
X	Switch to Scale mode.
A	Switch to Add mode.
U	Switch to Auto mode.
M	Switch to Move mode.
D	Switch to Delete mode.
Q and drag softness vertex	Separate softness vertices from mask vertices when they are close together or overlapping.
TAB and drag picker	Scale a picker.
1	View the Image clip (reference image).
2	View the Result clip in the image window.
3	View Context Point 1 in the image window.
4	View Context Point 2 in the image window.
5	View Action context (if the Modular Keyer was accessed from Action).
6	View the Matte clip in the image window.
9	Toggle the display of the splines and tangents (wireframes) of all garbage masks.
0	Toggle between advanced gradient mode and picker mode for selected vertices.
CTRL-drag	Select multiple vertices.
SHIFT-click	Add a single vertex to a multiple selection.

Press:	To:
CTRL+SHIFT-drag	Add multiple vertices to a multiple selection.
H	Hide Selected/All/Branch.
SPACEBAR	Pan in image window or Channel Editor. Click and drag in image window or Channel Editor.
Z	Zoom in image window or Channel Editor. Click and drag left or right in image window or Channel Editor.
C	Activate Rectangular Zoom (in Channel Editor).

This image shows a full page of white paper with horizontal dashed lines, resembling notebook paper. The lines are evenly spaced and run across the width of the page. There is no handwriting or other markings on the paper.

37

Compositing and Optics

The Sandwich Shoppe

Compositing is the process of blending together a front and back clip. To create a composite, a matte is applied to the two clips, and the front clip is superimposed on the back clip. You can apply glow effects to a composited clip using the Optics module.

Summary

In this chapter, you learn about:

“The Compositor” on page 773

- “The Quick Composite Module” on page 786
- “The Optics Module” on page 787

The Compositor

Use the Compositor to composite a front clip, a back clip, and a matte clip. It is similar to Action except that you can only have one layer and you do not have access to the Keyer. You can scale, rotate, translate, or shear the front clip in two dimensions. You can create animations in the Compositor, apply motion paths to the front clip and the camera in the scene, as well as move and animate the camera-eye position. In addition, you can colour correct all three clips, and track points on the back clip.

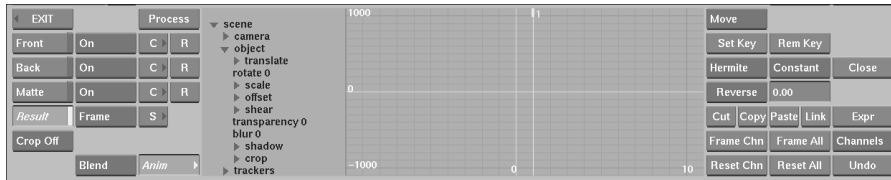
Accessing the Compositor

You load three clips into the Compositor: a front clip, back clip, and matte clip.

To access the Compositor:

1. Click Compositor in the Effects menu.
2. Select the front, back, and matte clips.
3. Select the destination.

The Compositor appears.



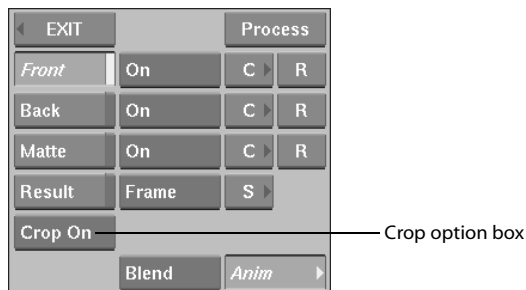
Cropping the Front Clip

You can crop portions of the front clip in the result clip, either to process as an effect, or to use temporarily as you work—to reduce processing time while working on the composite. Turn the crop box on or off any time as you are working.

You can set the borders of the crop box manually using the Channel Editor, or use Crop Auto to get a crop box that surrounds the white area of the matte clip.

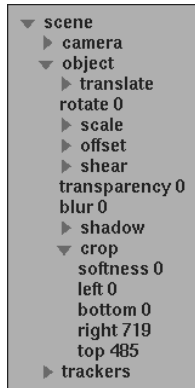
To crop the front clip manually:

1. Click Front to view the front clip.



2. From the Crop option box, select Crop On.
Notice the dotted crop border around the image.
3. Click Anim to display the Channel Editor.

4. Open the Object folder, and then the Crop folder.



5. Select a channel.

Select:	To Adjust:
Softness	The softness of the edge of the crop box.
Left	The position of the left border of the crop box.
Right	The position of the right border of the crop box.
Top	The position of the top border of the crop box.
Bottom	The position of the bottom border of the crop box.

6. Drag the cursor in the image window to change the value of the selected channel. You can also enter the values directly in the Channel Editor.

NOTE: When you modify crop box borders in the image window, you can modify two borders at once. For example, when you select the left or bottom channel, you can change both borders at once. To modify a single border, use the Channel Editor.

To crop the front clip using Crop Auto:

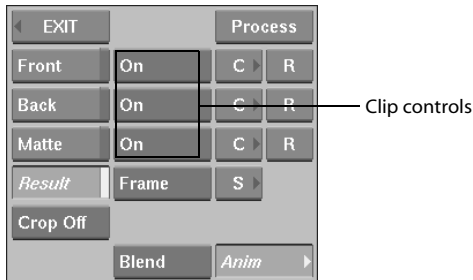
1. Click Front to view the front clip.
2. From the Crop option box, select Crop Auto.

To reset the crop box:

1. In the Channel Editor, click the Crop folder to select all crop channels.
2. Click Reset Chn.

Disabling and Locking the Clips

Use the clip controls to enable and disable clips, and to lock clips to a specific frame.

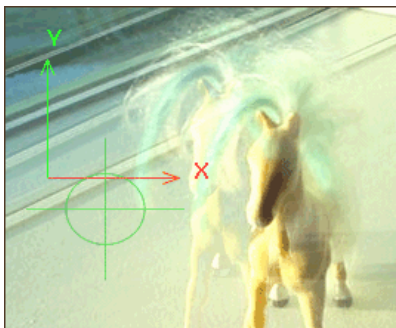


For more information, see “Changing a Layer’s Clips” on page 910 in Chapter 43, “Action: Layers and Surfaces.”

Controlling the Camera in the Scene

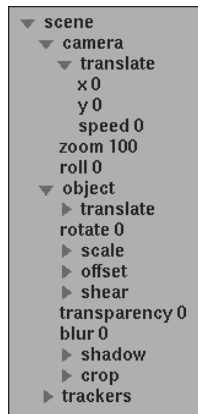
The Compositor includes a camera for the scene. You can change the camera’s position, zoom, and roll. For example, you can animate the camera channels to simulate a camera pan or zoom.

The crosshair that you see in the image window when Result view is selected represents the point of interest.



To change the camera's attributes:

1. Change to Result view.
2. Click Anim.
The Channel Editor appears.
3. Open the Camera folder, and the Translate folder.



4. Select a channel.

Select:

X or Y channels in the Translate folder
 Speed channel in the Translate folder
 Zoom channel
 Roll channel

To:

Change the camera's position.
 Change the camera's speed.
 Change the camera's zoom.
 Change the camera's roll.

5. Drag the cursor in the image window to change the value of the selected channel. You can also set the value in the Channel Editor.

The Speed channel value can only be modified in the Channel Editor.

NOTE: When you modify an X or Y channel in the image window, you can modify both channels. To modify a single channel, use the Channel Editor.

For more information on using the Channel Editor, see Chapter 9, "Animation."

Controlling the Front Clip

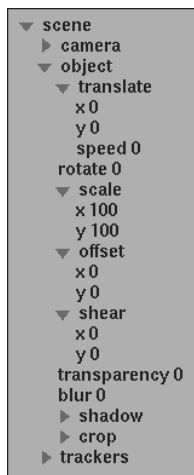
You can move, scale, rotate, skew, and offset the front clip. You can also blur, crop, add transparency, and add a shadow to the front clip.

To change the front clip's attributes:

1. Click Anim.

The Channel Editor appears.

2. Open the Object folder.



3. Select a channel.

Select:

X or Y channels in the Translate folder

Rotate channel

X or Y channels in the Scale folder

X or Y channels in the Shear folder

X or Y channels in the Offset folder

Transparency channel

Blur channel

To:

Translate the front clip.

Rotate the front clip.

Scale the front clip.

Skew the front clip.

Offset the front clip from its axis.

Change the front clip's transparency.

Blur the front clip.

4. Drag the cursor in the image window to change the value of the selected channel. You can also enter the values directly in the Channel Editor.

NOTE: When you modify an X or Y channel in the image window, you can modify both channels. To modify a single channel, use the Channel Editor.

For more information on using the Channel Editor, see Chapter 9, “Animation.”

Adding a Shadow to the Front Clip

You can add a shadow to the front clip. You can move, scale, rotate, and skew the shadow. You can also set the shadow's colour, softness, and transparency.

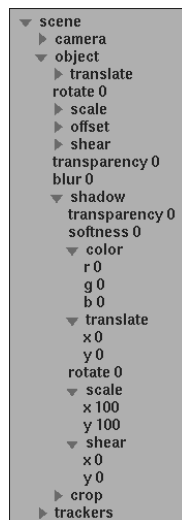
To add a shadow:

1. Disable Anim.

The Setup menu appears.



2. Enable Shadow.
3. Click Anim.
4. In the Object folder, open the Shadow folder.



5. The shadow is currently located directly underneath the image. Select the X or Y translate channel or the Rotate button and drag the cursor in the image window to move the shadow into view.

6. Select another channel.

Select:

Transparency channel
 Softness channel
 R, G, or B channels in the Colour folder
 X or Y channels in the Translate folder
 X or Y channels in the Scale folder
 Rotate channel
 X or Y channels in the Shear folder

To:

Change the shadow's transparency.
 Change the shadow's softness.
 Change the shadow's colour.
 Change the shadow's position.
 Scale the shadow.
 Rotate the shadow.
 Skew the shadow.

7. Use the other channels to create the desired shadow effect. You can work with the shadow in the image window or enter the values in the Channel Editor.

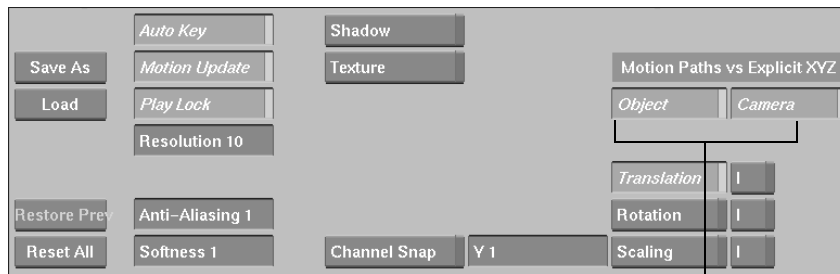
Using Motion Paths

In the Compositor, you can use motion paths to animate the translation of the front clip and the camera.

To create a motion path:

1. Disable Anim.

The Setup menu appears.



Object and Camera buttons

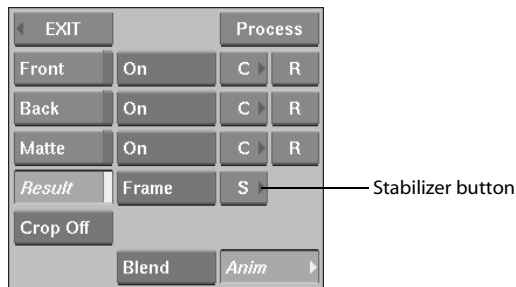
2. Enable Object to enable the motion path for the front clip.
3. Enable Camera to enable the motion path for the camera.
4. Click Anim.
5. Open the Translate folder in either the Camera or Object folder.
6. Select either the X or Y channel.

7. Set the position of the front clip by dragging it in the image window. You can also do this in the Channel Editor.
8. Move to a different frame.
9. Set the new position of the front clip by again dragging it in the image window.
A motion path appears.

For more information on motion paths, see Chapter 9, “Animation.”

Tracking Your Clips

You can track one or two points on your back clip and apply that data to your front clip. Use one-point tracking to apply translation data to the front clip, and two-point tracking to apply translation and scaling and/or rotation data. For more information on tracking, see Chapter 38, “Tracking and Stabilizing.”



To track your clip:

1. Click S.
The Stabilizer appears.
2. Track your clip.
3. Click the Save button to save the analysis and return to the Compositor.

NOTE: To return to the Compositor without applying the tracking data to your front clip, hold down the Save button and select Cancel.

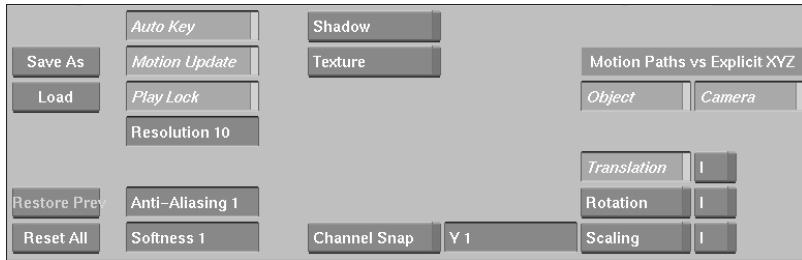
Applying the Tracking Data

Once you track your clip, you must set how that data is applied to the front clip.

To apply the tracking data to the clip:

1. Disable Anim.

The Setup menu appears.



2. Enable Translation to apply the translation data to the front clip.

You can move the tracker box for Tracker1 in the image to adjust the tracking data.

3. Enable | next to the Translation button to invert the translation data being applied to the front clip.

4. Enable Rotation to apply the rotation data to the front clip.

The rotation is calculated by the relative position of the two trackers in each frame.

5. Enable | next to the Rotation button to invert the rotation data being applied to the front clip.

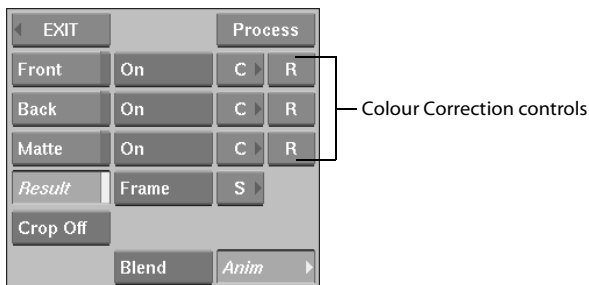
6. Enable Scaling to apply the scaling data to the front clip.

The scaling is calculated by the relative position of the two trackers in each frame.

7. Enable | next to the Scaling button to invert the scaling data being applied to the front clip.

Colour Correcting Your Clips

You can colour correct the front, back or matte clips.



To colour correct your clips:

1. Click Result to view the composite.
2. Click C corresponding to the clip that you want to colour correct.
The Colour Corrector appears.
3. Modify your clip. For more information, see Chapter 26, “Colour Corrector.”
4. Click Save to return to the Compositor.

NOTE: To reset the clip’s colour values, click the corresponding R (reset) button.

Animating Your Composite

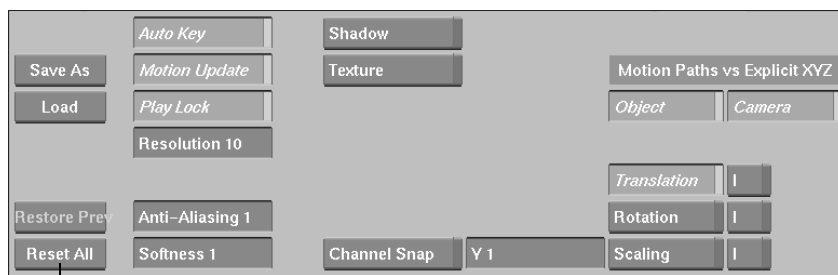
Use the Channel Editor to animate the following parameters:

- Camera position, speed, roll, and zoom
- Front clip translation, rotation, scaling, and shearing
- Front clip shadow
- Front clip blur
- Cropping
- Tracking data

For more information on using the Channel Editor, see Chapter 9, “Animation.”

Setting Preferences

Use the Setup menu to specify preferences, and to save and load setups and preferences.



Reset All button

Reset All

Click Reset All to reset all setup parameters and preferences to their default values.

Resolution

When working with layers and playing clips in the Compositor, you work with image proxies. You set the resolution of the image proxies using the Resolution field in the Setup menu. To increase the resolution of the image proxies, decrease the value in the Resolution field. A lower value requires more processing time to generate proxies.

NOTE: When hardware texture mapping is enabled using the Texture option, the Resolution option is not applicable, as the hardware controls resolution during interaction and playback.

Anti-Aliasing and Softness

The jagged lines that often occur along the edges of diagonal or curved lines when processing high-frequency images such as text are the result of aliasing. Aliasing is caused by insufficient spatial sampling of the image. The process of reducing or removing the effects of aliasing by increasing the sampling rate is called anti-aliasing.

Use the Anti-Aliasing and Softness fields to control the level of anti-aliasing. Increasing the number of samples increases the degree of anti-aliasing but also increases the processing time. Adjusting the softness also affects the degree of anti-aliasing.

The best results are obtained when the softness value is set as indicated in the following table.

Table 1: Anti-aliasing and Softness Values

Anti-aliasing	Softness
1	See Note
2	1
3	2
4	2

NOTE: When the Anti-Aliasing value is 1, no anti-aliasing occurs. In this case, the Softness value has no effect on the image.

Although the cleanest images are obtained when the softness value is set to 1 or 2, higher softness values can also be used for creative effects.

Shadow

Enable the Shadow button to place a shadow behind your front clip. Shadow attributes such as position, rotation, scaling, softness, transparency, and colour can be set and animated in the Channel Editor.

Other Preferences

For information on other preferences, see the following sections.

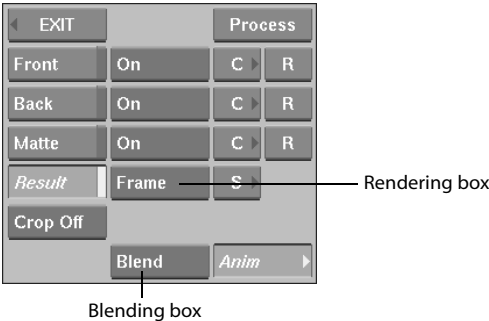
Preference:	See:
Auto Key	"Using the Main Setup Menu" on page 896.
Motion Update	"Using the Main Setup Menu" on page 896.
Play Lock	"Using the Main Setup Menu" on page 896.
Texture	"Using the Main Setup Menu" on page 896.
Snap	"Using the Main Setup Menu" on page 896.
Channel Snap	"Using the Main Setup Menu" on page 896.
Object & Camera	"Using Motion Paths" on page 780.
Translation, Rotation & Scaling	"Applying the Tracking Data" on page 782.

Saving, Loading, and Removing Setups

Use the Save button in the Setup menu to save setups and preferences. Use the Load button to load and remove setups and preferences from the libraries. When you use the Save button to the right of the Setup name box to re-save an existing setup, you can use the Restore Prev button to restore a backup copy (the version saved before the last save). For more information, see Chapter 7, "Saving Setups and Preferences."

Processing Your Clips

When you finish creating your composite, use the options in the Compositor menu to improve the quality of your rendered clip.



To render a clip:

1. Select either Field or Frame in the Rendering box.

If you select Frame, the Compositor combines the two fields and renders each frame. If you select Field, the Compositor renders both fields separately. Use the Field option if the two fields in the image are not properly aligned.

2. Select either Blend or Add in the Blending box.

The Blend option uses both the front and back luminance curves of the matte clip. The Add option only uses the front luminance curve, and is used to eliminate the black fringe created by using a front clip with a black background.

3. Click Result.
4. Click Process.

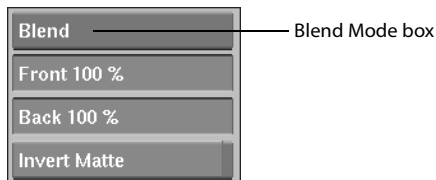
The Quick Composite Module

Use the Quick Composite module for one-step compositing of a front and back clip using a matte clip.

To use the Quick Composite module:

1. Click Quick Composite in the Effects menu.

The Quick Composite menu appears.



2. Select the blend mode from the Blend Mode box. You have a choice of three modes for blending the front and back clips.

Select:	To:
---------	-----

Blend	Use subtractive blending.
-------	---------------------------

Additive	Use additive blending. Select this option if the front clip has a black background, as with a computer-generated image. This option eliminates the black fringe at the edge of the front image.
----------	---

FrontMin	Prevent loss of detail from the front image. For example, if the front clip is a human model, select the FrontMin option to prevent losing details such as hair in the composite clip.
----------	--

3. You can use the Front and Back fields to increase or decrease the luminance of the front and back clips. The values specify the gain for the clips. Quick Composite multiplies the luminance values in the clips by the gain values you set. The resulting luminance values are clipped at the maximum and minimum values of 255 (white) and 0 (black), respectively.

Gain is expressed as a percentage value. The default value of 100% has no effect on the image since the luminance values are multiplied by 1. To increase the gain, set the value above 100%. To decrease the gain, set the value below 100%. Adjust the gain for the front and back clips as needed in the Front and Back fields.

4. You can enable the Invert Matte button to invert the matte if required.
5. Select the front, back, and matte clips.
6. Select the destination.

The composite clip appears on the destination reel.

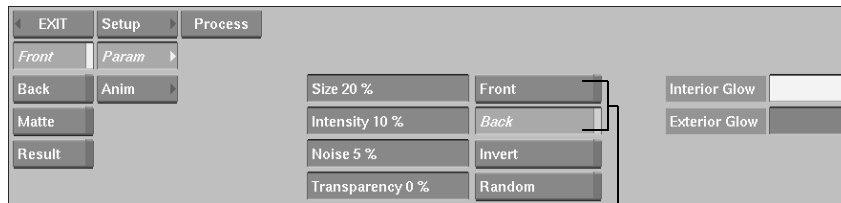
The Optics Module

Use the Optics module to add glow effects around front clips that are composited over back clips. The shape of the glow is determined by the matte you select when entering Optics.

To add a glow effect to a clip:

1. Click Optics in the Effects menu.
2. Select the front, back, and matte clips, and a destination.

The Optics menu appears.



Front and Back buttons

3. By default, Front view is enabled when you open the Optics menu to speed up processing. Click Result to view the glow effect.
4. You can use the Front and Back buttons to enable or disable the front and back clips. You can also invert the matte by enabling the Invert button.
5. If necessary, use the Size and Intensity fields to adjust the size and intensity of the glow effect.
6. Adjust the noise factor to make the glow jitter in the processed clip.

If you want to create a second processed clip where the glow jitters in a slightly different way from the first one, enable the Random button before processing the second clip.

7. If necessary, use the Transparency field to adjust the transparency of the front clip.

NOTE: The front clip must be enabled (see step 4, above).

8. Use the colour pots to change the colour of the glow. Click on a colour pot to display the colour picker. For information on using the colour picker, see “The Colour Picker” on page 57.

The interior and exterior glow colours are blended.

NOTE: To reset the glow, click the Anim button to open the Channel Editor, then click the Reset All button and Confirm.

9. Click Process. The generated clip appears on the destination reel.

Animating Glow Effects

The following Optics parameters can be animated:

- The size and intensity of the glow.
- The amount of glow noise.
- The transparency of the front clip.
- Red, green, and blue values for the glow’s interior and exterior colour.

To display the Animation menu, click the Anim button. For a description of the animation controls, see Chapter 9, “Animation.”

Setting Keyframes Automatically

A keyframe consists of a channel value set at a specific frame. A keyframe is saved automatically each time you change a channel value. You can turn off the automatic keyframe feature by disabling Auto Key in the Setup menu.

When Auto Key is disabled, you must explicitly set and remove keyframes using the Set Key and Rem Key buttons in the Anim menu. The procedures for using Set Key and Rem Key are described in “Setting and Removing Keyframes” on page 151.

Saving, Loading, and Removing Setups

Use the Save As and Load buttons in the Setup menu to save and load Optics setups. The general procedure for using setups is described in Chapter 7, “Saving Setups and Preferences.”

38 Tracking and Stabilizing

Keeping your images in line

The Stabilizer removes camera instability and motion jitter, and tracks reference points in your clips. You can also use the Stabilizer to produce 2D motion or lock a bilinear surface to the clip's background.

Summary

In this chapter, you learn about:

- “How the Stabilizer Works” on page 790
- “Working with Trackers” on page 793
- “Tracking” on page 798
- “Stabilizing” on page 806
- “Smoothing Out Motion” on page 811
- “Correcting Errors during Analysis” on page 812
- “Strategies for Tracking Difficult Shots” on page 813
- “Setup Options” on page 819
- “Importing and Exporting Data” on page 823
- “Copying Shift Channels” on page 823

About Tracking and Stabilizing

To track an image, you use the Stabilizer to make an object follow or track a reference point in another image. To stabilize a clip, track a point that you want to remain stable throughout the clip and the Stabilizer applies the opposite movements to the clip.

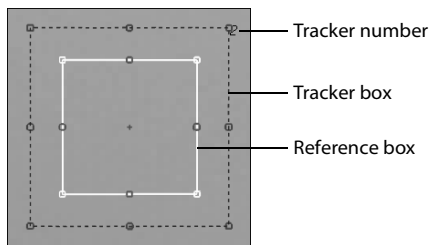
Tracking and stabilizing are often processes of trial and error. You are recommended to track or stabilize with default settings, and if the tracker box strays, fine-tune the analysis.

You can access the Stabilizer from the following modules.

Access the Stabilizer from:	To:
The desktop	Stabilize.
Action	Track or stabilize.
Compositor	Track or stabilize.
Keyer	Apply tracking data to a garbage mask.
Paint	Apply tracking data to an AutoPaint stroke.
Warper	Track mesh points or an axis.

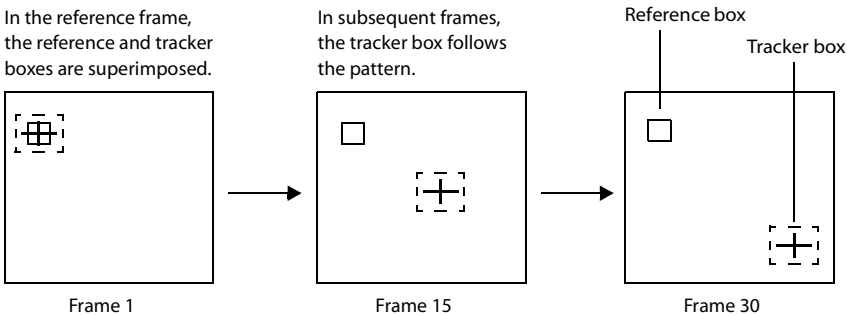
How the Stabilizer Works

The Stabilizer uses trackers to generate tracking data. Each tracker consists of a solid box, called the *reference box*, and a dashed box, called the *tracker box*. The reference box establishes the reference point (the feature to track or stabilize) in any frame of the clip. The tracker box follows the frame-to-frame movement of the reference point.

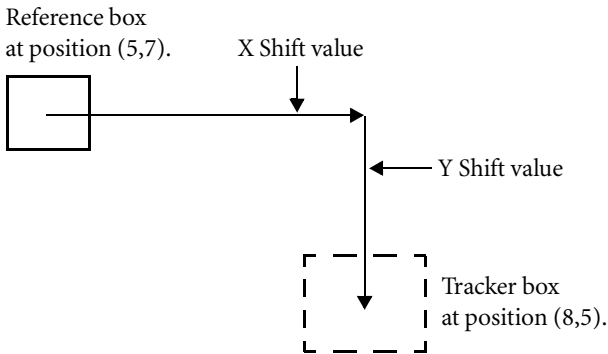


You start by selecting one or more reference points on your clip. Locate the first frame containing the movement to be tracked (the reference frame). In general, the reference frame is the first frame of the sequence. The choice of reference point depends on whether you are tracking or stabilizing. When tracking, the reference point is a feature you want to track; when stabilizing, the reference point represents the point around which the image is stabilized. See “Choosing a Reference Point” on page 793 for details. Place the reference box(es) around the selected feature(s).

Once you have set the tracker positions, start the tracking process, also referred to as analyzing the clip. During the analysis, the tracker box associated with each tracker moves as the Stabilizer looks for a pattern that matches the reference in each frame of the clip.



The Stabilizer calculates the difference between the position of the tracker box and the position of the reference box to produce X and Y Shift values. Shift values are a measurement in pixels and subpixels of how much the reference point has moved.



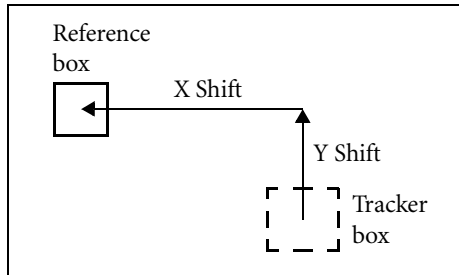
Tracker position in X	8	Tracker position in Y	5
-Reference position in X	5	-Reference position in Y	7
X Shift value	3	Y Shift value	-2

When the analysis is complete, you fine-tune it if a tracker box has strayed from the reference it was supposed to follow. Once you are satisfied with the results, you apply the data to the clip.

To track, the Stabilizer applies the Shift values “as is.” To stabilize, the Stabilizer inverts the X and Y Shift values in each frame of the sequence, and moves the image according to these values. This gives the impression that the reference point stays in the same position throughout the

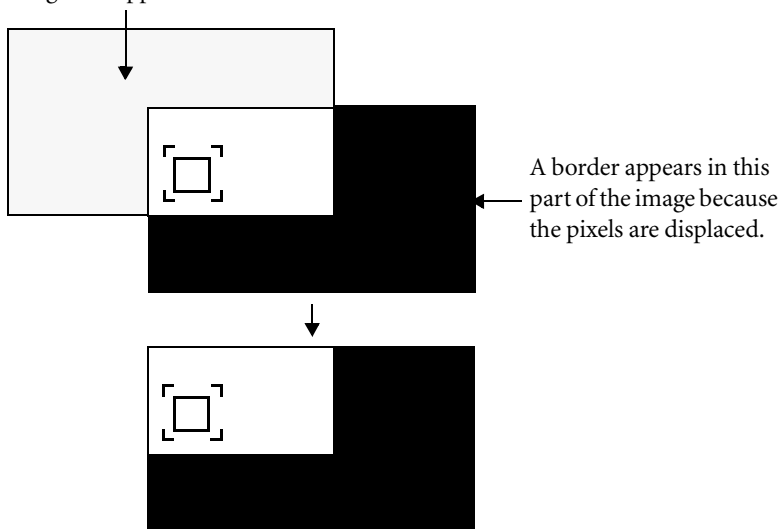
sequence. Because the image is moved during stabilization, a border appears on one or more edges, which means that you lose some pixels. The illustrations below summarize the process.

The Shift values are inverted.



The image is moved so that the contents of the tracker box are brought back to the position of the reference box.

By default, this part of the image is cropped out.



The image is offset in the direction of the inverse Shift values.

Working with Trackers

This section provides information that is common to many procedures. It is recommended that you first read the procedure you want to perform in “Tracking” on page 798 or “Stabilizing” on page 806, and then consult this section as needed.

Choosing a Reference Point

A good reference point is a high-contrast pattern that has good definition both vertically and horizontally, which allows for perfect registration in both directions. Preferably, the reference point should not change much over time.

To choose a good reference point, first play the clip several times to become familiar with the material. Ideally, you should try to find a pattern that is present in every frame. However, this is not always possible, and you may have to track two different features, track an object that disappears behind another one, or track an object that moves out of the frame. The section “Strategies for Tracking Difficult Shots” on page 813 explains how to work with these situations.

The reference frame (the frame on which you place the reference box) should be the frame where the reference point is most representative in terms of shape, size, and rotation. Generally, you position the reference box over the reference point in the first frame of the sequence, but you could also place it on the last frame and analyze backwards. Another alternative is to place it on a frame in the middle of the clip and analyze from that frame backwards to frame 1, and then analyze from that frame forwards to the end of the clip.

Selecting a Tracker

You can use any of the following methods to select a tracker:

- Select any tracker by selecting Move or Select in the Edit Mode box and clicking the tracker box, the tracking path, or the reference box of the desired tracker in the image window.
- Select trackers one through five by clicking the corresponding button (Tracker1, Tracker2, etc.).
- Select trackers one through nine by pressing the corresponding key.

NOTE: You cannot select trackers using the numeric keypad.

- Select the last tracker (regardless of the number of trackers present) by pressing 0 (zero).
- Select any tracker by pressing the **UP** and **DOWN ARROWS** or the **PAGE UP** and **PAGE DOWN** keys to scroll through the list.
- Select any tracker by selecting its channel in the Channel Editor.

When you select a reference box or a keyframe in the image window, the timeline automatically goes to the frame where the last reference point or keyframe was set.

Positioning the Reference Box

When you position the reference box, the tracker box automatically moves to the same location so that when you start analyzing, both boxes are positioned over the reference point.

To position the reference box:

1. Go to the frame where you want to position the reference box.
2. If the tracker you want to position does not appear on the image, click the appropriate Tracker button and enable the Active button.

Tracker buttons



Active button

3. **CTRL**-click anywhere on the image to select the reference box, and drag it over the pattern you want to track.

When you **CTRL**-click, the reference box is magnified to make it easier to select a position. By default, the tracker box stays with the reference box.

Resizing the Reference and Tracker Boxes

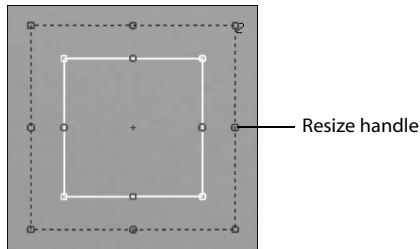
Resize the reference box to make it surround the feature you have chosen to track most closely. By only including the recognizable feature in the box with no other details that may change throughout the clip, you minimize the possibility of the tracker losing the reference point.

HINT: Sometimes a small reference point does not give good results. Try enlarging the box to include more of the selected feature. Note that the larger the box, the slower the analyzing speed.

You need to resize the tracker box so that it is large enough to accommodate the most frame-to-frame movement of the reference point. For example, if the movement of the reference point is mostly horizontal, you can increase the width and reduce the height of the tracker box. If there is a large amount of movement both horizontally and vertically, increase both the width and the height of the tracker box. Note that the smaller the tracker box, the faster the analysis.

You can resize the reference box and tracker box interactively on the frame:

- To resize either box, press one of the resize handles and drag. To maintain the proportions of the box, press **P** and drag a handle on one of the sides of the box.



- To move and resize the reference box and tracker box for the current tracker, use the dedicated menu fields.

Adjust the position and size of the reference box using these fields

Adjust the position and size of the tracker box using these fields



Resetting Trackers

You can reset the reference and tracker box size and position, the tracker Shift values, the entire tracker, or all trackers to their default position and size. You can use the buttons mentioned in this section, or open the Channel Editor and select the channel or channels you wish to reset, then click the Reset Chn button.



Reset Chn button

To reset the tracker box:

1. Click the tracker box to select it.
The tracker box turns white when selected.
2. Click R below the Tracker settings column.



Reset Tracker Box button

The tracker box size is reset and is repositioned over the reference box.

To reset the width, height, or position, open the Channel Editor, select the individual channel and click Reset Chn.

Resetting the Shift Values

To reset all keyframes, and retain the size of the selected reference point, tracker box, and reference box, select the tracker and click the Reset Shift Values button.



Reset Shift Values button

Resetting a Tracker

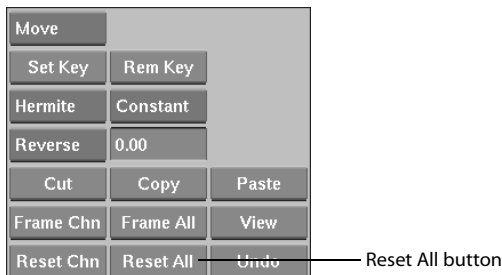
To reset the size and position of the reference and tracker boxes and the X and Y Shift values, select the tracker, click the Reset button and Confirm.



Reset button

Resetting All Trackers

To reset all trackers, click the Reset All button and Confirm.



Tracking

You can use the Stabilizer to make a clip or part of a clip track an object in another clip. For example, you can paste a logo to the side of a moving car and track the logo. You can track a clip using one, two, or four reference points:

- In one-point tracking, one reference point is tracked to produce 2D translation with no rotation or scaling.
- In two-point tracking, two reference points are tracked to generate translation, scaling and/or rotation data that is applied to the foreground element.
- In four-point tracking, the tracking data generated from four reference points is used to lock the four corners of a bilinear surface to the background in Action.

Since tracking involves compositing at least two clips, you open the Stabilizer from Action or from the Compositor. The procedures to track from the Compositor are similar to tracking from Action. The following sections explain how to track from Action.

One-Point and Two-Point Tracking

For one-point tracking, you use one tracker (Tracker1) to generate position information. For two-point tracking, you use a second tracker (Tracker2) to generate rotation and/or scaling information. The Stabilizer obtains this information by comparing the position of Tracker2 to that of Tracker1. Before you select the two reference points on the back clip, note the task of each tracker:

- Tracker1 follows the horizontal and vertical translation of the reference point. You should position Tracker1 over a point on the feature that you want to track.
- Tracker2 tracks the rotation and/or the change in size of the feature. In the first frame, the rotation is always 0 and the scaling factor is always 100%. In subsequent frames, a rotation and/or scaling factor is added if the relative position of the two trackers changes. You should position Tracker2 over a point that represents the rotation or change in size of the feature.

When loading the clips in Action, you load the clip that contains the feature you want to track as the back clip. In both one-point and two-point tracking, you specify the object to follow the feature on the back clip by selecting its axis before accessing the Stabilizer. This automatically assigns the tracking data to that object.

When you access the Stabilizer from Action for two-point tracking, you can select whether you want rotation information, scaling information, or both. Your selection depends on the movement of the object to track and on the camera movement in the clip.

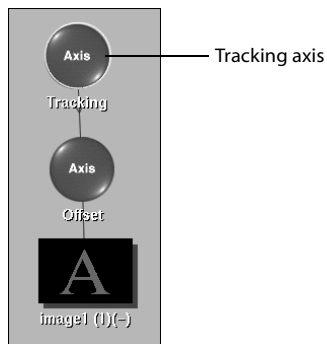
NOTE: To perform one-point or two-point tracking, you should be familiar with adding and parenting elements, and with using the Axis menu in Action (see “Working in the Scene” on page 939).

To perform one-point or two-point tracking using Action:

1. Load the clips in Action.

The front clip supplies the object and the back clip supplies the feature to track (the reference).

2. In Schematic view, add an axis to the front clip axis and surface branch. You can use this Offset axis to offset the object.



Making the Tracking axis the parent of the Offset axis allows you to reposition, scale, and rotate the object without modifying the tracking data.

3. Select the axis to use and click the Axis button to open the Axis menu.

In this example, the Tracking axis is used.

4. Select the Track option.

Process	Position	Rotation	Scaling	Shearing	Center
Layers ▶	X 0.00	X 0.00	X 100.00	X 0.00	X 0.00
Camera ▶	Y 0.00	Y 0.00	Y 100.00	Y 0.00	Y 0.00
Surface ▶	Z 0.00	Z 0.00	Z 100.00	Z 0.00	Z 0.00
Axis ▶	MotionPath		Prop Scale		
Light ▶	Global				
Text ▶	S ▶ Track	Rot Off	Scale Off		

Track option

5. Select the tracking options.

Select:

Rotation and Scaling Off

Rotation and/or Scaling On

Rot Inv

Scale Inv

To:

Do one-point tracking.

Do two-point tracking.

Invert the rotation data (for stabilizing).

Invert the scaling data (for stabilizing).

Process	Position	Rotation	Scaling	Shearing	Center
Layers ▶	X 0.00	X 0.00	X 100.00	X 0.00	X 0.00
Camera ▶	Y 0.00	Y 0.00	Y 100.00	Y 0.00	Y 0.00
Surface ▶	Z 0.00	Z 0.00	Z 100.00	Z 0.00	Z 0.00
Axis ▶	MotionPath		Prop Scale		
Light ▶	Global				
Text ▶	S ▶ Track	Rot Off	Scale Off		

Use these boxes to enable or invert rotation and scaling data

6. Access the Stabilizer by clicking the S button.

The Stabilizer opens and the back clip appears.

NOTE: If the wrong clip appears in the image window, exit the Stabilizer, exit Action, and reopen Action. Make sure you load the clip that contains the feature you want to track as the back clip.

7. Position the tracker(s) over the feature(s) to track.
8. Click Analyze to generate the tracking data. If either tracker strays from the reference point, see “Correcting Errors during Analysis” on page 812.

9. Fine-tune the tracking data if necessary. For more information, see “Strategies for Tracking Difficult Shots” on page 813.
10. When you are satisfied with the tracking, make sure the trackers you used are still active by clicking each tracker button and checking that the Active button is enabled.
11. Click Save.

The Axis menu in Action reappears. The tracking data is applied to the front clip.

12. Place the object in the desired position by selecting the Offset axis, opening the Axis menu, and entering the required values in the X, Y and Z position fields. You can also reposition the object by dragging its axis in the image window.

Four-Point Tracking

With four-point tracking, you use four trackers in the Stabilizer to generate tracking data for anchoring the four corners of a bilinear surface to the background clip.

The reference points you use must be well-defined; it is recommended that you plan them when shooting the sequence (for example, add markers to the scene). It is not always possible to do four-point tracking when the reference points are not well-defined.

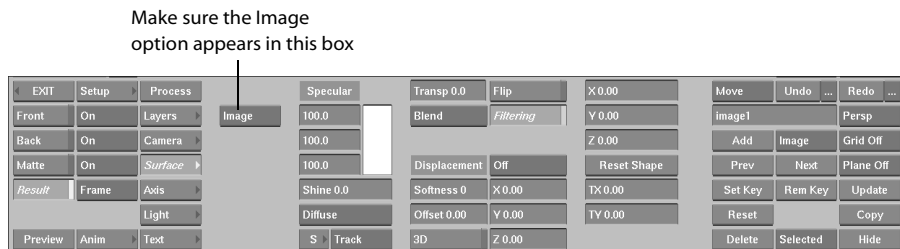
When loading the clips in Action, you load the clip that contains the feature you want to track as the back clip. Because four-point tracking is applied to bilinear surfaces, you select the bilinear surface before accessing the Stabilizer. The tracking data is automatically assigned to that surface. Note that this differs from one- and two-point tracking, where you select the axis of the surface before accessing the Stabilizer, rather than the surface itself.

NOTE: To perform four-point tracking, you should be familiar with adding and parenting elements, and with the Surface menu in Action. For more information on parenting elements, see “Working with Branches in the Schematic” on page 963. For more information on surfaces, see “Working with Surfaces” on page 921.

To perform four-point tracking:

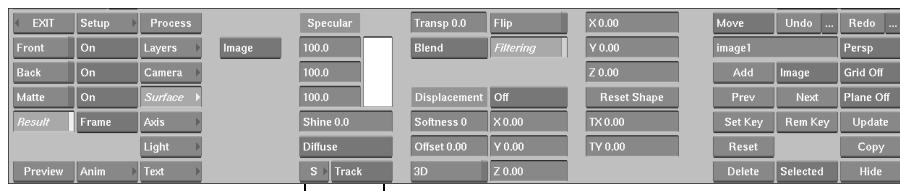
1. Load the clips in Action. The front clip supplies the surface and the back clip supplies the feature to track.
2. Select the front clip layer and use the Add command to add a bilinear surface. Delete the default axis and image surface branch.

3. Select the bilinear surface and open the Surface menu by clicking the Surface button.



NOTE: Do not move, scale, or rotate the surface axis before accessing the Stabilizer; otherwise, the tracking data will not be correct.

4. Select the Track option and then click the S button.

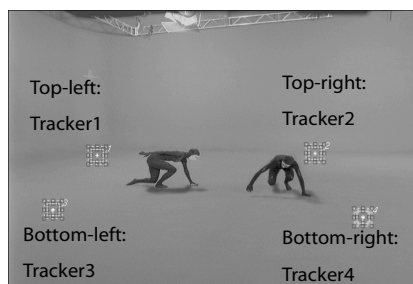


Select the Track option and click the S button.

The Stabilizer opens and the back clip appears.

NOTE: If the wrong clip appears in the image window, exit the Stabilizer, exit Action, and reopen Action. Make sure you load the clip that contains the four anchor points you want to track as the back clip.

5. Position the four trackers on the background clip using the following layout.



6. Once you set the size and position of the trackers, click Analyze to generate the tracking data.
7. After the analysis is complete, make sure the trackers are still active and click Save.

The Surface menu in Action reappears. The tracking data is automatically applied to the four corners of the bilinear surface.

- Place the object in the desired position by selecting the axis of the bilinear surface, opening the Axis menu, and entering the required values in the X, Y and Z position fields.

Using Offsets

Instead of anchoring the entire bilinear surface to the back clip, you may prefer to use a smaller region of the image. You can move the four-corner anchor points anywhere inside or outside of the front image to define the area of the front image that you want to use, and you can animate the resulting offsets.

Setting the Offsets Manually — To change the offsets manually, go to the Surface menu in Action, display the Image submenu and select the bilinear surface that you want to use as the front image. Click the Edit Offsets button to display the anchor points on the bilinear surface. The anchor points appear at the four corners of the surface. You can move the anchor points by dragging them directly on the image.

To see the result, disable the Edit Offsets button.

Generating the Offsets Using the Stabilizer — If you want the anchor point offsets to change from frame to frame in order to track a feature on a bilinear surface, you can use the Stabilizer to generate the offsets automatically. Note that the image can be deformed when you use this method.

To generate the offsets using the Stabilizer:

- Track the four reference points on the back clip as explained in “To perform four-point tracking:” on page 801.
- Make sure the bilinear surface you used in the previous procedure is still selected.
- Display the Image submenu.
- Select the Offsets option and then click the S button.



Select the Offsets option
and click the S button.

When you select the Offsets option, the front clip of the selected bilinear surface appears in the Stabilizer instead of the back clip.

5. Position the four trackers on the front clip using the same layout you used when tracking the back clip.
6. Click Analyze to generate the tracking data.
7. After the analysis is complete, make sure that the trackers are still active, then click Save. The Surface menu in Action reappears. The tracking data is automatically applied to the offset channel of the front clip.

Animating the Offsets — The animation channels for the anchor point offsets are found in the upper_left, lower_left, upper_right and lower_right folders of the Offsets folder of the bilinear surface (in the Action Animation menu). Each X and Y value is expressed as a percentage relative to its corresponding corner. Each anchor point offset starts with a default value of zero. For example, changing the top-left X offset value to 20 moves the top-left anchor point inside the bilinear surface by 20%. This has the effect of stretching the image outside of the bilinear shape.

NOTE: You can use the Copy and Paste buttons in the Channel Editor to copy one anchor point offset setup to another. This is useful, for example, if you want to maintain the proportions of the image.

Working with a Large Number of Trackers

Use the following tips when working with a large number of trackers.

Speeding Up the Analysis

To speed up the analysis, disable the Icons and Motion Path buttons in the Setup menu before starting. This turns off the display of the tracker and reference boxes, as well as the tracking path. Re-enable the buttons to fine-tune the analysis.

Affecting Several Trackers at Once

By using the options in the Tracker Selection box, you can change the parameters of a single tracker or all active trackers, and choose whether you can see only the selected tracker or all

trackers. For example, you can change the tracker box dimensions for all trackers, or edit the tracking path of a single tracker while hiding the other trackers.



Select: **To:**

Selected Affect only the selected tracker and see all other trackers in the image window.

Solo Affect only the selected tracker and hide all other trackers.

Gang Affect all active trackers (except when changing their colour or resetting, where all trackers are affected whether or not they are active).

Locating a Tracker or a Keyframe

To find the number of a tracker, click any part of its tracking path in the image window. To find the frame number associated with a particular keyframe, click the keyframe. The timeline moves to the corresponding frame.

NOTE: Use Select mode instead of Move mode to avoid moving a keyframe accidentally.

When you select a tracking path,
the tracker number appears here.



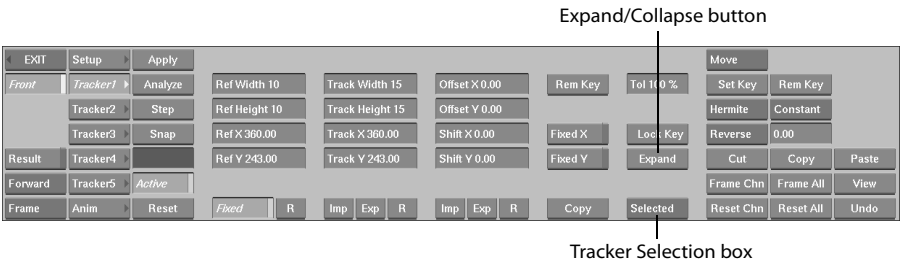
Locating a Shift Channel

Use the Expand/Collapse button to locate a Shift channel quickly. This button is available when the Anim button is disabled.

To locate a Shift channel:

1. Collapse all channels. Select Gang in the Tracker Selection box to change a parameter for all active trackers at once.
2. Click the Collapse button.

NOTE: If the button reads Expand, click it to toggle back to Collapse.



3. In the image window, click the tracking path of the tracker containing the Shift channel you want to locate.
4. Click the Expand button.
5. Click the Anim button to display the Channel Editor.
The shift curve of the tracker appears in the Channel Editor.
6. Scroll up or down to display the Shift folder.

Stabilizing

There are several methods for stabilizing a clip.

Use:	To stabilize a clip where:
Plain stabilization	There is no camera pan. Plain stabilization removes jitter, but if there is a camera pan, it will be removed along with the jitter.
Jitter (in Channel Editor)	There is a pan or tilt. The Jitter option removes jitter and keeps the pan by averaging the pan over time and subtracting it from original Shift values.
Fixed X and Fixed Y	There is a strictly vertical tilt or strictly horizontal pan. Fixed X and Fixed Y remove motion in one direction only. Use Fixed X and Fixed Y only when you do not wish to stabilize one of the channels.

After stabilizing a clip, a border appears at its edges. The Stabilizer provides a number of different methods to removes the border.

Stabilizing from the Desktop and from Action

There are different options available when you access the Stabilizer from the desktop and from Action. When you access the Stabilizer from the desktop, you stabilize with one tracker only, but when you access it from Action, you have the option to use two trackers. You need two trackers when the clip you want to stabilize has a camera roll—the second tracker enables you to track the rotation of the camera.

When stabilizing from the desktop, you have rendering options to remove the border that results from stabilizing. When stabilizing from Action, you must remove the border manually. In addition, when stabilizing from the desktop, you can use the Oversampling option to obtain high quality rendering. For more information, see “Rendering Options” on page 822.

You should stabilize from Action to remove jitter when there is a camera roll that you want to remove, and from the desktop in all other cases.

Stabilizing with Plain Stabilization

Use plain stabilization to stabilize a clip with no pan or tilt. The procedure for stabilizing is very similar to the procedure for one-point and two-point tracking.

To stabilize a clip with no camera roll:

1. Click the Stabilizer button in the Effects menu.
2. Select the source clip and the destination reel.

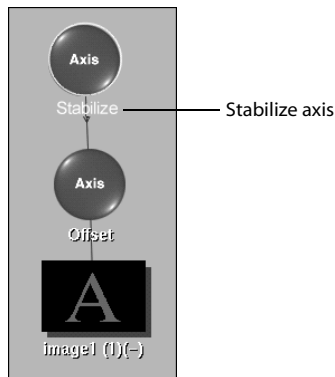
The Stabilizer menu appears.



3. Position Tracker1 over a feature to stabilize.
4. To have the border at the edges of the clip automatically removed, select the Fit All option in the Setup menu. See “Scaling and Shifting Options” on page 822.
5. Click Analyze to generate the stabilization data.
6. Fine-tune the data if necessary. For more information, see “Correcting Errors during Analysis” on page 812.
7. When you are satisfied with the stabilization, go to frame 1 and click Apply.

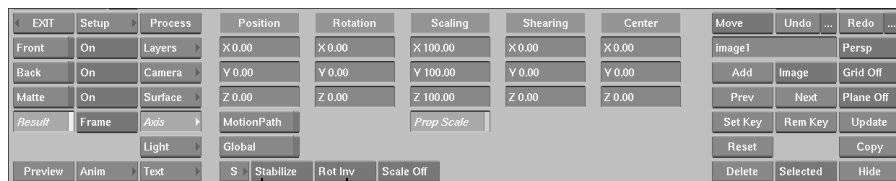
To stabilize a clip with camera roll:

1. In Action, load the clip to stabilize as front, back, and matte.
2. Set the matte to Off.
3. In Schematic view, add an axis to the front clip axis and surface branch. You can use this Offset axis to offset the object.



Making the Stabilize axis the parent of the Offset axis allows you to remove the border that results from stabilization without affecting the stabilization data.

4. Select the Stabilize axis and open the Axis menu.
5. Select the Stabilize option.



Stabilize option Rotation Invert option

6. Select Rotation Invert.

NOTE: You can also stabilize scaling by using the Scale Invert option.

7. Access the Stabilizer by clicking the S button.
8. Position Tracker1 over a feature to stabilize.
9. Position Tracker2 so that the two trackers are as far as possible from each other. This allows the Stabilizer to calculate the change in relative position with greater accuracy.

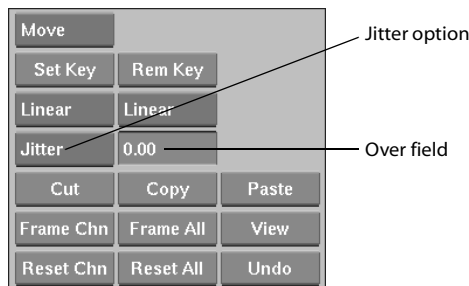
NOTE: If there is a zoom in the scene, select two points that are the same Z-depth throughout the shot. For example, if the camera zooms toward a building, you select two points on the building, as each point remains at the same distance from the camera.

10. Click Analyze to generate the stabilization data.
11. Fine-tune the data if necessary. For more information, see “Correcting Errors during Analysis” on page 812.
12. When you are satisfied with the stabilization, go to frame 1 and click Save. You are returned to the Axis menu in Action.
13. Remove the border that results from stabilization by selecting the Offset axis, and entering a scaling factor in the Scale fields and/or position information in the Pos fields.

Removing Jitter while Keeping Overall Motion

Use the Jitter option to remove the jitter from a clip and keep the overall motion. For example, if a shot is filmed from the back of a moving truck, use the Jitter option to remove the jitter and retain the rest of the movement.

To use this option, analyze the clip then apply the Jitter option to the shift curve.



At each frame, the Stabilizer compares the position of the reference point with its position on a range of frames before and after the current frame. It calculates the average motion over this range and subtracts it from the original tracking data. Applying this calculation leaves the jitter values only. You specify the range in number of frames in the Over field.

Removing jitter is often a process of trial and error, and you can try different Over values until you are satisfied with the result. As a general rule, start with a large Over value to remove slow jitter and a small Over value to remove fast jitter.

To remove jitter and keep overall motion:

1. Click the Stabilizer button in the Effects menu.
2. Select the source clip and the destination reel.

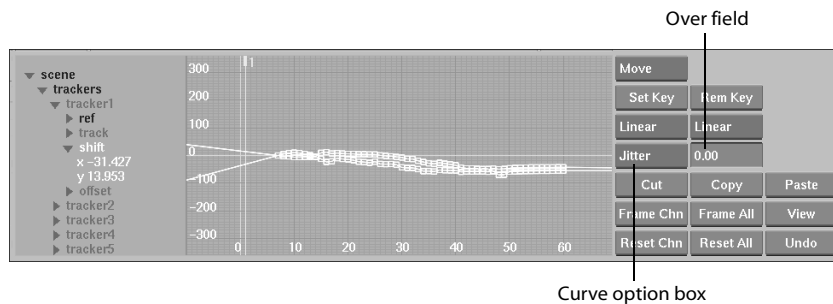
The Stabilizer menu appears.

3. Position Tracker1 over a feature to stabilize.
4. If you want to automatically remove the border at the edges of the clip, select the Fit All option in the Setup menu. See “Scaling and Shifting Options” on page 822.
5. Go to the reference frame and click Analyze to generate the shift data.
6. Click the Anim button to display the Channel Editor.
7. Expand the Shift folder for the tracker you are using and select the Shift folder.



The the x and y channel curves turn white, indicating that they are selected.

8. Click the Copy button.
9. Enter a value in the Over field.



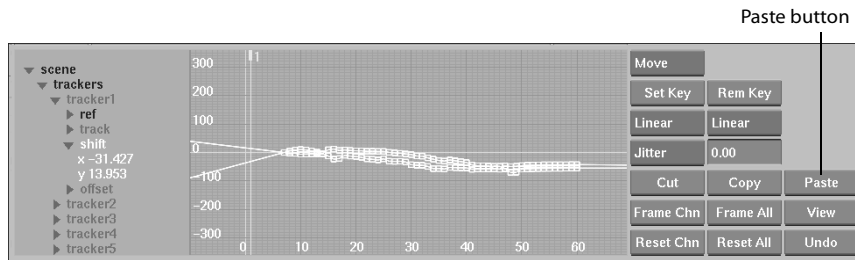
10. Select the Jitter option in the Curve option box.

The transformation is applied to curves as soon as you select Jitter option. The curves show the amount of motion the Stabilizer removes. The curves should oscillate around the zero point, but should not be entirely flat. A flat curve would indicate that Stabilizer removed no jitter.

11. Click Result and play the clip to determine whether it is stable enough.

HINT: You may want to process a few frames to see the result in the correct frame speed.

12. If the clip is stable enough, click Apply. If it is not, click Paste to revert to the original shift data.



13. Repeat the procedure from step 9 with different Over values until you are satisfied with the results.

Removing Jitter with Fixed X and Fixed Y

The Fixed X and Fixed Y buttons remove motion in one direction only. Enable the Fixed X button to stabilize vertically only. Movement on the horizontal (X) axis, such as a camera pan, is not affected. Enable the Fixed Y button to stabilize horizontally. Movement on the vertical (Y) axis is not affected.

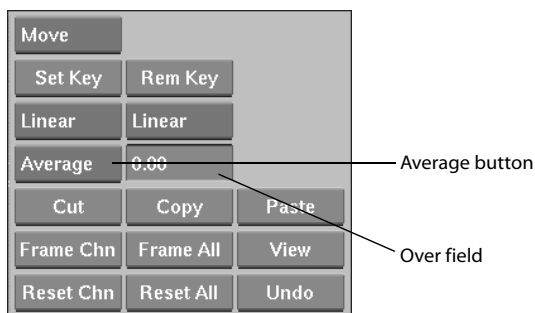
Smoothing Out Motion

Use the Average button to smooth out uneven camera motion. For example, if the camera moves at a certain rate and suddenly drops or increases in speed, the Average option can stabilize the camera motion.

NOTE: Use this feature when tracking a difficult shot where the tracker box oscillates around the reference point.

The Average option smooths camera motion over a group of keyframes. Use the Over field to determine how much stabilizing is applied to the clip. A larger value averages camera motion over more keyframes and increases camera smoothness. Use Average to affect the tracker's Shift values since Shift represents the amount of movement in a clip.

Analyze your clip to generate the initial channel data, then select the channel to apply average to.



To smooth out camera motion using Average:

1. Position the tracker(s) on your image and click Analyze.
2. Click Animation to open the Channel Editor and expand the Shift folder for the track(s).
3. Select the X or Y Shift folder and enter an average value in the Over field.
4. Click the Average button.

Correcting Errors during Analysis

If the tracker box strays from the reference point that it is supposed to be tracking, incorrect shift keyframes result. If such an error occurs, you can stop the analysis, correct it, and restart it at any frame.

There are a number of different methods you can try to correct tracking errors. Try the strategies given below, then redo the analysis. Click the Analyze button to generate new keyframes based on the updated information you provide.

Updating the Reference Point Using Snap

If the tracker loses the reference point, you can delete the incorrect keyframes, then, at the last good keyframe, update the reference point to the current contents of the tracker box.

To update the reference point to the tracker box contents:

1. Select the appropriate tracker by clicking on the tracker button.
2. Press **BACKSPACE**.
The last keyframe is deleted and the play bar moves back one frame. The tracker box is now on the previous keyframe.
3. If you want to remove this keyframe, press **BACKSPACE** again.
4. Repeat step 2 for all incorrect keyframes.
5. When you have reached the last good keyframe, click the Snap button. This updates the reference point from the one originally specified in the reference frame to the image currently inside the tracker box.
6. Click Analyze to continue the tracking process.

Resetting the Tracker

If the tracker loses the reference point early in the analysis, you may want to reset the entire tracker and find a new reference point. See “Resetting a Tracker” on page 797 for instructions.

Adjusting the Size of the Tracker Box

If the Stabilizer cannot find the reference point within the boundaries of the tracker box during analysis, the tracker box strays from the reference point and produces incorrect keyframes. Although you can manually correct these keyframes, it is easier to make the tracker box large enough to accommodate the movement of the reference point. Note that processing time increases as the size of the tracker box increases. For more information, see “Resizing the Reference and Tracker Boxes” on page 794.

To adjust the size of the tracker box:

1. Stop the analysis by pressing on the stylus or holding down the left mouse button.

NOTE: When stopping an analysis, make sure to press below the timeline to avoid moving the tracker box.

2. Press **BACKSPACE** until you reach the last good frame before the tracker box strayed from the reference point. Adjust the size of the tracker box so that it is large enough to accommodate the largest frame-to-frame movement of the reference point.

NOTE: Backspacing deletes the keyframes as you go back. If you do not delete the keyframes, they will be overwritten when you re-analyze. Backspacing in this instance makes it easier to see the tracking path clearly.

3. Click Analyze to continue the analysis.

Strategies for Tracking Difficult Shots

This section provides different strategies that you can use to track difficult shots.

Positioning the Tracker Box Manually

When the reference point is temporarily covered by another object, position the tracker manually.

When you manually position the tracker box, the new position is recorded in the Track X and Track Y channels of the Channel Editor and the Shift X and Shift Y values are calculated accordingly.

To manually position the tracker box:

1. Stop the analysis by pressing on the stylus or holding down the left mouse button anywhere below the timeline.
2. Advance the clip to the frame where the reference point becomes visible again, and reposition the tracker box over the reference point. Click Analyze to restart the analysis at this frame.

Since the X and Y shifts are recorded as keyframes in channels, the Stabilizer calculates the translation values for the frames in which the reference point was covered. The final result will be a smooth motion.

Locking Keyframes in Place

You can lock Shift keyframes so that they stay in place even if you try to move them manually or perform another analysis. This is useful, for example, when you have set keyframes manually and want to prevent the Stabilizer from overwriting them when redoing the analysis.

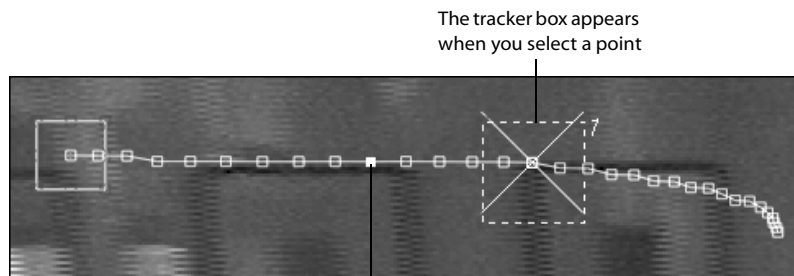
In the following procedure, you select the keyframes directly in the image window. You can also select the Shift keyframes in the Channel Editor.

NOTE: You cannot lock several keyframes at once. You must lock each keyframe individually.

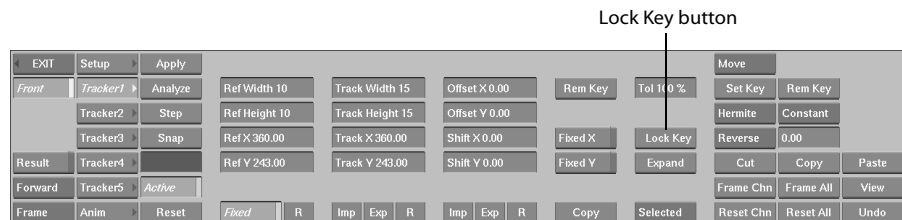
To lock keyframes in place:

1. If the tracking path does not appear in the image window, enable the button in the Setup menu.
2. Zoom in on the image until you can see the points clearly and select the point you want to lock on the tracking path.

The tracking path turns white when you click it, and the frame that corresponds to the keyframe appears in the image window.



3. Click the Lock Key button.



NOTE: To unlock a keyframe, select it and click the Unlock Key button.

Changing the Reference Point

Change the reference point to another feature if the feature you have been tracking moves out of the frame, or alters such that it is no longer trackable. You should do this before the original reference point moves out of the frame. This allows the Stabilizer to calculate the offset between the two reference points and predict the position of the original reference point.

Ideally, before analyzing the clip, play it several times to determine where you need to change the reference point, and then set the new position of the reference point at the appropriate frame.

When you select the reference box in the image window, the play bar automatically goes to the frame where you last set the reference point. For example, if you set the reference in frame 10, then go to frame 20 and select the reference box, you are automatically brought back to frame 10. Use Add mode to change the reference point at a particular frame.

To change the reference point before analyzing:

1. In the Setup menu, enable the Motion Path button so that you can see the interpolated path of the original reference point.
2. Select the tracker for which you want to change the reference point.
3. Go to the desired frame.
4. Select Add in the Edit Mode box.
5. Press **CTRL** as you drag the reference box to the new reference point. Notice that the tracker box moves in relation to the reference box.
6. Go to the first frame of the sequence and click Analyze.

NOTE: You can also change the reference point during analysis. Stop the analysis at the appropriate frame, and use Add mode to move the reference box to a new position.

Finding the Best Tolerance Value

The Tolerance value determines how exact a match the Stabilizer requires when identifying the reference point. With a tolerance of 0%, the feature being tracked must be exactly the same as the reference point. With a tolerance of 100%, the Stabilizer accepts anything as a match. If a match is not found, a keyframe is not set, and the position of the reference point is interpolated between the previous keyframe and the next keyframe.

If the reference point is temporarily hidden by another object, you can use the Tolerance value to make the tracker ignore the reference point in parts of the clip where it is hidden, then continue tracking it normally when it reappears. This can prevent or reduce the need to manually reposition the tracker box during an analysis.

To use tolerance to track a temporarily hidden object:

1. In the Setup menu, enable the Motion Path button so that you can see the tracking path.
2. Analyze the clip with full (100%) tolerance.
3. The tracker loses the reference point when it is hidden. Press **BACKSPACE** to delete keyframes and go back to the last bad keyframe. Lower the Tolerance value until the crosshair disappears from the tracker box.

The keyframe at that frame has been deleted. The crosshairs indicate that a frame has a keyframe.

4. Click Analyze to continue the analysis.

Further analysis automatically discards the undesirable keyframes, and the position of the reference point is interpolated until the reference point reappears.

HINT: Raise the tolerance to make a desirable keyframe that has been filtered out reappear. Make sure you are at that frame to accomplish this.

Analyzing Backward

Analyze the clip backward when the feature you want to track grows larger or when it is off screen at the beginning of the clip.

NOTE: You cannot analyze backward if you need to change or snap the reference point. In this case, reverse the clip before entering the Stabilizer and analyze it forward, then reverse the tracker data in the Channel Editor and apply it to the original clip.

To analyze backward:

1. Select the Backward option in the Direction box.

Direction box



2. Go to the last frame of the sequence.
3. Position the trackers.
4. Click Analyze.

Tracking Manually

Track manually when the feature you are tracking disappears behind an object for several frames, moves out of the frame, or is extremely difficult to track.

Using a Fixed Reference Point for Manual Tracking

When tracking manually, it is easier to use a non-fixed reference point. However, the results may not be as accurate because any small errors you make when positioning the tracker box are compounded when Fixed is off. Use a fixed reference point wherever possible.

To track a feature that disappears:

1. Go to the last keyframe before the reference point disappears.

NOTE: Make sure to delete any bad keyframes.

2. Redefine the reference point to the current tracker box contents by clicking the Snap button.
3. Select Add in the Edit Mode box.
4. Press **SHIFT**, click and drag the tracker box. The timeline advances one frame forward or backward (the direction depends on the setting in the Direction box).

Using the reference image you see in transparency as a guide to find the new position, drag the tracker box to the desired position. A keyframe is set when you release the cursor.

NOTE: If you do not see the reference image in transparency, set the Opacity option in the Setup menu to a value around 50%.

5. Repeat step 4 as necessary.

HINT: Lock the keyframes as you add them.

Tracking an Erratic Feature Manually

With this method, you create part of the tracking path manually. You can either add a keyframe in each frame, or add keyframes in significant frames of the clip and interpolate the curve for the other frames.

You must delete unwanted keyframes before adding new ones.

To track an erratic feature manually:

1. Remove any bad keyframes by selecting Delete in the Edit Mode box and clicking the bad keyframes in the image window. Alternatively, press the **BACKSPACE** key.
2. Go back to the last good frame before the frame where you want to start tracking manually.
3. Click the Snap button to redefine the reference point on this frame.

NOTE: If you are tracking a whole clip manually, position the reference box on the feature to track.

4. Select Add in the Edit Mode box.

5. Do one of the following:

- To set critical keyframes only, select Hermite in the Interpolation Mode box.
- To set keyframes on every frame, select Linear in the Interpolation Mode box.

Interpolation Mode box

6. Add a keyframe by holding **SHIFT** and dragging the tracker box over the reference point.

You should see the reference image in transparency and the next frame in the background. The keyframe is added in the next frame when you release the mouse button or lift the stylus.

NOTE: If you do not see the reference image in transparency, make sure you have defined a reference and check that the Opacity is set to approximately 50% in the Setup menu.

7. Repeat step 6 for every frame where you want to define a keyframe.

8. Play the clip and adjust the tracking path by moving or adding keyframes.

Analyzing One Frame at a Time

This is useful for difficult shots because you can adjust the position of the tracker after each frame. Click the Step button to analyze a single frame and advance to the next frame.

Editing the Tracker Channels on the Image

You can edit the tracker channels directly on the image instead of editing them in the Channel Editor. Note that the curve on the image shows the X and Y components together and not separately as in the Channel Editor. The tracking path on the image shows the actual position of the tracker box.

To edit the curve directly in the image window, Motion Path must be enabled in the Setup menu. You can use all the options in the Edit Mode box except Rect. Zoom, and Pan. For more information on the options in the Edit Mode box, see “Editing Animation Curves and Keyframes” on page 134.

When you select a keyframe on the curve, the timeline goes to the frame that corresponds to the keyframe. When you select the reference box, the timeline goes to the frame where you set the reference.

Tracking a Colour-Corrected Copy of the Clip

As a last resort, you can try colour correcting a copy of the image until the contrast of the feature to track is high enough. You then generate tracking data from this copy, save the setup, and apply it to the original clip. Try the following methods to modify the clip:

- Increase the contrast in the Colour Corrector.
- Apply filters (for example, emboss).
- Use Degrain to remove film grain.

Setup Options

When tracking and stabilizing, you can set various options for:

- The way tracking is performed
- The interface
- Rendering

NOTE: The rendering options are only available when you access the Stabilizer from the desktop.

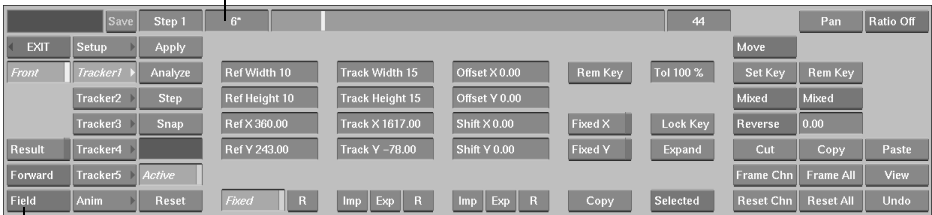
Tracking Options

Tracking options define the way in which the Stabilizer sets keyframes or tracks the reference point.

Tracking on a Field or Frame Basis

By default, the Stabilizer works in Frame mode. Select Field mode when working with interlaced images, or with images that display a lot of field jitter. In Field mode, the Stabilizer sets two keyframes for every frame: one for the even field and one for the odd field.

In Field mode, a star appears next to the second field in time



Field/Frame box

Fixed Reference Point

Using Fixed Reference button, you can analyze with either a fixed or a non-fixed reference point.



Fixed Reference button

When Fixed is on, the Stabilizer searches for the reference point specified at the reference frame (where you last set the reference point). When Fixed is off, the reference point is updated at each frame as the analysis takes place. In each frame of the analysis, the Stabilizer looks for the reference point from the previous frame.

It may be helpful to turn Fixed off if you are tracking a feature that changes considerably over time. For example, the feature may be rotating or may change size or shape.

HINT: Tracking a feature that changes over time is easier when the Fixed option is off, but the results are often not as accurate. This is because any small errors in the analysis are compounded when Fixed is off. Leave Fixed on whenever possible.

NOTE: Unless you are in Gang mode, you must set Fixed button for each active tracker individually.

User Interface Options

User interface options let you control the appearance of the tracker, the display of the tracking path and the opacity of the reference image when tracking manually.

Changing the Colour of Trackers

You can change the colour of a single tracker using the Selected option in the Tracker Selection box, or of all trackers at once using the Gang option.

To change the colour of the tracker:

1. Click the Tracker button that corresponds to the tracker you want to change.
2. Click the Tracker colour pot.

Tracker colour pot

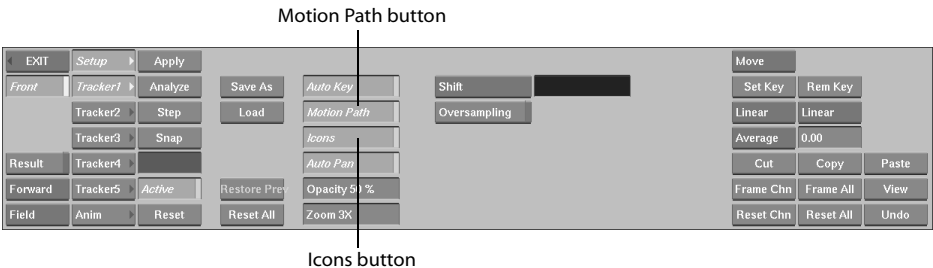


The colour picker appears.

- 3. Select a colour with the colour picker (see “The Colour Picker” on page 57).
- 4. Click in the Tracker colour pot to apply the new colour to the tracker.

Showing the Tracking Path

By default, the tracking path appears in the image window. You can turn it off by disabling the Motion Path button in the Setup menu. The tracking path is the path that the reference point makes as it changes position from frame to frame.



Hiding the Reference and Tracker Boxes

By default, the tracker and reference boxes appear in the image window when a tracker is active. You can hide them by disabling the Icons button in the Setup menu. This is useful when working with many trackers.

Automatically Zooming In on the Selected Tracker

When you zoom in, the tracker you are working with may no longer be visible, and you have to pan to see it. Enable the Auto Pan option in the Setup menu to make the zoom command automatically zoom in on the selected tracker. As the stabilizer analyzes, if the tracker moves to the edge of the viewer, the image is panned to keep the tracker visible.

This feature keeps the tracker centered not only in the case of a zoom, and is useful for film resolution because the image is larger than monitor.

Opacity

By default, the reference image (the image where you placed the reference box) appears in transparency when you select a keyframe. Adjust the opacity of the image to make it more or less transparent. When the opacity is set to 0, the reference image does not appear; when it is set to 100, the reference image is completely opaque.

Magnifying Glass

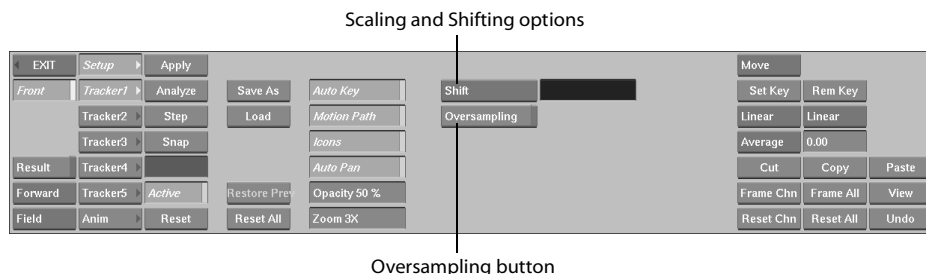
By default, the reference box turns into a magnifying glass when you select it. Increase or reduce the magnification factor in the Zoom field.

Select a zoom factor of: **To get:**

- | | |
|------------|---|
| 0 | No zoom, but a crosshair in the tracker box after analysis. |
| 1 | No zoom and no crosshair. |
| 2, 3, 4, 5 | A magnification factor of 2, 3, 4, and 5, and a crosshair in the tracker box. |

Rendering Options

The following options are available only when you access the Stabilizer directly from the desktop.



Scaling and Shifting Options

When a clip is stabilized, the image is shifted. You can choose any of the following options to fill or remove the area where the image has been shifted.

- | | |
|----------------|---|
| Select: | To: |
| Shift | Use a colour to fill the area. Use the adjacent colour pot to pick the colour. |
| Roll | Wrap the image around to fill the area. |
| Letterbox | Rescale the longest edge of the image to fit into the frame and fill the rest of the image with a coloured border. |
| Crop Edges | Rescale the shortest edge of the image to fit into the frame and crop the longest edge. |
| Fill | Rescale both dimensions of the image independently to make it fit into the frame. Note that this can change the aspect ratio. |

Oversampling

Use Oversampling to toggle high-quality subpixel image adjustment on and off.

Importing and Exporting Data

Use the Import and Export buttons below the Track fields to import and export tracking data values to an ASCII file. The tracking data specifies the X and Y position of the tracker box in relation to the origin point of the image; these are absolute coordinate values. The origin point has coordinates (0,0), and is located at the lower-left corner of the image.

NOTE: If the position of the reference box is changed during the analysis, the offset is compensated for and the exported tracking data still shows a continuous tracker path.



Import and Export buttons

Use the Import and Export buttons below the Shift fields to import and export Shift data values in an ASCII file. The shift data specifies the difference between the position of the reference box (0,0) and the position of the tracker box in the current frame; these are relative coordinate values.

Both the tracking values and the Shift values are formatted as follows:

frame#: X position, Y position

Copying Shift Channels

Using the Channel Editor, you can copy a tracker's Shift channel values and paste them into any other channel in any module. For example, you can copy jitter values to the X and Y position channels of an axis to add realism to a static scene, or to a channel to add noise.

There are two Copy buttons in the Stabilizer. One takes pixel aspect ratio of the clip into account and the other one does not.



This Copy button takes the aspect ratio into account

This Copy button does not take the aspect ratio into account

The coordinate system that the Stabilizer uses for mapping pixels on the image differs from the one used by other modules. In the Stabilizer, the pixel with coordinates (0,0) is in the lower-left corner, while in most other modules it is in the centre of the screen. When pasting shift curves to the channels of an axis (for example, a garbage mask axis or surface axis), use the Copy button that takes the aspect ratio into account. Since this button also accounts for the differences in pixel coordinates between modules, it ensures that the values are mapped properly.

To copy and paste a shift channel:

1. In the Channel Editor, open the folder of the tracker whose Shift values you want to copy.
2. Select the X or Y shift channel, or the entire Shift folder to select both X and Y shift channels.
3. Click one of the Copy buttons, depending on whether or not you want to maintain the aspect ratio.
4. If necessary, load the clip to which you want to apply the Shift values into the appropriate module.
5. Open the folder of the channel to receive the copied Shift values and select the appropriate channel(s).
6. Click Paste.

The Warper is a powerful tool for warping and morphing source clips. Warping is free-form distortion of an image—elongating someone’s nose. Morphing is a transition effect that matches the morphology of one clip gradually to that of another clip—a human morphing into an alien.

Summary

In this chapter, you learn about:

- “Loading the Clips” on page 825
- “Setting Up the Clips” on page 827
- “Creating Meshes” on page 828
- “Modifying a Mesh” on page 829
- “Modifying the Meshes for Live-Action Clips” on page 836
- “Warping” on page 840
- “Morphing” on page 842
- “Setup Options” on page 848

Loading the Clips

You must select the required clips upon opening the Warper. You can enter the Warper with one to five clips. The number of clips that you load depends on the effect that you want to produce.

To open the Warper:

1. In the **flame** menu, click Warper.
2. Select the types of clips you want to load into the Warper from the Input Mode box.

Select:	To load:	To produce:
Front	Front clip only	A warp of a single image.
Front Back	Front and back clips	A morph, a wipe, or a regional warp.
Front Matte	Front and matte clips	A warp of a single image using a matte.
Front Back Matte	Front, back, and matte clips	A warp over a background using a matte.

Select:

Front Back Matte
Back/M Bkg

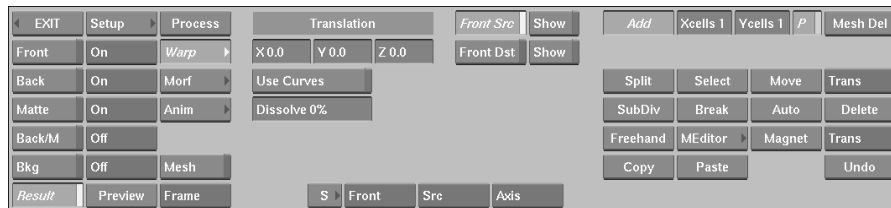
To load:

Front, matte, back,
backmatte, and back-
ground clips

To produce:

A morph using two clips and their
mattes on a background clip instead
of on black.

3. Select the source clip(s).
4. Select the destination reel.
The Warper menu appears.



NOTE: Regardless of which clips you load into the Warper, there are always two compositing layers available: a front layer and a back layer. If you do not load a back clip, black is used for the back layer.

The Warper menu includes the following controls.

Select:

To:

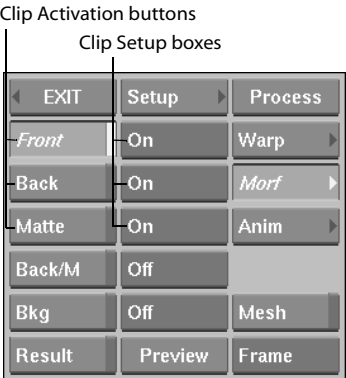
- | | |
|-------|--|
| Setup | Perform setup customization, such as saving and loading setups and preferences, setting playback options, changing display colors, inputting slip values, etc. |
| Warp | Access warping controls. |
| Morf | Access morphing controls. |
| Anim | Access the animation channels and “tweak” the animation curves. |

To reset the Warper:

1. In the Warper menu, click Setup.
2. Click Reset All, and click Confirm when prompted.
Warper options are restored to their default settings.

Setting Up the Clips

Use the Clip Setup boxes next to the Clip Activation buttons to specify how each clip loaded into the Warper will be used.



For Front and Back clips, the Clip Setup box contains On, Off, and Lock options. For Matte and Back Matte clips, the Clip Setup box contains On, Off, and Invert options.

- Select:** **To:**
- On Warp the clip.
- Off Disable the clip. When a clip is disabled, it is not displayed and is not used in the processed result.
- Lock For Front and Back only, process the front or back clip in the final composite without warping it.
- Invert For Matte and Back/M only, invert the front or back matte.

Activating Clips

The following table shows how to set up the clips to produce various effects.

Front	Back	Matte	Back/M	Bkg	Effect
On	Off	Off	Off	Off	Warp the front clip only. This is the same as loading a front clip only.
On	On	Off	Off	Off	Morphs between two clips. This is the same as loading a front clip and a back clip.
On	Lock	Off	Off	Off	Wipes (page turns, rolls); regional warps (if the same clip is loaded as the front clip and the back clip).
On	Off	On	Off	Off	Warp the front clip and matte clip, composited on black.
On	On	On	Off	Off	Warp over a background with a matte clip.

Front	Back	Matte	Back/M	Bkg	Effect
On	Lock	On	Off	Off	Warp the front and matte clips composited on the back clip.
On	On	On	On	On	Morphs between two clips and their mattes onto the selected background clip.

Creating Meshes

After you load and set up your clips in the Warper, you create meshes to define the shape and position of the input image—the original image before the warp or morph—and the output image—the result of the warp or morph. The meshes use control points to determine the change, in shape and position, and create a convincing transition from the input image to output image.

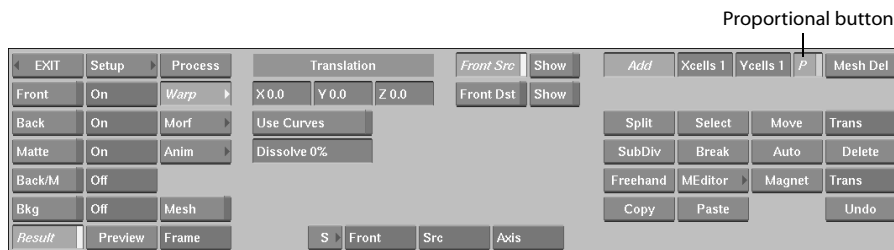
You use two meshes—a source mesh and a destination mesh—to generate the warp effect. For instance, if you want to morph a square into a circle, the *source mesh* defines the input image, which is a square, and the *destination mesh* defines the output image, which is a circle.

Defining a Mesh

When creating a mesh, you anchor the mesh to the surrounding area. To ensure this, create the mesh over a slightly larger area of the image than just the part you want to affect. For example, to warp a person’s mouth, you should extend the mesh to cover the lower half of the person’s face. Doing so “anchors” the area surrounding the warp so that it is not affected by the warp. You begin by defining the number of patches in the mesh and then placing it over the required area in the image.

To define a mesh:

1. Select the clip and go to the frame where you want the effect to begin.
2. In the Warper menu, click Add.



3. In Xcells and Ycells fields, set the number of patches for the mesh. For example, to create a mesh that is 10 patches wide by 5 patches high, enter 10 in the Xcells field and 5 in Ycells field. Enable the Proportional button to automatically adjust the X and Y patch fields to the same value, creating a proportional mesh. For example, if you enable the Proportional button and then type 3 Xcells, Ycells shows the same value, thereby creating a 3 x 3 mesh.

Adding Patches

A patch is part of a mesh that you can add to an existing mesh when the mesh you define has too many spaces or not enough patches. Initially, when you create a mesh, all mesh patches are the same size.

To add patches:

1. In the Warper menu, click Add.
2. To add only one patch, click the edge of the patch that is closest to where you want to add a patch.

The new patch is approximately the size of the adjacent patch.

3. To add several patches, drag from the edge of an existing patch over the area in which to add patches.

Each new patch added using this method will be approximately the size of its adjacent patch.

Deleting Patches

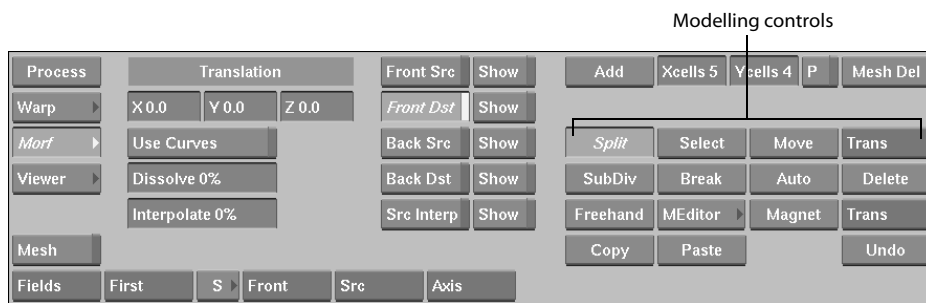
You can delete patches if you make a mistake while creating patches for the mesh.

To delete patches:

1. In the Warper menu, click Delete.
2. To delete a patch that lies along the boundary of the mesh, click the middle of the patch.
3. To delete a line that connects two patches that are not along the boundary, click on both sides of the line.

Modifying a Mesh

Use the modelling controls in the Warper menu to modify the mesh such that it defines the original and the final shape of the image. The modelling controls can be used to modify the shape and size of the patches, modify the location and orientation of the mesh, divide the patches into smaller subpatches, and split the edges into smaller segments.



NOTE: The more detail you add to the mesh, the more control you have when warping or morphing a clip. By contrast, if you have too much detail, the meshes may be difficult to modify.

If you cannot see the mesh properly against the colours in the image, you can customize the colours of the various meshes, as well as those of the splines, tangents, and vertices. For more information, see “Customizing Mesh Colours” on page 848.

Moving Control Points, Edges, and Tangent Handles

In a mesh, you can manipulate control points and tangent handles to add precision to your warp effect. You click control points to view that handles that are tangent to it. You click tangent handles to change the shape of a mesh segment. Tangent handles can be broken apart from control points to make them work independently. Independent tangent handles tangent hands are filled.

Click the Move button to move a control point, an edge, or a tangent handle.

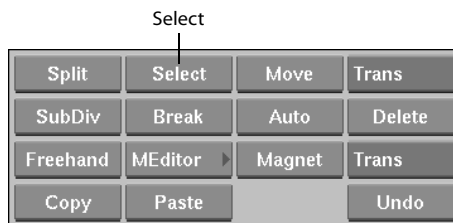
To:	Press:
Move a control point	On the point and drag. Note that the X, Y, and Z coordinates of the selected control point appear in the corresponding fields in the menu.
Move an edge	Near the centre of the edge and drag.
Move a tangent handle	On the corresponding control point to display its tangents, then press on a tangent handle and drag.

Selecting Multiple Control Points

Selecting multiple control points is useful if you want to translate, scale, or rotate all or part of the mesh. Doing so maintains the spatial relationship between the control points in that part. If a control point has four handles, you can click one handle of each pair to make all four handles work independently.

To select multiple control points:

1. In Warper menu, click Select.



2. Hold down **CTRL** key and drag a selection box over the control points that you want to select.

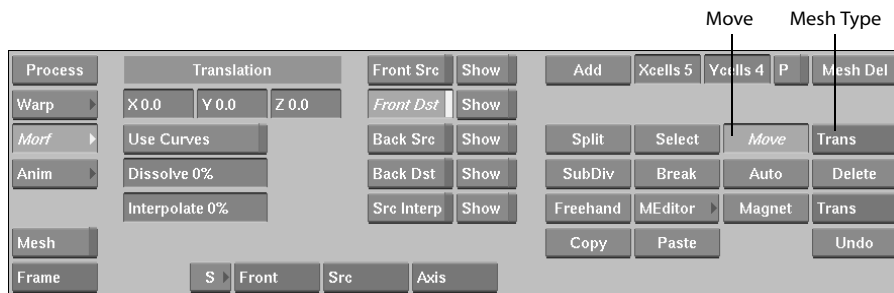
NOTE: To deselect individual points, click the points. To deselect the whole thing, click outside the area of selected control points.

Translating, Scaling, and Rotating the Mesh

Use the Translate, Scale, and Rotate options to change the position, size, and orientation of part or all of the mesh. For example, if you are warping or morphing a live-action clip, you can track the image by moving part or all of the mesh. For more information on tracking an image, see “Modifying the Meshes for Live-Action Clips” on page 836.

To translate, scale, or rotate the mesh:

1. Select the appropriate area of the mesh. See “Selecting Multiple Control Points” on page 830.
2. In the Mesh Type box, select Trans, Rotate, or Scale.
3. Click Move.



4. Click one of the selected control points and drag.

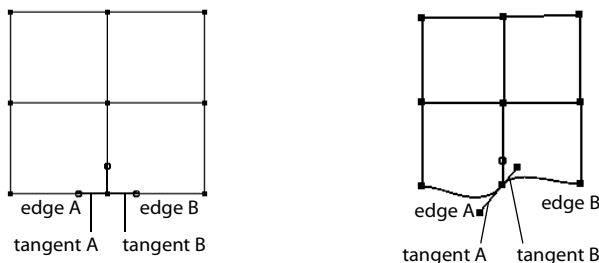
The selected area of the mesh is translated, scaled, or rotated in the direction indicated by the cursor movement.

NOTE: You can also translate, scale, and rotate the entire mesh using the Translate, Scale, and Rotate channels in Channel Editor. However, you cannot use Channel Editor to affect only part of the mesh.

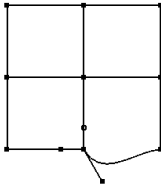
Breaking Tangents

By default, a control point's tangents are not independent of each other. Moving one tangent has the inverse effect on the opposite one such that the joint at the intersection of the two edges remains smooth. In certain cases, you may want tangents to move independently of each other.

Consider the mesh in the following example:



Suppose that you want to move tangent A to change the shape of edge A. Moving tangent handle A also causes tangent B to move in the opposite direction (as shown above); tangent A and tangent B are locked. You can use the Break command to break the lock and move the tangents independently (as shown here).



After using the Break command to break the lock between two tangents, you can use the Auto command to rejoin or “unbreak” the two tangents. Note that resetting the tangents also resets the corresponding edges to their default positions.

To break tangent handles:

1. In the Warper menu, click Move.
2. Click a control point to display its tangents.
3. Click Break, then click a tangent handle.

The tangent and the opposite one are now independent of each other and the tangent handles are solid indicating they are broken.

NOTE: If the control point has four tangents, click one handle of each pair to make all four tangents move independently.

To reset broken tangent handles:

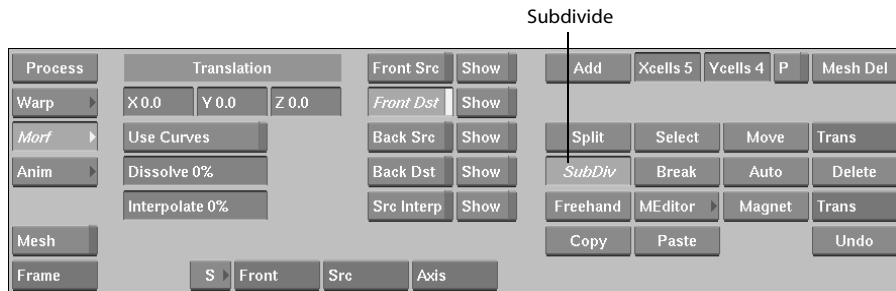
1. In the Warper menu, click Move.
2. Click on a control point to display its tangents.
3. Click Auto, then click a tangent handle. The tangents and the shape of the edges are automatically reset to their default positions.

Subdividing a Patch

You can divide patches in a mesh into smaller subpatches and split the edges into smaller segments using the SubDiv button.

To subdivide a patch:

1. In the Warper menu, click SubDiv.



2. Click the edge of a patch and drag the cursor to the opposite edge.

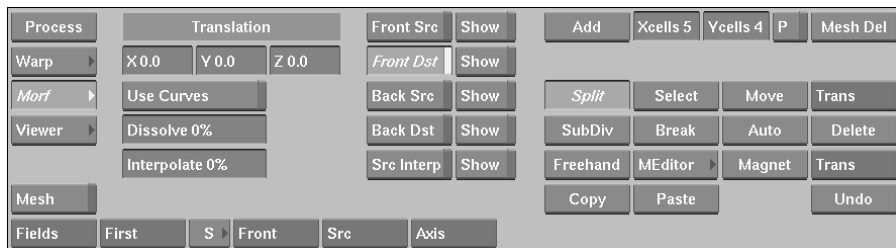
NOTE: To delete an edge created with SubDiv, enable the Delete button and click on the edge.

Adding a Control Point

Adding a control point along an edge gives you more control to manipulate the edge.

To add a control point:

1. In the Warper menu, click Split.



2. Click on the edge of a patch where you want to add a control point.

NOTE: To delete a control point created using the Split button, click the Delete button, then click the control point.

Using the Freehand Mesh Tool

Use the Freehand Mesh tool to perform intricate mesh modeling on an existing mesh.

To create a freehand mesh:

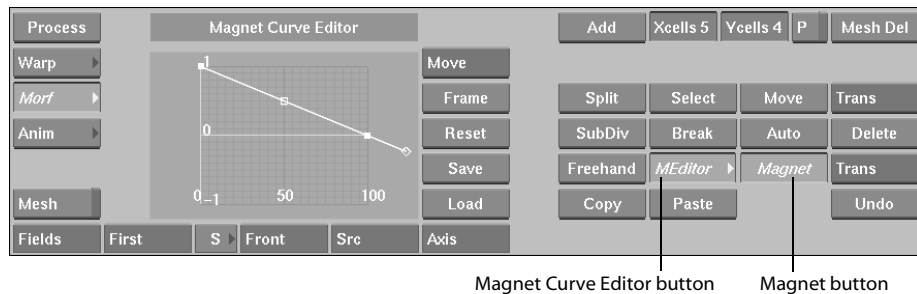
1. In the Warper menu, click Freehand.
2. Click the edge of a patch, and continue clicking until you complete the mesh.
3. **CTRL**-click on the final control point.

To delete a freehand mesh:

1. Click Delete.
2. Click the spline or control point you want to delete, or click Undo.

Warping with the Magnet Tool

Use the Magnet tool to define the shape of the warped image. The Magnet tool warps the area of the mesh under the Magnet. You can set the size of the Magnet and define its effect on the selected area.



Note that the Magnet's effect is most noticeable when the selected area includes many control points (that is, when the grid has many patches, when you are using a large Magnet, or both).

To use the Magnet tool:

1. In the Warper menu, click Magnet.
2. Place the cursor over the mesh and adjust the Magnet size by holding down **CTRL+S** and dragging.
3. Place the Magnet over the control points you want to warp and click.
The affected control points turn red.
4. Drag the Magnet to produce the effect you want.

NOTE: By default, the Magnet translates the mesh. To scale or rotate the mesh, select the Scale or the Rotate option in the selection box beside the Magnet button.

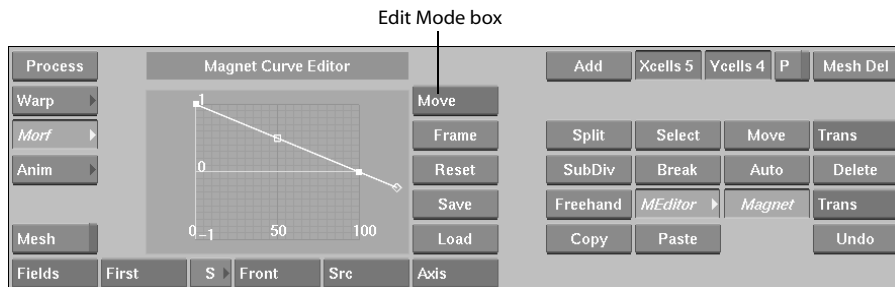
Customizing the Magnet's Effect

You can customize the Magnet's effect by changing the shape of the curve in the Magnet Curve Editor. Note that, by default, the Magnet has the greatest effect on the pixels at the centre of the Magnet and the least effect on the pixels at its perimeter.

To customize the Magnet's effect:

1. In the Warper menu, click MEditor.

The Magnet Curve Editor appears.



2. Adjust the shape of the Magnet curve.

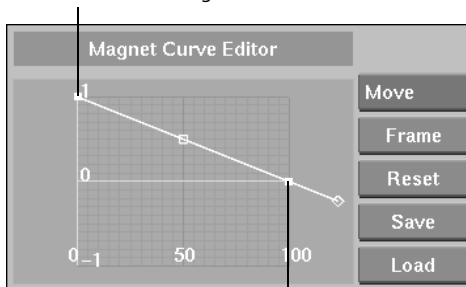
You can also use the Edit Mode options to modify the curve.

NOTE: You can save and load custom Magnet settings using the Save and Load buttons in the Magnet Editor.

Tips and Tricks

You can experiment with the Magnet curve to achieve different effects:

This point represents the effect at the centre of the Magnet



This point represents the effect at the perimeter of the Magnet

- Invert the curve to affect the pixels at the perimeter of the Magnet more than those at the centre.
- Flatten the curve along the horizontal axis to have the same effect on all pixels under the Magnet.

- Add extra control points and create a sine or wave curve to make the control points move in opposite directions.
- Move the curve into the negative horizontal axis to create a magnet with an opposing magnetic effect.

Modifying the Meshes for Live-Action Clips

If you are warping or morphing a live-action or moving clip, you have to define the source mesh at several frames in the clip. The Warper interpolates the location and shape of the mesh between frames.

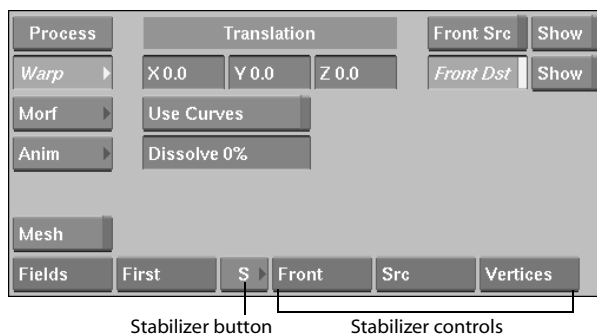
Creating a mesh over the length of a live-action or moving clip can be time-consuming. To simplify the mesh-building process, create the mesh for the first frame of the required clip, then use the Stabilizer to generate the successive meshes.

NOTE: You can track the source or destination mesh for either the front or the back clip.

With the Stabilizer, you can track the movement of the object in the clip, and then apply the tracking data to the mesh (tracking by Axis). This translates the mesh so that it follows the object over the length of the clip. Alternatively, you can track the movement of selected points in the object, and apply the tracking data to the mesh (tracking by Vertices). This translates and changes the shape of the mesh so that it matches the position and shape of the object over the length of the clip.

Loading a Clip into the Stabilizer

Use the Stabilizer controls in the Warper menu to select the source or destination clip and the tracking mode.



Select:	To:
Front	Choose the front clip.
Back	Choose the back clip.
Src	Choose the source clip.

Select: To:

- Dst** Choose the destination clip.
Note that this option can be useful if front and back destination meshes are different.
- Axis** Track the axis of the mesh.
You can use one or two trackers in the Stabilizer.
- Vertices** Track and move mesh control points (vertices).
You must select the control points you want to track before entering the Stabilizer.
You can track up to 1000 control points.
Note that tracking by vertices sets a shape keyframe at every frame of the clip.

For more information on using the Stabilizer, see Chapter 38, “Tracking and Stabilizing.”

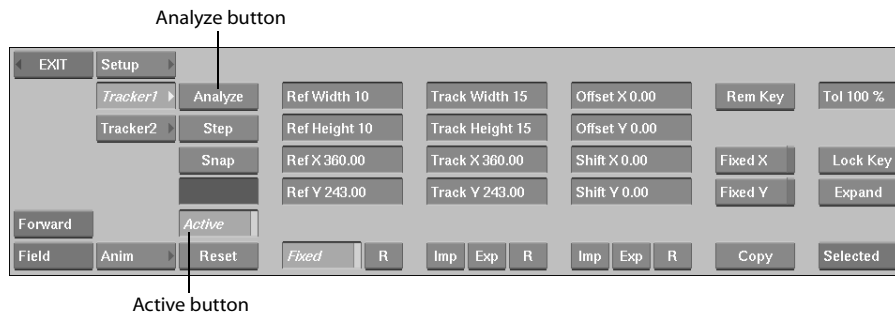
Tracking by Axis

When warping or morphing a live-action clip, the mesh must track the movement of the object. You can track movement using the object’s axis as the point of reference for generating tracking data in the Stabilizer.

To track by axis:

1. Use the Stabilizer controls to choose a clip and set the tracking mode to Axis, then click the Stabilizer button.

The Stabilizer menu opens.



By default, only one tracker is activated. To activate the other tracker, click the Tracker2 button, and then enable the Active button.

2. Position the tracker box on the image.
For more information, see “Positioning the Reference Box” on page 794.
3. Click Analyze to generate the tracking data.
4. If necessary, fine-tune the analysis.

- When you are satisfied with the tracking results, click Exit to apply the results and return to the Warper.

If you want to save the tracking setup, click Setup and then click Save.

Tracking by Vertices

When you track movement using the vertices (control points) of a mesh, a keyframe shape is set at *every frame* of the clip.

To track by Vertices:

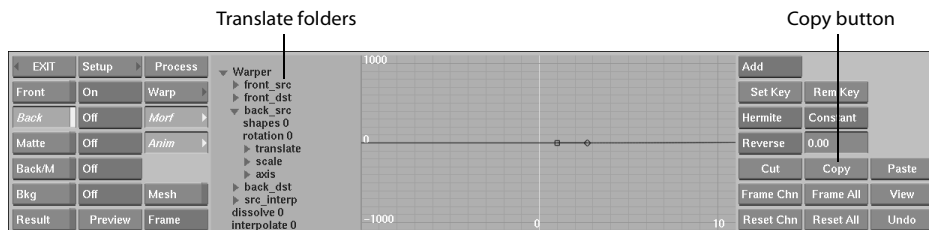
- Select the control points to track from the mesh. Refer to “Selecting Multiple Control Points” on page 830 for more details.
- Use the tracking controls to choose a clip and to set the tracking mode to Vertices, then click the Stabilizer button.

The Stabilizer menu opens. In the image, a tracker appears on each vertex that you selected. The reference frame is the frame from which you opened the Stabilizer.

- If necessary, adjust the position of the reference and tracker boxes to ensure that each tracker has a good reference point.
- Click Analyze to generate the tracking data, and if necessary, fine-tune the analysis.
For example, disable a problem tracker and re-analyze, as described in “Correcting Errors during Analysis” on page 812.
- When you are satisfied with the tracking results, click Exit to apply the results and return to the Warper.

Copying and Pasting Tracking Data

You can copy and paste tracking data from one mesh to another mesh. To do so, click the Anim button to display the Channel Editor. For more information, see “Copying and Pasting Channels or Keyframes” on page 156.



If tracking was done by axes:

1. Select both the X and Y channels in the frnt src translate folder.
2. Click Copy.
3. Select both the X and Y channels in the frnt dst translate folder.
4. Click Paste.

The translation channels containing the tracking data from the front source are pasted onto the front destination.

If tracking was done by vertices:

1. Select both the X and Y channels in the frnt src shapes folder.
2. Click Copy.
3. Select both the X and Y channels in the frnt dst shapes folder.
4. Click Paste.

The shapes channels containing the tracking data from the front source are pasted onto the front destination.

Copying and Pasting Meshes

For a still clip, use the Copy and Paste buttons on the Warper menu to copy a mesh from one clip—the front source mesh—and paste it onto another clip (for example, the front destination mesh).

For a live-action clip, use the Channel Editor to copy the mesh curve from one clip and paste it onto another clip. The mesh on the image window will also be copied. This method is faster since it copies all keyframes (representing translation information over several frames of a clip) in only one action.

To copy and paste meshes for a still clip:

1. Create the mesh in the first frame of a clip, and then click Copy in the Warper menu to copy the current mesh — including its translation, scale, and rotation values.
2. Go to the required frame and display the mesh (source or destination) where you want to paste the copy.
3. Click Paste.

To copy and paste meshes for a live-action clip:

1. Create the mesh in the first frame of a clip and modify it over several frames to define the movement of the object in the clip. Then click Anim to view the Channel Editor.
2. Select the shapes curve in the current clip (frnt src, frnt dst, back src, or back dst).
3. Click Copy.
4. Select the shapes curve in the clip (frnt src, frnt dst, back src, or back dst).
5. Click Paste.

NOTE: If Auto Key in the Setup menu is enabled, a keyframe is created in the associated animation channel at the current frame every time the mesh is copied to a new frame and modified. When you copy the shapes curve to a new clip, you copy the mesh and all its keyframes.

Warping

Warping changes the shape of the image in a clip. When you warp an image, you start by drawing a mesh over the region of the image or object you want to warp. You then reposition the mesh over the image at specific keyframes to continue the distortion of the image. The repositioned lines mark the change in shape of the object.

When defining the mesh and creating the warp, you need to set up a *source mesh* and a *destination mesh* for the frames in the *front clip*. The source mesh defines the area of the image that you want to warp; it outlines the shape of the non-distorted object in each frame of the clip. The destination mesh defines what the warped image looks like; it outlines the shape of the distorted object in each frame of the clip. Typically, you will set the destination mesh at several frames of the clip. The Warper generates a smooth transition between consecutive frames.

The general steps for warping:

1. Load the appropriate images into the Warper. If you are warping just a region of the clip, load the same clip as the Front and the Back clip (see “Warping a Region of a Clip” on page 842).
2. In the Warper menu, click Warp.
3. Set up the clips (see “Setting Up the Clips” on page 827). In general:
 - Set Front to on.
 - If you are warping just a region of the clip, set Back to Lock.
4. Click Front to see the front clip.
5. Click Front Src to define the meshes for the front source clip.

Defining Original Image Meshes

You define meshes using modelling tools, such that the source mesh outlines the original shape of the object to warp. See “Modifying a Mesh” on page 829.

If you are using a live-action or moving clip, the source mesh must follow the movement of the image. Click Front Src in the Setup menu and change the frame of the clip and add more of the mesh over the area to warp. Alternatively, use the Stabilizer to track the movement in the clip. See “Modifying the Meshes for Live-Action Clips” on page 836.

To define the mesh for the original image:

1. Go to the first frame of the clip.
2. Add a mesh over the area to warp.

Defining Warped Image Meshes

When defining the warped image mesh, you must ensure the destination mesh is the same as the front source mesh at the first frame. You do this by copying the front source mesh and pasting into the front destination mesh.

To define the mesh for the warped image:

1. In the Setup menu, click Front Dst.
2. Go to the first frame in the clip.
3. Copy the front source mesh.
4. Paste the mesh into the front destination mesh.
5. At several frames in the clip, modify the mesh to outline the required shape of the warped image.

NOTE: Click the Result button to see the pixels move as you move the mesh points.

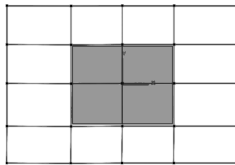
6. When you finish modifying the meshes, go to frame 1 and click Process to process the clip.

Warping a Still Clip

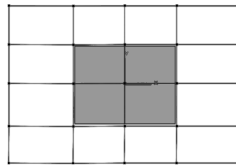
To warp a still clip, you define the source mesh only for the first frame of the clip since the object to warp has the same shape and location in each frame of the clip. Typically, you have to define the destination mesh only for the first and the last frames of the clip. The Warper generates a smooth transition between the two frames.

The following illustration shows the source and destination meshes used to warp a square into an oval.

Source Mesh

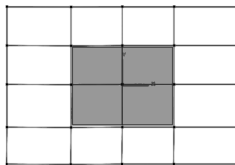


First frame

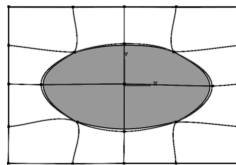


Last frame

Destination Mesh



First frame



Last frame

Warping a Region of a Clip

When a clip is processed, only the area defined by the mesh is processed. If you want to warp only a region in an image (known as regional warping), but you want to see the entire image in the result, you must load the same clip as the front and the back clip, and then lock the back clip.

Morphing

Morphing gradually transforms an image in a front clip into an image in a back clip. The effect is achieved by warping the two images and dissolving between the front and back clips. Unlike warping, which requires only a front source mesh and a front destination mesh, morphing requires both a *source mesh* and a *destination mesh* for both the *front clip* and the *back clip*.

The front source mesh defines original shape of the image in each frame of the front clip. Similarly, the back source mesh defines the original shape of the image in each frame of the back clip.

NOTE: You should create the mesh over a slightly larger area of the image than just the part you want to affect (see “Defining a Mesh” on page 828).

Both the front and the back destination meshes correspond to the warped image. Since the front clip transforms into the back clip:

- At the first frame, the front and back destination meshes correspond to the original shape of the front source clip.
- At the last frame, the front and back destination meshes correspond to the final shape of the back source clip.

You can either use Source interpolation to automatically modify the destination meshes or copy and paste the meshes to set the keyframes manually. For more information, see “Using Source Interpolation” on page 847 and “Setting the Keyframes Manually” on page 848.

The general steps for morphing:

1. Load appropriate images into the Warper. You need to load at least a front clip and a back clip.
2. In the Warper menu, click Morf.
3. Set up the clips (see “Setting Up the Clips” on page 827). In general:
 - Set Front to On.
 - Set Back to On.
4. Enable Front to see the front clip.
5. Click Front Src to define the meshes for the front source clip.

Defining Original Image Meshes

A good morph effect is based on well-drawn meshes. Drawing meshes can be a time-consuming process especially for a live-action or moving clip. If you are using a live-action or moving clip, the source mesh must follow the movement of the image. For specific details about drawing meshes, see “Creating Meshes” on page 828.

Repeat this procedure at several frames in the clip, and make sure Front Src is enabled in the Setup menu. You can also use the Stabilizer to track the movement in the clip.

To define the mesh for the original image:

1. Go to the first frame of the clip.
2. Add a mesh to the object to be morphed.
3. Modify the mesh, using the modeling tools, such that the mesh outlines the original shape of the object to be morphed, see “Modifying a Mesh” on page 829.

Defining Morphed Image Meshes

The back source mesh always contains the same number of control points and patches as the front source mesh. Therefore, do not add patches or control points to the mesh you pasted into the back source clip because the result in the front clip may be hard to predict.

If you are using the Stabilizer and you need more detail for the back source clip, add subdivisions to the back source. While adding subdivisions to the back source, be sure to add the same subdivisions to the front source in a way that conforms to the front source shape.

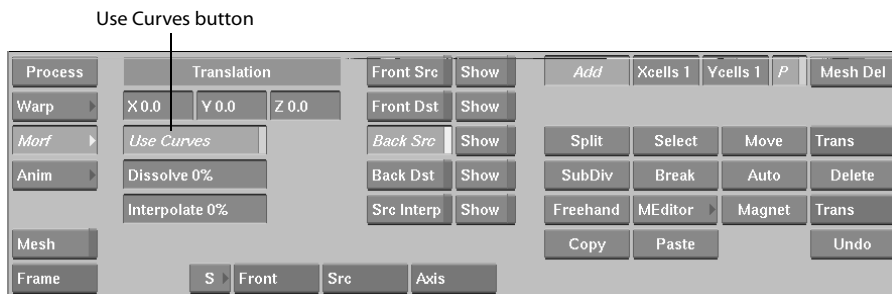
NOTE: Enable the Fixed button in the Stabilizer menu.

To define the mesh for the morphed image:

1. In the Morf menu, click Back Src.
2. Go to the first frame in the clip and copy the front source mesh.
3. Paste the mesh into the back source clip.
4. Click Back to see the Back clip.
5. Modify the mesh, using the modelling tools, such that the mesh outlines the original shape of the object.

To create the transition from the original image to the morphed image:

1. In the Morf menu, enable Use Curves.



2. Modify the front destination mesh and the back destination mesh using Source interpolation. See “Using Source Interpolation” on page 847.

NOTE: You can also modify the destination meshes manually (see “Setting the Keyframes Manually” on page 848).

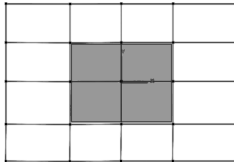
3. When you finish modifying the meshes, go to frame 1 and click Process to process the clip.
4. To view the result, enable Result and play the clip.

Morphing Between Two Still Clips

To morph between two still clips, you define the source meshes only for the first frame of the front source and back source clips (since the objects have the same shape and location in each frame of the clips). You can use Source interpolation to create a smooth morph.

The following illustration shows the source and destination meshes needed to morph a square into a circle.

Front Source Mesh

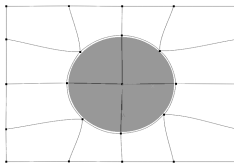


First Frame

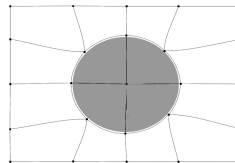


Last Frame

Back Source Mesh

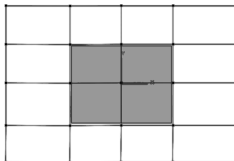


First Frame

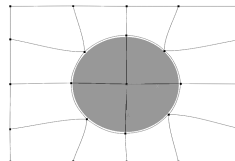


Last Frame

Front and Back Destination Meshes



First Frame



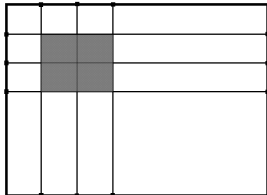
Last Frame

Morphing Between Two Live-Action Clips

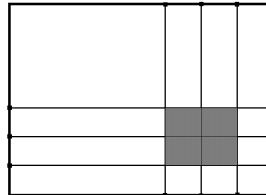
To morph between two live-action clips, you have to define the source meshes for the front source and back source clips as well as for the front destination and back destination clips throughout the duration of the clips—since the objects change shape and location in each frame of the clips. You can use the Stabilizer to track the movement of the objects and Source interpolation to create a smooth morph.

The following illustration shows the source and destination meshes needed to morph a square into a circle, both moving from the upper left to the lower right corner of the image window.

Front Source Mesh

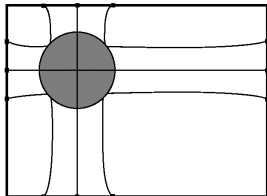


First Frame

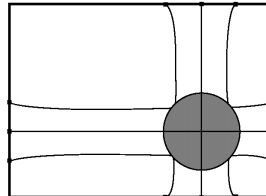


Last Frame

Back Source Mesh

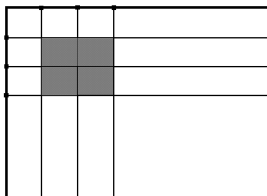


First Frame

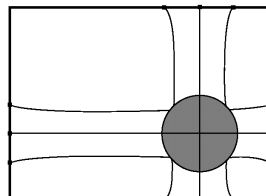


Last Frame

Front Destination Mesh

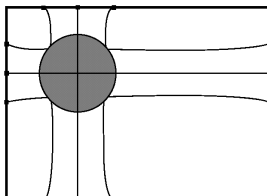


First Frame

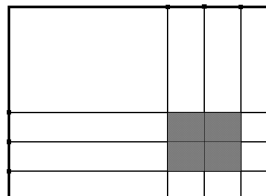


Last Frame

Back Destination Mesh



First Frame



Last Frame

Using Source Interpolation

Use Source interpolation to automatically modify the destination meshes. Source interpolation automatically creates the transition from the front source mesh at the first frame to the back source mesh at the last frame of the clip.

NOTE: If you want to add another effect, such as scaling, during the morph, do not use Source interpolation. You must define the front and back destination meshes manually at several frames (see “Setting the Keyframes Manually” on page 848).

Dissolve and Interpolation Curves

Source interpolation uses two predefined animation curves—dissolve and interpolation—to calculate the percentage change from the front source mesh to the back source mesh over the length of the clip. These values are set in the Dissolve and Interpolate fields:

- The Dissolve field defines the percentage change in the colour of the pixels at the current frame.
- The Interpolate field defines the percentage change in the position of the pixels at the current frame.

The default values in these fields are 0% at the first frame and 100% at the last frame. Using the default values results in a smooth dissolve from the pixels in the front image to those in the back image, as well as a smooth transition from the shape of the front image to that of the back image.

To use Source interpolation:

1. In the Warper menu, click Setup.
2. In the Set Front Dst box, select Src Interp.
3. In the Set Back Dst box, select Src Interp.
4. Click Morf to return to the Morf menu.
5. Enable Use Curves.
6. To view the result of the Source interpolation, click Result and play the clip.

To modify the dissolve and interpolation curves:

1. In the Warper menu, click Anim to view the Channel Editor.
2. Select the dissolve or the interpolate channel.
3. Modify the curve.

NOTE: For more information on using the Channel Editor, see Chapter 9, “Animation.”

Setting the Keyframes Manually

If you are not satisfied with the Source interpolation result, you can set the keyframes manually. However, you will have to set the keyframes throughout the clip—you cannot use Source interpolation in conjunction with manual keyframes.

To set a keyframe manually:

1. Go to the frame corresponding to the keyframe you want to set.
2. Click Front Dst to display the front destination mesh.
3. Modify the mesh.
4. Copy the front destination mesh and paste it into the back destination mesh for that keyframe.

Alternatively, if you want the back destination mesh to always match the front destination mesh, select the Front Dst option adjacent to the Back Dst button in the Warper Setup menu.

Setup Options

From the Setup menu, you can customize mesh colors, save setups, and set processing options.

Customizing Mesh Colours

If you cannot see the mesh properly against the colours in the image, you can customize the colours of the various meshes, as well as those of the splines, tangents, and vertices.

To customize mesh colours:

1. In the Warper menu, click Setup.
2. Click the colour pot beside the item whose colour you want to change.



Colour pots for modifying mesh colours

3. Use the colour picker to select the new colour for the item. For more information, see “The Colour Picker” on page 57.

Saving Setups

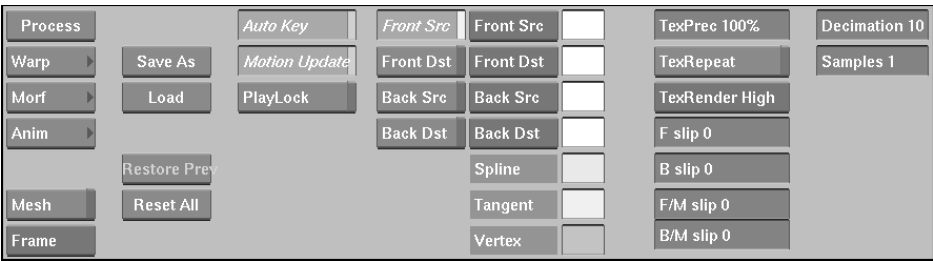
You have two options when saving setups: Setup Morf and Prefs. Morf saves the mesh and the current settings, whereas Prefs saves only the current settings.

NOTE: Custom Magnet curves must be saved separately as described in “Customizing the Magnet’s Effect” on page 835.

For more information about saving setups, see Chapter 3, “Project Management.”

Processing Options

Use the controls in the Warper Setup menu to select the processing options.



Textures

Use the following controls to choose between hardware texture-mapping or polygons for processing.

TexPrec (Texture/Precision Ratio) — Indicates a speed/quality ratio for interaction as well as the rendering quality when TexRender Low is selected as the rendering option. The higher the value, the higher the quality of the display during interaction, and the higher the quality of final processing when rendering under TexRender Low.

TexRepeat — Specifies how pixels are handled when a source mesh goes outside of the borders of an image. When this button is disabled, anything outside of the image is considered black. When the button is enabled, the image is repeated. TexRepeat is disabled by default.

TexRender — Provides three texture rendering options.

Select:	To:
TexRender High	Use high-quality hardware texture rendering. This option always uses the highest level of precision. TexRender High tiles images according to the available texture memory. This option is the default and is the recommended rendering option for film images or any image that is greater than the available texture memory.

Select:	To:
TexRender Low	Use lower-quality hardware texture rendering, specifically during interactive rendering mode or when you want to render an image quickly. This option uses the precision level that you set in the TexPrec field.
Pixel Rendering	Use polygon rendering.

Click Preview to hide the mesh and preview the resulting frame.

Slip — Slips a clip by a specified number of frames. Click in the field for the clip you want to slip and enter the number of frames. The first frame of the clip will be repeated by the number specified to result in a delayed start.

Select:	To:
F	Slip the front clip.
B	Slip the back clip.
F/M	Slip the front matte.
B/M	Slip the back matte.

Decimation

Decimation is the number of subdivisions for the main patches of the mesh (not the small subdivided elements). The default decimation value is 10, which means that each patch is divided into 10 by 10 elements. You should increase this value when:

- You want to use smoother curves along the edges of the patches.
- A patch has been greatly subdivided. When the Decimation value is too low, the subdivided sections will appear to “swim” instead of locking to points and splines.
- A patch is twisted extensively. When the Decimation value is too low, the interior of the patch may appear to fold or break up unexpectedly. Increase the value to smooth the interior of the patch.

Keep the following points in mind when you are adjusting the decimation value:

- Increasing the Decimation value will cause interaction and processing speed to decrease dramatically.
- The Decimation value remains in effect during final processing.
- There is no relationship between the Decimation value and the TexPrec value. The Decimation value defines the precision of the geometry, whereas the TexPrec value affects the precision of the image that is mapped to this geometry.

Samples

The Samples field specifies the anti-aliasing factor. This number specifies the number of samples per pixel during final processing.

Warper Hot Keys

Refer to the following tables for the hot keys that are available in the Warper module.

Freehand Mesh Hot Keys

The following hot keys are only accessible when the Freehand button is enabled in the Warp or Morph menus.

Press:	To:
CTRL-click any control point	Abort freehand mesh draw.
SPACEBAR-drag	Pan freehand mesh draw.
CLICK + ↑	Zoom in freehand mesh draw.
CLICK+ ↓	Zoom out freehand mesh draw.

Selection Box Hot Keys

Use the following hot keys to draw a box and select or unselect multiple control points on a mesh.

Use:	To:
CTR-drag	Draw a rectangular selection box on a mesh.
SHIFT-drag	Draw a rectangular deselection box on a mesh.

Menu Button Hot Keys

Use the following hot keys to enable buttons on the Warper menus.

Use:	To enable:
A	The Add button.
B	The Break button.
D	The Delete button.
F	The Freehand button.
G	The Magnet button.
M	The Move button.
U	The Auto button.
S	The Select button.

Magnet Scale Hot Key

Use:	To:
CTRL+S-drag	Scale the magnet.

View Clip Hot Keys

Use:	To view:
F1	The Front clip.
F2	The Back clip.
F3	The Front Matte clip.
F4	The Result clip.

*In the **flame** Text module, you can add any kind of effect, for example, rotation and colour animation, to layers, paragraphs, and even individual characters.*

Summary

In this chapter, you learn about:

- “Adding Text to Clips” on page 855
- “Defining Styles” on page 873
- “Formatting Text” on page 860
- “Working with Fonts” on page 876
- “Creating Text Rolls and Text Crawls” on page 868
- “Setting Text Options” on page 879
- “Animating Text” on page 872

About Text

In **flame**, text is comprised of layers, paragraphs, and characters. You enter and edit text in layers that you can use for character and layer animation. You can make text characters spin, dance, and change colour.

You use the Text module to add text to a clip, which you can then use in the Action or Batch module depending on the effects you want to achieve. Using the **flame** Text module, you can create effects such as a text roll of credits, text crawls, and bumpers. You can use logo images in a text roll so that the exact logo of a sponsor appears in the credits list.

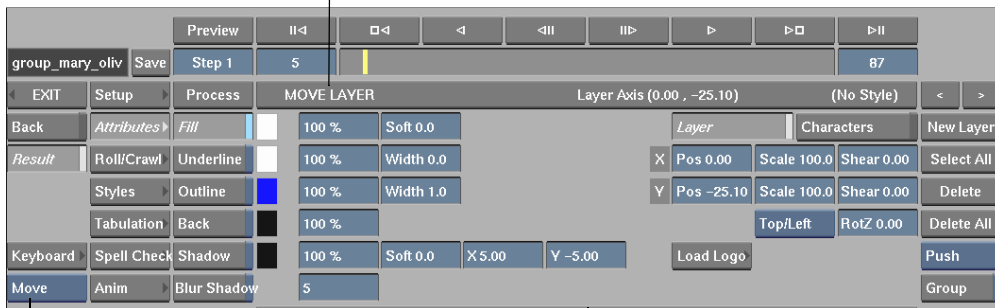
In addition, use Action to create 3D text deformation effects, including sliding, magnifying and oscillating effects. For details, see Chapter 46, “Action: 3D Geometry.”

To access the Text module:

1. In the **flame** menu, click Text.
2. From the desktop work reels, select the clip to which you want to add text.
3. Select a destination for the processed clip.

The Text Attributes menu appears and the source clip is loaded in the image window.

Message bar displays text mode and context



Text Mode box

Swipe bar

The first time you open the Text module in a **flame** session, the frame contains a clip image, but no text. The next time you open it, the text and settings from the previous Text session appear.

Select:

Setup

Anim

Attributes

Roll/Crawl

Styles

Tabulation

Spell Check

Text Mode box

Swipe Bar at
bottom of menu

Swipe Bar at left
and right of menu

To:

Access the Text Setup menu. See "Setting Text Options" on page 879.

Create a text animation by setting different text properties at specific keyframes in a clip. See "Animating Text" on page 872.

Set properties for layers and characters. See "Modifying Layer and Character Properties" on page 862.

Set text motion properties for vertical or horizontal text scrolling. See "Creating Text Rolls and Text Crawls" on page 868.

Create preset text style formats. See "Defining Styles" on page 873.

Tabulate text. See "Tabulating Text" on page 865.

Check text for spelling errors.

Apply various text settings to layers, paragraphs, and characters. Text modes include: Edit, Move, Resize, Rekern, Leading, Y Offset, Safe Title, and Align Sel.

The message bar displays the mode that is currently enabled.

Access the Paragraph menu to set fonts and paragraph properties. Swipe the bar again to switch back to the Text menus.

Display the Channel Editor in the work area. Swipe either the left or right bar again to switch back to the clip and text view.

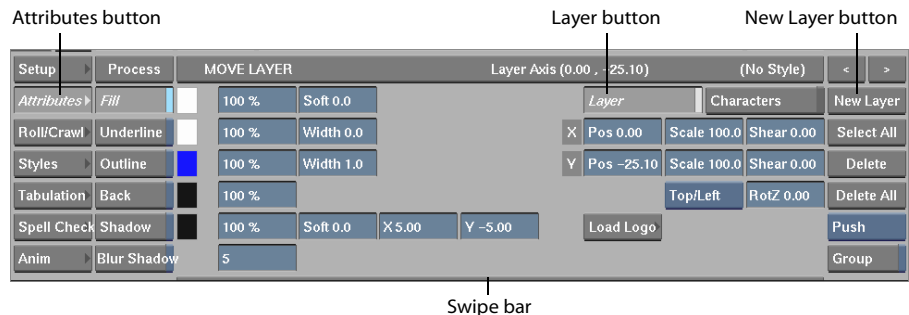
Adding Text to Clips

You add text to a clip by first creating a new layer for entering the text, choosing the font you want to use, and then typing the text.

You can create several layers of text that overlap each other.

To create a layer:

1. Click New Layer.

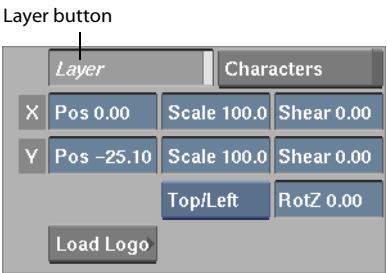


A red ruled boundary appears in the image area indicating Edit mode, and the layer is positioned in the upper left corner of the safe title area by default.

NOTE: The default layer width does not account for the pixel aspect ratio that you specify on the Framestore Setup menu. If the pixel aspect ratio is not 1:1, the default layer width is different from the frame width. See “Maximum Resolution by Product and Frame Depth (FD)” on page 94.

2. To set the text layer attributes, click Attributes and then click Layer.

Each layer has its own parent axis that you use to translate, rotate, resize, and shear the layer using the Layer fields.



For more information about text attributes and text layer properties, see “Formatting Text” on page 860.

Entering Text

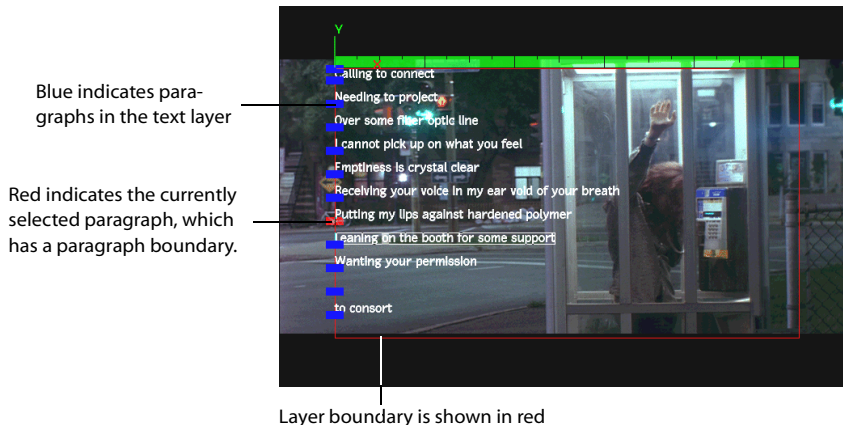
The text you type may be as short as one letter or word or as long as several paragraphs. If the text is long or exists in a word processor, you can load the text file into the current layer.

In the Text module, you can enter and edit characters much like in a word processor, by typing characters using the keyboard and using many known text editing conventions, for example, **SHIFT** + arrow keys to select text lines. Use either the workstation or on-screen keyboard to enter characters in a layer. You can also input Asian characters using their corresponding ASCII codes with the numeric keypad.

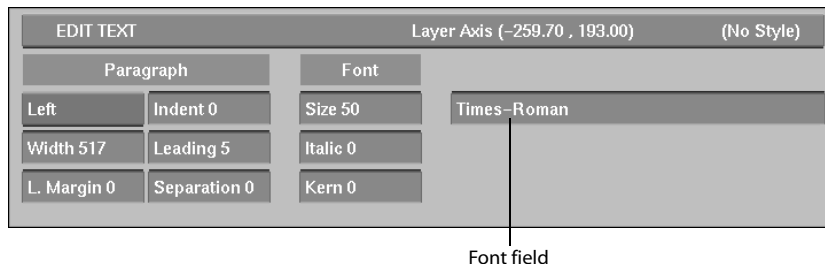
To enter text in a layer:

1. In the Text Mode box, select Edit.
2. In the image window, click the layer and begin typing.
3. Press **ENTER** to begin a new paragraph within the layer.

Notice each paragraph has its own coloured boundary and paragraph tag that is a coloured rectangle in the upper left corner of the paragraph. When you select the paragraph, its boundary colour is red in Edit mode and green in Move mode.



4. Swipe the bar at the bottom of the screen to display the Paragraph menu.



5. Click the Font field to select a new font from the file browser.

See “Working with Fonts” on page 876.

To type extended characters:

1. In the Text menu, click Keyboard.

The on-screen keyboard appears.



Up ASCII button

2. Type the characters in the text layer.

The keyboard characters appear in the current font.

Using the Text Keyboard

When you use the on-screen keyboard, consider the following:

- Special characters use the ISO Latin1 encoding vector. You specify the encoding vector in the *init.cfg* configuration file using the FontMapping keyword. See the **flame** *Installation Guide*.
- The extended keyboard uses the Standard encoding vector.
- Enable the Up ASCII button on the on-screen keyboard to display extended characters contained in the selected font set.

NOTE: Enabling Up ASCII also affects your workstation keyboard, so be sure to disable the button again before exiting the Text module.

- Asian character sets may not display all characters on the on-screen and extended keyboards. Use the numeric keypad or load an ASCII file that contains unavailable characters for the selected font. See “Loading Text Files” on page 882.

Using Text Modes

As you work with text layers, paragraphs, and characters, you use various text modes to manipulate layers, add text to a clip, or edit existing text in a clip. These modes are available from the Text Mode box and are described in the following sections.

Aligning Layers

With the Align Sel mode, you can align multiple layers and specify the direction for the alignment.

To align text layers:

1. In the image window, select two or more layers to be aligned. To select multiple layers, hold down the **CTRL** key and click the layers.
2. In the Text Mode box, select Align Sel or press **ALT+J**.
The numeric keypad appears.
3. Press the number on the numeric keypad that corresponds with the direction in which you want to align the selected layers. For example, press **7** if you want the layers aligned at the upper left corner of the image window.

The text mode returns to Move when the Align Sel operation is done.

Editing Text

Use Edit mode to type text strings, select text, and edit text in layers. When you use Edit mode, the boundary of the selected text layer changes to red. You edit text using the settings that appear on the Attributes and Paragraph menus. See “Modifying Layer and Character Properties” on page 862.

Press **ESC** to switch between Edit and Move text modes. The message bar displays the current mode.

Adjusting Text Leading

Use Leading mode to adjust the spacing between lines of text in a paragraph. The selected lines are adjusted in proportion to the leading values already specified. Use this mode when you want to adjust the leading among paragraphs that have varying leading values.

Moving Text Layers

Use Move mode to move text layers in the image window while the text remains intact. When you use Move mode, the boundary of the selected text layer changes to green.

To move text layers:

1. In the Text Mode box, select Move.
2. Select the text layer you want to move. To select multiple layers, hold down the **CTRL** key and click the layers.
The boundaries of the selected layers are green.
3. Move the layer to the new location and press **ESC** to switch back to Edit mode.

Rekerning Text

Use the Rekern mode to change the kerning of all selected characters in relative proportion to the current kerning values. Kerning refers to the space between characters that you can either increase or decrease.

Use this mode when you want to adjust the kerning among paragraphs that have varying kerning values.

To rekern text:

1. Select the string of characters you want to rekern.
2. In the Text Mode box, select Rekern or press **ALT+K**.
In the Paragraph menu, the Kern field changes to the Rekern field.
3. Enter a new value in the Rekern field or use the up and down arrow keys to rekern the text in single increments. **SHIFT + ↑** or **↓** rekerns in increments of 10 pixels.

Resizing Text

Use the Resize mode to change the font size of all the characters in a selection in relative proportion to the current font sizes. Use this mode if you have several font sizes represented in a selection.

The Size field allows you to apply an absolute font size value to selected text.

To resize text:

1. Select the string of characters you want to resize.
2. In the Text Mode box, select Resize or press **ALT+S**.
The Size field that you used to specify the original size of the text changes to the Resize field.
3. Enter a new value in the Resize field or use the up and down arrow keys to resize the text in single increments. **SHIFT + ↑** or **↓** resizes in increments of 10.

Offsetting Text Layers

Use Y Offset mode to shift selected layers along the vertical axis—the Y-axis. Use the **↑** and **↓** keys to offset the selection.

Using Safe Title

Use Safe Title mode to align selected text within the safe title overlay. Use the numeric keypad (1-9) to specify the direction of alignment. Safe title is used to define the boundary for positioning text and how it will appear in a rendered clip. By default, new layers appear in the upper left corner of the safe title area.

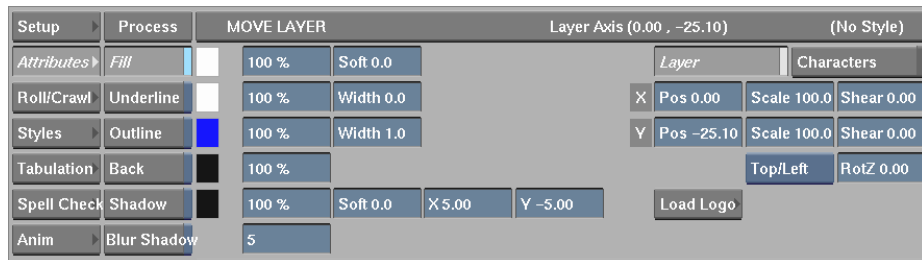
See “Grid and Guides Menu” on page 66.

To display the safe title overlay:

1. Select the string of characters you want to rekern.
2. In the Text Mode box, select Safe Title or press **ALT+Z**.
3. Press the number on the numeric keypad that corresponds with the direction in which you want to align the selected layers. For example, press **6** if you want the layers aligned on the right side of the safe title.

Formatting Text

Use the Attributes menu to change the appearance of individual characters, paragraphs, and layers. In the Text menu, click the Attributes button. In the Text module, layers and characters share similar properties that you set in the Attributes menu.



As you create text formats that you plan to reuse, you can save them using the Styles menu. See “Defining Styles” on page 873.

The general attributes are described in the following table.

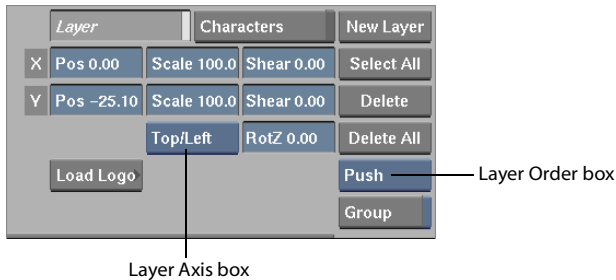
Enable:	To:
Fill	<p>Click the colour pot to choose a colour with the colour picker.</p> <p>To set fill softness, use the Soft field. To increase the softness, enter a value close to the maximum of 100. To decrease the softness, enter a value close to the minimum of -100. Enter 0 for no softness.</p>
Underline	<p>Underline the characters with a solid line using the associated colour, transparency, and width.</p> <p>Click the colour pot to choose a colour from the colour picker.</p> <p>To set underline transparency, enter a percentage in the Transparency field. To make the underline more opaque, enter a value close to the maximum of 100. To make the underline more transparent, enter a value close to the minimum of 0.</p> <p>To set the outline width, enter a value in the Width field.</p>
Outline	<p>Outline the characters with a solid colour using the associated colour, transparency, and width.</p> <p>Click the colour pot to choose a colour from the colour picker.</p> <p>To set outline transparency, enter a percentage in the Transparency field. To make the outline more opaque, enter a value close to the maximum of 100. To make the outline more transparent, enter a value close to the minimum of 0.</p> <p>To set the outline width, enter a value in the Width field.</p> <p>Use anti-aliasing rendering options with outlined text. These options are found in the Text Setup menu. See “Setting Text Options” on page 879.</p>

Enable:	To:
Back	<p>Apply a solid colour background using the associated text layer colour and transparency.</p> <p>Click the colour pot to choose a colour from the colour picker.</p> <p>To set layer transparency, enter a percentage in the Transparency field. To make the layer more opaque, enter a value close to the maximum of 100. To make the layer more transparent, enter a value close to the minimum of 0.</p>
Shadow	<p>Apply a shadow to text characters using the associated colour, softness, transparency, and Pos X and Pos Y fields.</p> <p>Click the colour pot to choose a colour from the colour picker.</p> <p>To set shadow transparency, enter a percentage in the Transparency field. To make the shadow more opaque, enter a value close to the maximum 100. To make the shadow more transparent, enter a value close to the minimum 0.</p> <p>To set shadow softness, use the Soft field. To increase the softness, enter a value close to the maximum 100. To decrease the softness, enter a value close to the minimum -100. Enter 0 for no softness.</p> <p>To set the shadow position on the X-axis, enter a value in the Pos X field. A positive value places the shadow to the right and behind the text characters. A negative value places the shadow to the left and in front of the text characters.</p> <p>To set the shadow position on the Y-axis, enter a value in the Pos Y field. A positive value moves the shadow upward. A negative value moves the shadow downward.</p>
Blur Shadow	<p>Apply a blur effect to a drop shadow you created using the Shadow button. The type of blur applied depends on whether Box Blur or Gaussian Blur is selected in the Text Setup menu.</p> <p>Choose a higher value for a greater shadow blur effect.</p> <p>Using Blur Shadow with animated text decreases processing performance.</p>
Layer	Set properties that apply specifically to text layers.
Characters	Set properties that apply specifically to text characters.

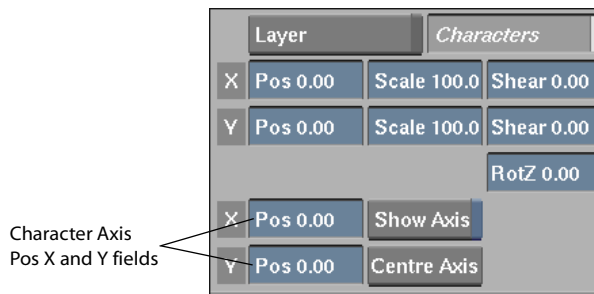
These text attributes options have a cumulative effect on the selected characters. For example, if you enable the Fill and Outline buttons, the text appears as solid characters with a coloured outline.

Modifying Layer and Character Properties

Modify text layers and characters by accessing their respective submenus on the Attributes menu. Use the controls in the Layer area to rotate, translate, scale, or shear a selected layer and to apply an effect uniformly to all characters.



Use the controls in the Character area to view and position the axis of each character. Use the position, rotation, scale, and shear settings for each character within a layer to create an effect of scrambled letters.



To edit text characters, you must be in Edit mode and some text must be selected.

Use:	To:
Pos X	Move the layer or selected characters along the X-axis. By default, the value is 0. Use a positive value to move right, and a negative value to move left.
Pos Y	Move the layer or selected characters along the Y-axis. By default, the value is 0. Use a positive value to move up, and a negative value to move down.
Scale X	Scale the layer or selected characters along the X-axis. This value is a percentage, so entering 50 means 50 percent. Use a larger value to increase along the X-axis. Use a negative value to create a mirror image on the X-axis. Enter 0 to make a layer or disappear. By default, the value is 100.

Use:	To:
Scale Y	Scale the layer or selected characters along the Y-axis. This value is a percentage, so entering 50 means 50 percent. Use a larger value to increase along the Y-axis. Use a negative value to create a mirror image on the Y-axis. Enter 0 to make the layer or selected characters disappear. By default, the value is 100.
Shear X	Shear or slant the layer or selected characters along the X-axis. Use a positive value to slant right. A negative value slants left. The maximum and minimum values are 60 and -60, respectively. By default, the value is 0.
Shear Y	Shear or slant the layer or selected characters along the Y-axis. Use a positive value to shear upward. A negative value near the minimum value -60 shears downward. The maximum and minimum values are 60 and -60, respectively. By default, the value is 0.
RotZ	Rotate a layer around its axis; or selected characters around their axes. Use a negative value to rotate clockwise. A positive value rotates counter-clockwise. By default the value is 0.
Layer Axis box	Set the alignment for the selected text layer according to the following options: <ul style="list-style-type: none"> • Top/Left—moves the axis to the upper left corner of the text layer. • Centre—moves the axis to the centre point of the text layer.
Layer Order box	Move a layer in front of or behind another layer according to the following options: <ul style="list-style-type: none"> • Push—moves the layer one position down in the stack behind another layer. • Pop—moves the layer one position up in the stack in front of another layer. • Bottom—moves the layer to the bottom of the stack behind all other layers. • Top—moves the layer to the top of the stack in front of all other layers.
Load Logo button	Insert front and matte logo clips from the desktop work reels in the current cursor position in the text layer. The Load Logo button appears only when you click Layer.
Character Axis Pos X and Y fields	Set the X and Y-axes for the selected characters. Using the Axis Pos X and Axis Pos Y fields, you set each letter in a word spinning on its own axis in a different way.
Show Axis button	Show the axis in the image area.
Center Axis button	Move the axis to the centre point of the character.

Word Processing Hot Keys

Typical word processing tasks include text selection, cutting, copying, and pasting. The Text module includes several character manipulation hot keys for these tasks.

Press:	To:
ALT+A	Select all characters in the selected text layer.
ALT+P	Select all characters in the selected paragraph within a layer.
ALT+SHIFT+A	Select all characters in all layers. This operation is the same as using the Select All Layers button.
SHIFT+ ↑	Extend the selection one line up.
SHIFT+ ↓	Extend the selection one line down.
SHIFT+ ←	Extend the selection one character to the left.
SHIFT+ →	Extend the selection one character to the right.
SHIFT+HOME	Extend the selection to beginning of current line.
SHIFT+END	Extend the selection to end of current line.
CTRL+SHIFT+PGUP	Extend character selection to beginning of text layer.
CTRL+SHIFT+PGDN	Extend character selection to end of text layer.

Loading Logos

You load logos from the desktop. You can use a logo so that it appears on every frame in a clip. Also, you can use logos in text rolls to credit sponsors, contributors, and products.

Logos can be in a variety of formats available when you choose Import Image in the Library menu. See Chapter 16, “Image Import and Export.”

You can place a logo directly in a paragraph and set text formatting properties for the logo.



Load Logo button

To insert a logo in a paragraph:

- 1. In the Attributes menu, click Layer and then click Load Logo.
The desktop appears.
- 2. Select a front and matte clip for the logo.
The logo is inserted at the cursor position in the text layer.
- 3. In the Paragraph menu, increase or decrease the size of the logo in the text layer by adjusting the font size, leading, and kerning.

Tabulating Text

You can format text into columns by setting tabs in the Tabulation menu. In a text layer, you can also set tab stops to align text at specific locations in a paragraph.



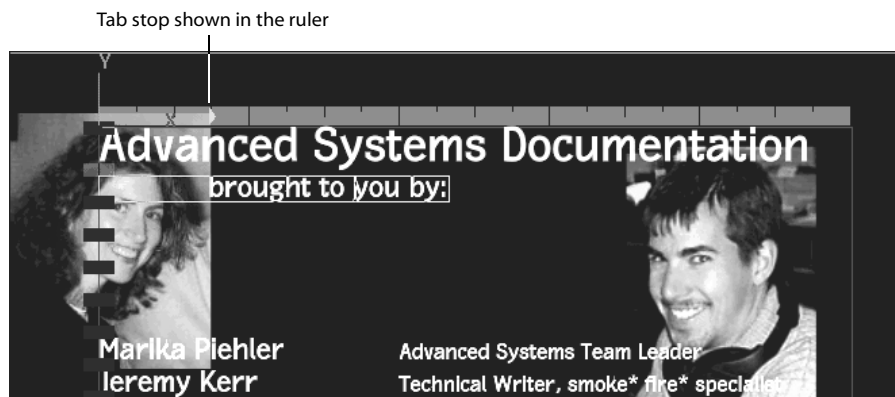
The Tabulation menu controls are described in the following table.

Select:	To:
Add button	Add a tab stop on the text layer ruler. By default, tab stops are set at every 100 pixels as you click Add.
Remove button	Remove the selected tab stop.
Previous button	Select the previous tab stop on the text ruler. Tab stops appear in yellow as they are selected.
Next button	Select the next tab stop on the text ruler.
Tab ID field	Verify tab number in the current paragraph. As you click Next or Previous button, Tab ID field changes, showing number of selected tab stop. This field is display only.
Position field	Set the exact tab stop position on the X-axis. This value is set in pixels.
Justification box	Select Left, Right, or Centre text alignment at the tab stop.
Show Ruler button	Display or turn off the tabulation ruler in the text layer. When you create a new layer, the ruler is shown by default.

To set a tab stop:

1. In the Tabulation menu, click Add to add a tab stop in a text layer.

The tab stop is created according to the current settings in the menu.

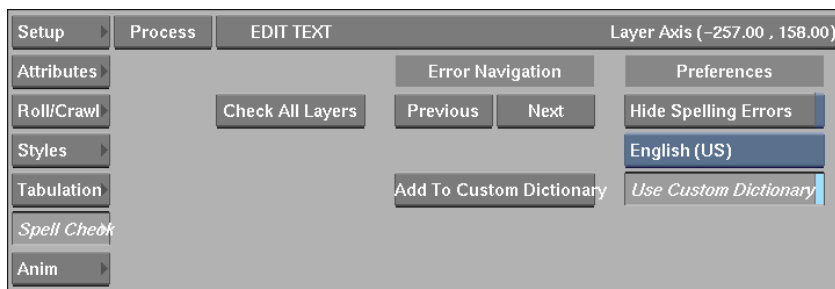


2. To change the position, click the tab stop to select it and drag it to its new location on the ruler. Alternatively, you can set its new position by entering a pixel value in the Position field.

Once you set tab stops for a paragraph, press **TAB** in subsequent paragraphs to type text in the location of the next tab stop. In a text layer, each time you press **ENTER** the tab stops you set in the previous paragraph are carried to the next paragraph.

Spell Checking

In the Text module, you can check the spelling of the text in a text layer. The spell checker draws a red line through words that are not in its dictionary. The spell checker uses the default language and dictionary to verify the spelling of words.



The Spell Check menu controls are described in the following table.

Select:	To:
Check All Layers button	Run the spell checker on the text layers. When the spell checker encounters a misspelled word, it draws a red line through it.
Previous and Next buttons	Navigate to the next or previous misspelled word.
Add To Custom Dictionary button	Add a word that spell checker has deemed misspelled to custom dictionary. The next time spell checker encounters this word it will be ignored.
Hide Spelling Errors button	Hide the red strikethrough line that appears in each misspelled word.
Language box	Select the language the spell checker should use.
Use Custom Dictionary button	Use your custom dictionary with the spell checker. Disable this button to check all spelling.

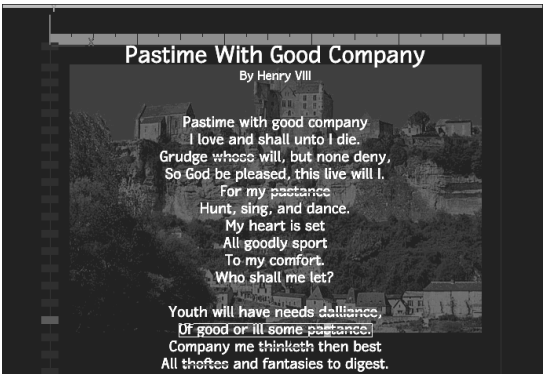
NOTE: If the IRIX spell checker is not installed on your system, the Spell Check menu controls are disabled.

To spell check a text layer:

1. In the Text menu, click Spell Check.
2. In the Edit Mode box, select Edit.

NOTE: You must be in Edit mode to spell check text.

3. Click Check All Layers.
Red lines are drawn through misspelled words.



4. To correct misspellings, click Next or Previous to navigate to each misspelled word.

Grouping Text Layers

Enable the Group button to group two or more layers together. Grouped layers can be saved and loaded as one unit. You can add more text layers to an existing group, resulting in a new single group.

To create a text layer group:

1. In the Edit Mode box, select Move.
2. Press **CTRL** and click to select two or more text layers, or click Select All to select all text layers in the scene.
3. Enable Group.
4. Disable Group to ungroup layers.

NOTE: When a group of layers is selected, you cannot switch back to Edit mode.

Creating Text Rolls and Text Crawls

You can create text rolls on clips, which are commonly used for credit rolls. A text roll is a layer of text rolls from the bottom to the top of a clip for a series of frames. In a text roll, you can use special fonts as well as coloured, animated, and tabulated text.

In text rolls, you can include logos of sponsors, contributors, and products. For example, you can create a list of credits that uses a green, sheared font for all contributors' names and includes a single, larger capitalized letter spinning on an axis at the beginning of each title line. The contributors' names can appear in a white, semi-transparent, Courier-type font.

In the Text module, click the Roll/Crawl button to display the Roll/Crawl menu.

Roll/Crawl button



Enable the Roll button to create a text roll.

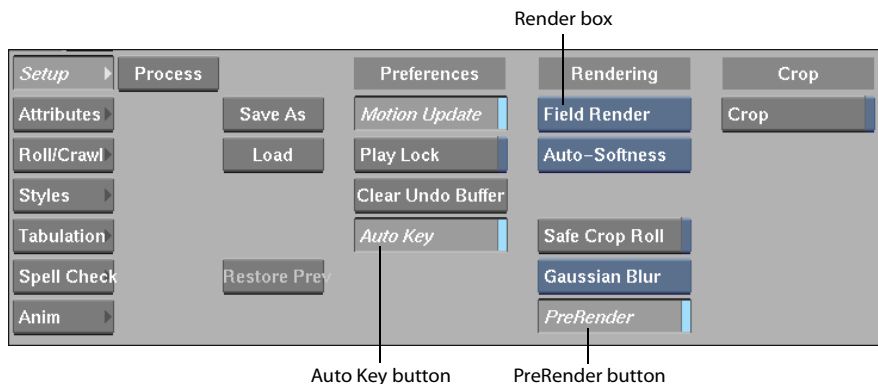
Enable the Crawl button to create a text crawl.

The Roll/Crawl controls are described in the following table.

Select:	To:
New Layer	Create a layer to contain a text roll or text crawl.
Roll	Create text that scrolls vertically over an image.
Crawl	Create text that scrolls horizontally across an image.
Scrollbar X and Y fields	Position the roll layer inside the crop box. These X and Y fields change value when you move the scroll bar at the right of the text layer.
Crop Box X and Y fields	Position the crop box inside the image area. The crop box determines X and Y coordinates of text roll or text crawl on clip.
Width and Height fields	Specify height and width at which to crop the text within the text roll layer.
Fit Best Speed	Create a broadcast quality text roll based on the number of frames in the clip and the lines of text in the text roll. Enable this button to make corrections to a text roll without altering the speed or duration of the clip. For NTSC and PAL, broadcast quality is calibrated at 4 pixels per frame. This rate ensures no flicker in the text roll.
Lock Leading & Scroll	Lock the leading of the paragraph text so that you can insert another paragraph to the text roll. If the duration of the clip lengthens after you make modifications, click Fit Best Speed to adjust the length.

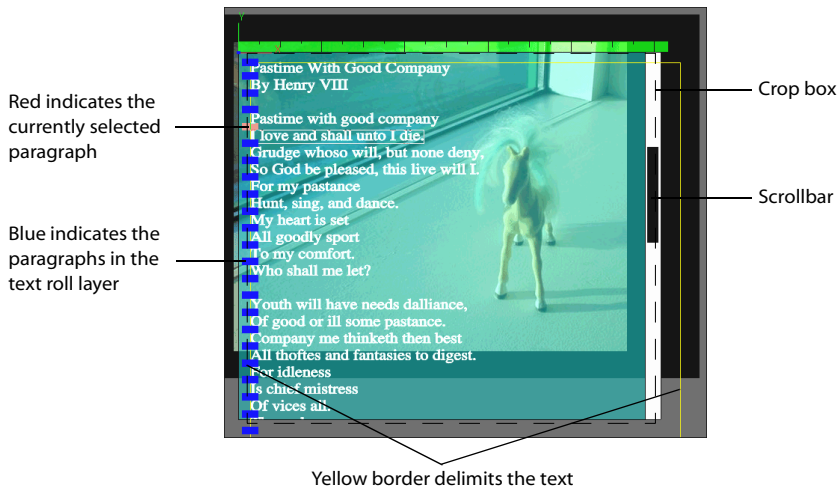
To create a text roll:

1. In the Setup menu, enable Auto Key, enable PreRender, and select Field Render.

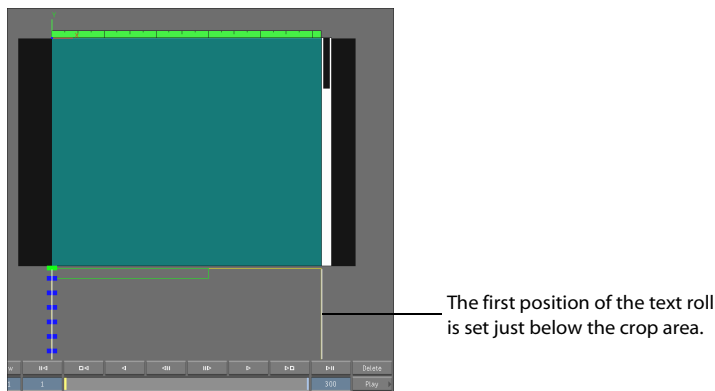


2. Click Roll/Crawl and go to the first frame of the clip and enable Roll.
3. Click New Layer and add the text you want to use for the text roll.

A shaded layer appears in the image area with a vertical scrollbar. The shaded area represents the crop box area.



- To set the first position of where the text appears in the text roll, use the Scrollbar X and Y fields or drag the scrollbar up or down.



- Go to the frame where you want the text roll to finish and set the final position by dragging the scrollbar or using the Scrollbar X and Y fields.
- Click Process to render the text roll and view the results.

When you play the clip, the text rolls through the image area from the first position to the final position.

NOTE: Using PreRender in the Text Setup menu when processing text rolls renders the text roll relatively fast while bypassing the animation.

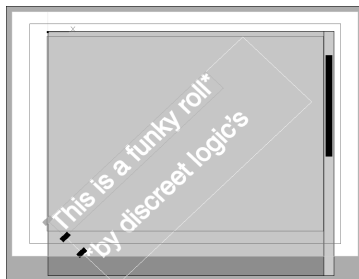
Cropping a Text Roll

The text roll layer defines the text roll region by default. You can control how wide or narrow the text roll appears using the Crop controls on the Roll/Crawl menu. Any text you place outside the cropped area is not displayed during the text roll.

When you rotate, translate, or scale a text roll, some of the text might fall outside of the safe title area. Because of the way the text roll scrolls on a vertical axis—either top to bottom or bottom to top, if part of a paragraph lies outside the safe title area, none of the characters will be visible until the entire paragraph is inside the safe area.

To crop a text roll:

1. In the Roll/Crawl menu, set the width and height of crop box using the Width and Height fields in the Crop Box area.
2. To view the Safe Title guide, click Grid/Guide and select a display option from the Safe box. By default, the Safe Title is set to None. See “Grid and Guides Menu” on page 66.
3. In the Setup menu, enable Safe Crop Roll so that text characters or portions of a paragraph appear as a paragraph scrolls by.



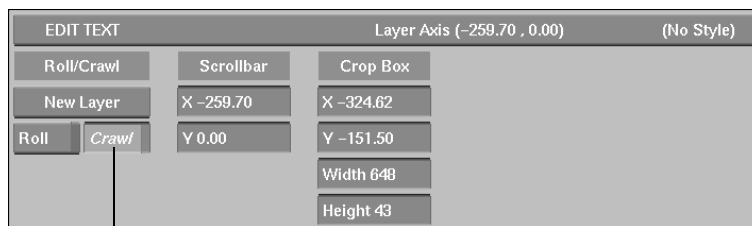
Safe Crop Roll disabled



Safe Crop Roll enabled

Creating a Text Crawl

A text crawl scrolls text horizontally instead of vertically across the frame. For example, you can create a list of phone numbers that scroll from left to right across the bottom of the screen during a telethon broadcast. You create this type of layer in the same way as a roll, except that you must click the Crawl button to apply a horizontal scrollbar.



Crawl button

Animating Text

You can animate colour, softness, size, shadow, and transformation effects such as rotation, scale, and shear.

Use Auto Key to create animations quickly so that as you make changes to text attributes, keyframes are created automatically. The movement between keyframes is interpolated according to the interpolation mode you set, allowing you to create animations quickly.

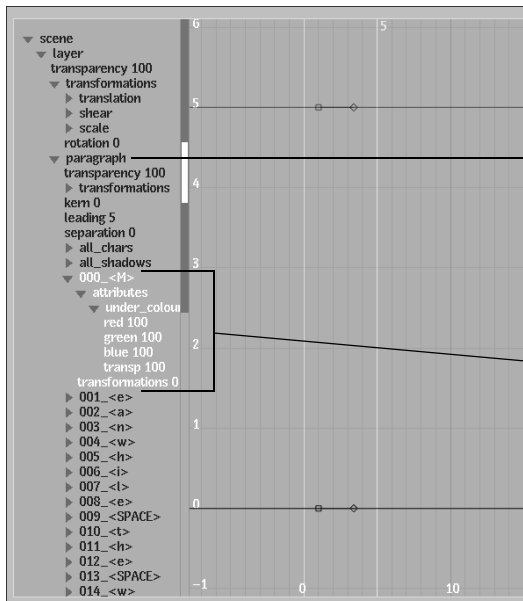
To animate text:

1. In the Setup menu, enable Auto Key and disable PreRender.
2. Go to the first frame in the clip.
3. Set the attributes and transformation properties using the controls in the Attributes menu.
4. Navigate to another frame in the clip and change properties according to the effect you want to produce. See “Modifying Layer and Character Properties” on page 862.
5. Click Process, and then click Play to view the result.

When you play the clip, the animation moves from the first frame to the last.

Animating Paragraph Channels

In the Text module, the Animation menu includes a Paragraph Channel View for viewing text channels in the Channel Editor. Use the attributes and transformation channels to animate paragraphs and characters, or fine-tune animations you already created using Attributes menu.

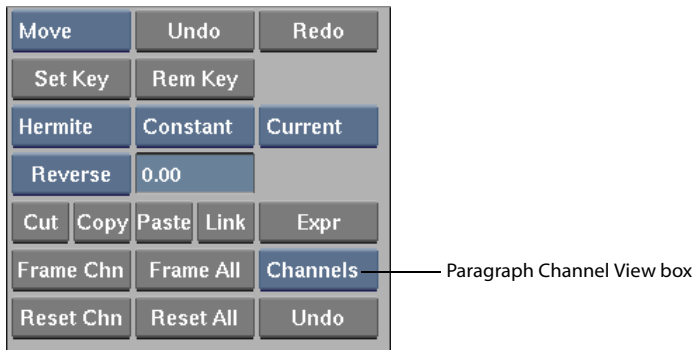


The paragraph folder contains all the characters, including spaces between words.

For each character you can animate specific channels.

To animate paragraph channels:

1. In a text layer, select the text you want to animate.
2. To access the Channel Editor, click Anim on the Text menu.

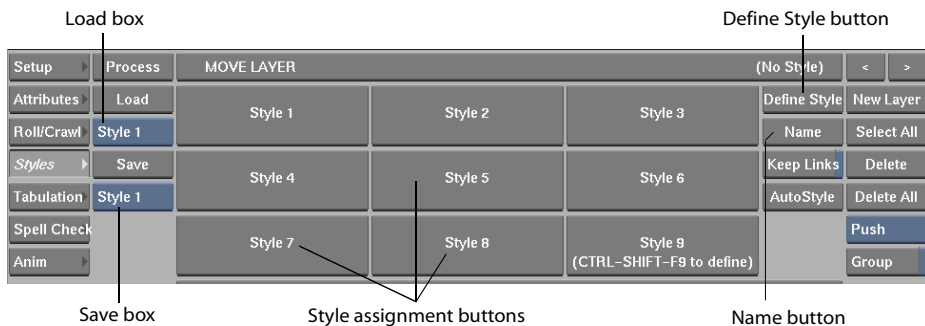


3. Use the Paragraph Channel View box to select the paragraph channels you want to view in the Channel Editor.

Select:	To:
Current	View the channels for the selected paragraph—current cursor location.
Layer	View the channels for all the paragraphs in the selected layer.
All	View all channels for all paragraphs in all layers.

Defining Styles

Styles are sets of text attributes and specifications that can be programmed and replicated whenever you need to use the same format for later text. You can create and modify styles, and save and load them from the file browser. Click the Styles button on the Text menu to access the Styles menu.



To define a style:

1. Select a string of characters whose style you want to save.
2. In the Styles menu, click Define Style and then click the style button to which you want to assign the style.
3. Click Name and enter a name for the style. Then click the style button to which you want the name to apply.

The new name appears on the button.

To modify a style:

1. Select a string of characters with whose style you want to overwrite another.
2. In the Styles menu, click Define Style and then click the style button to which you want to assign the style.

NOTE: You can use **CTRL+SHIFT** and **F1** to **F9** to assign a style to Style buttons 1 through 9 from within any menu.

To apply a style:

1. Select a paragraph or string of characters to which you want to apply a style.
2. Press **CTRL+F1** to **F9** to apply the style to the selected text.

Saving and Loading Styles

After you create your styles, you may want to save one or all of them. You can load any of them in future sessions.

To save a style:

1. Do one of the following in the Styles menu:
 - Select All Styles from the Save box.
 - Select the style you want to save (Style 1 to Style 9) from the Save box.

2. Click Save.

The Save Styles menu and file browser appear.

3. Choose a directory in either Proxies or Titles mode.

NOTE: If you browse in Titles mode, you can click the Create Dir button to create a new directory in which to save your styles.

4. Enter the filename using the on-screen keyboard.

The file browser automatically closes and the Styles menu reappears.

To load a style:

1. If you want to switch the style used by your current text with the styles that you are loading, enable Keep Links. This button overwrites the currently used style(s) within each section of text with the new styles.

NOTE: Loading styles replaces the current styles available. If you have not saved your current styles, you changes will be lost.

2. From the Load box, select either All Styles or One Style.

Select:	To:
All Styles	Load files saved with multiple styles. The file browser lists setups saved with multiple styles.
One Styles	Load files saved with only one style. The file browser lists setups saved with a single style.

3. Click Load.
The Load Styles menu and file browser appear.
4. Choose a directory in either Proxies or Titles mode and select the style(s) you want to load.
5. If necessary, click Exit Load Text to exit from the file browser.

Applying Existing Styles

If you create a section of text with multiple sets of attributes that you might want to use again, you can use a shortcut to assign each set as a separate style and then save them for future use. The Text module automatically detects the different sets of attributes used in the section and assigns each set to a Style button.

To assign styles from an existing layer using shortcuts:

1. Select the section of text.
2. Click Auto Style on the Styles menu.

NOTE: If you have used more than nine sets of attributes in the selection, only the first nine are assigned to a Style button.

3. Save the styles.

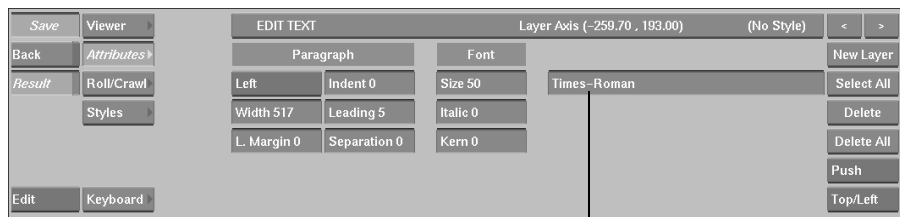
Working with Fonts

When you open the Text module, the default font is loaded and ready to use. You specify the default font using the TextDefaultFont token in the *init.cfg* configuration file. See the “Configuration File” chapter in the *flame Installation Guide*.

When you choose a different font, it becomes the current font and all text you type will appear in the current font. If you edit text that has a different font, the current font is replaced by the font of the cursor position. For example, if you are working with Carta font and you edit text that uses Helvetica, Helvetica becomes the current font and all subsequent text you type appears in Helvetica.

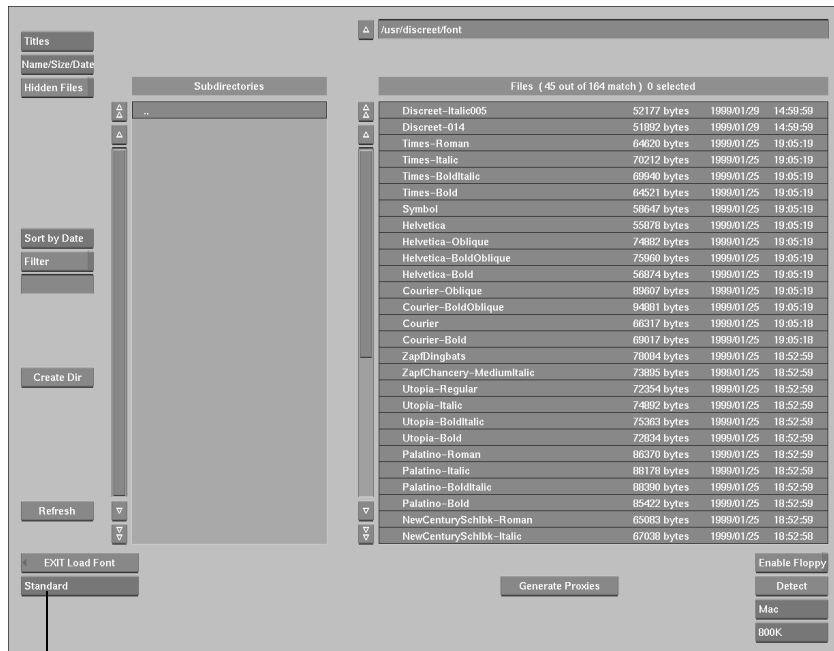
To change the current font:

1. Swipe the bar at the bottom of the screen to display the Paragraph menu.



2. In the Paragraph menu, click the Font field.

The font library appears.



Font Type box

3. In the Font Type box, select the font type you want to load:

Type	Description
Standard	These fonts are furnished by default with flame and are located in the directory <code>/usr/discreet/font</code> .
TrueType	These fonts you can transfer TrueType fonts from a PC to your UNIX system.
Asian	Some Asian fonts are furnished by default with flame and are located in the directory <code>/usr/discreet/font</code> . You can transfer Asian fonts to your UNIX system.

4. In the file browser, navigate to the directory that contains the font you want to load.
5. In the Proxies/Titles box, choose Proxies to preview a font. If the no proxy appears, click Generate Proxies to generate a font proxy.

This process may take a while, but once you create the proxies the first time, **flame** retains the settings. Each time you return to the font library, you can toggle between viewing font titles and font proxies by clicking Proxies or Titles.

6. Select the font.

The Paragraph menu reappears with the name of the new font in the Font field.

Using New Fonts

flame comes with a few basic fonts installed in the */usr/discreet/font* directory. These fonts are available the first time you start **flame**. **flame** supports the following font types:

- Adobe Type1 version 1.0 and third-party fonts that follow the Adobe standard. Fonts that adhere to the Adobe Type1 standard contain the naming convention “%!PS-AdobeFont-1.0” in its list of font files.
- TrueType
- Asian
- CID (Character Identifier) type fonts. CID fonts are a new format of composite Type 1 fonts that address the requirements of Asian fonts and Far East fonts. CID fonts were developed to support the large character set fonts used with PostScript language printing.

To use new TrueType fonts:

1. Copy the TrueType fonts to the directory */usr/discreet/fonts*.
2. In the Paragraph menu of the Text module, click the Font field.
The Font file browser appears.
3. In the Load Font menu, select TrueType from the Font Type box.
The TrueType fonts appear in the list.

NOTE: Fonts produced by software applications such as FontMonger are not Adobe Type1 version 1.0 fonts. You can convert FontMonger fonts using the conversion utility described in the following section. Once converted, you can load the fonts into **flame**.

Additional fonts are supplied with the Display Postscript software and are found in the following directories:

- */usr/lib/DPS/outline/base* (font program)
- */usr/lib/DPS/AFM* (Adobe Font Metrics)

Place any other fonts that you want to load into **flame** in these directories.

To use new Display Postscript fonts:

1. Start **flame** by typing:

flame -F

The -F option creates the link between the font directory and the */usr/lib/DPS/...* directories, making the fonts available in the Text module.

Converting Nonstandard Fonts

You can use SGI's *convertmacfont* utility to convert PC and Macintosh Postscript fonts into a format that is compatible with **flame**.

NOTE: The *convertmacfont* utility does not work well with non-roman typefaces.

To convert PC and Macintosh fonts:

1. Using FTP Explorer or a similar file transfer utility on your PC or Macintosh, select the Postscript fonts you want to convert.
2. In the transfer utility, set the transfer format to Binary mode.

NOTE: This type of transfer is usually called "Mac Binary" in programs such as TCP Connect.

3. Transfer the font files to the SGI machine in either of the following directories:

/usr/lib/DPS/outline/base directory or

/usr/lib/DPS/AFM if using an Adobe Metric Font

4. On your SGI system, go to the directory where you transferred the font files.
5. Type:

```
convertmacfont <font name>.bin
```

For example, to convert the Myriad font, type:

```
convertmacfont myriad.bin
```

6. After converting the font, clean up the directory by deleting the file *.bin*.
7. Start **flame** by typing:

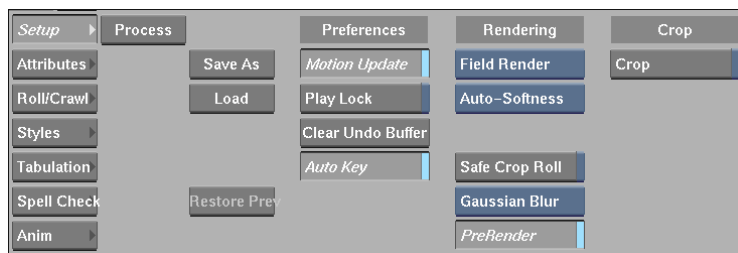
```
flame -F
```

The *-F* option forces **flame** to read the new font.

Setting Text Options

The Setup menu has options for setting preferences, text rendering modes, and grid and guides for the image area. You use these options for text animation, motion, softness, blur, and system performance while rendering text effects.

Click the Setup button to display the Text Setup menu.



The Setup menu controls are described in the following table.

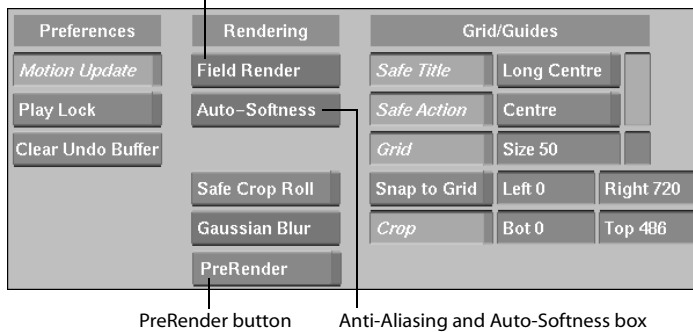
Select:	To:
Motion Update button	Control the playback of animated text. When enabled, animated text and text position is updated dynamically in the image area when you play the clip.
Play Lock button	Control the playback of frames and to view the playback of animated clips, exclusively. When enabled, the first frame is locked while you play the clip
Clear Undo Buffer	Reset the Undo entries.
Restore Prev button	Restore factory preference settings in the Text module.
Frame Render box	Select either Frame rendering or Field rendering. See "Processing Text" on page 881.
Auto-Softness box	Select the sampling level. Auto-Softness mode is set by default, and provides a softness equal to the display quality that you see within the Text module. Selecting an anti-aliasing of 4, 9, or 16 increases the sampling rate, but it also increases processing time. Selecting an anti-aliasing of 1 means no anti-aliasing is performed and the result is a hard edge.
Safe Crop Roll button	Display paragraph text even when part of the paragraph lies outside the Safe Title area in a text roll. See "Cropping a Text Roll" on page 871.
Blur box	Select Gaussian Blur or Box Blur. Gaussian Blur uses sub-pixel resolution and creates a more subtle effect with the Blur Shadow attribute. Gaussian has rounded, smoother edges and is better for animation, however the processing time is increased. Box Blur has rectangular, rougher edges, but is more economical with processing time especially if you are working on a rough draft.
PreRender button	Accelerate rendering on a static layer only, such as a text roll. PreRender has no effect on text layers that have animation.
Crop button	Create a custom-size overlay.

Processing Text

You can verify the process quality of the current frame with the Preview button. You can also sample the text effect you created for the current frame before processing the entire clip. You can process clips in frame or field rendering modes. Select either Field Render or Frame Render on the Setup menu before processing your finished clip. Keep in mind that Field Render produces a better result, especially if you use keyframes that are far apart from each other, but it takes longer to process.

Processing options are located in the rendering options column on the Setup menu.

Field Render or Frame Render box



To process a clip with text:

1. Position the clip at the first frame or the particular frame from which you want to start processing.
2. In the Text menu, click Process.

The clip is processed from the currently displayed frame until the end of the clip.

The Text processing options include a prerendering function called PreRender that speeds up the rendering process for a layer; however, animation in the text layer is ignored.

Saving and Loading Files

You can save, load, and import text files, text setups, and images for specific use with the Text module. You can save the text effects you create, the preferences you set in the Text module, and new default text module settings. You can load text files from third-party word processing packages, existing **flame** text module setups, and logos.

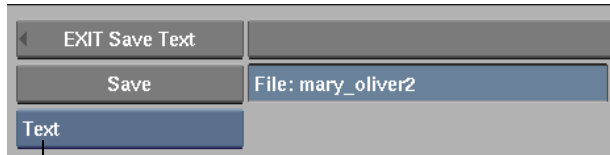
Saving Text Files

When saving files, use the Save options to save text setups, preferences, and default settings.

To save a file:

1. In the Setup menu, click Save As.

The Save menu and file browser appear.



Save Options box

2. In the Save Options box, select an option.

Select:	To:
Preferences	Save Setup menu specifications only and omit text and text attributes.
Text	Save text file with text attributes and Setup menu specifications.
Defaults	Override the default menu specifications in the Setup menu and replace them with your specifications.

3. Navigate and choose the directory where you want to save the file.
4. Type a name for the file and press **ENTER**.

The file is saved and you return to the Text menu.

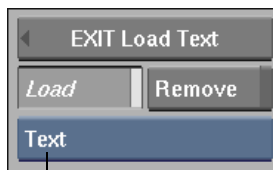
Loading Text Files

When you load files in the Text module, use the Load options to load text files, preferences, ASCII files, or reset the Text module settings with its factory defaults. The Load menu also contains the Remove button so that you can discard files you no longer use.

To load a file:

1. In the Setup menu, click Load.

The Load menu and file browser appear.



Load Options box

2. In the Load Options box, select an option.

Select:	To:
Preferences	Load Setup menu specifications that you have saved.
ASCII	Load ASCII files and text files from third-party word processing products.
Multiple Text	Load several flame text files at a time with preferences. Text module preferences that were saved with the selected file are loaded as well.
Text	Load a text file created in flame 's Text module. Text module preferences that were saved with selected file are loaded as well.
Factory Defaults	Load the original Text module settings that were delivered with the software.

3. Choose a directory in either Proxies or Titles mode.
4. Select the file that you want to load in the file browser.
If you select Multiple Text, press **CTRL** and click the files you want to load.

NOTE: If you do not select Multiple Text and you use the **CTRL** key to load multiple text files, only the last selected file is loaded.

5. If necessary, click Exit Load Text to exit from the file browser.

[illegible]

The specific procedures for using Sparks are as varied as the Sparks themselves.

Here, you'll learn how to load and use Sparks in general.

Summary

- In this chapter, you learn about:
- “Loading a Spark” on page 886
 - “Using a Spark” on page 886
 - “Developing Sparks” on page 888

About Sparks

Sparks are software plug-ins that have been created by Discreet or by third-party developers. Load and use Sparks from the effects menu. Since the exact procedure for using a Spark varies according to its function, only general information can be provided here.

In the Effects menu, there are five Spark buttons. These buttons allow you to use software plug-ins that have been created by third-party developers for use in **flame**.

Keyer	Sparks	L	Spark buttons
Compositor	Sparks	L	
Quick Composite	Sparks	L	
Optics	Sparks	L	
Modular Keyer	Sparks	L	

Loading a Spark

You can load up to five different Sparks at one time.

To load a Spark:

1. Click one of the Spark buttons in the Effects menu.

Keyer	Sparks	L
Compositor	Sparks	L
Quick Composite	Sparks	L
Optics	Sparks	L
Modular Keyer	Sparks	L

The L indicates that the button has not yet been used to load a Spark.

The Spark Library menu appears. The menu consists of the usual file browser and library control buttons.

2. Select a Spark from the library.

You are returned to the previous menu, and the name of the Spark appears on the selected button. The L no longer appears on the button.

If you want to load a new Spark but all of the Spark buttons are already in use, you must replace one of the existing Sparks with the new Spark.

To replace an existing Spark with a new Spark:

1. Press **ALT** and click the existing Spark button.

The Spark Library menu appears.

2. Select a Spark from the library.

You are returned to the previous menu, and the name of the new Spark appears in the selected button.

Using a Spark

A Spark functions in the same way as any other command or module in **flame**, and can be designed for use at either or both of the desktop and the module levels. Sparks can be used as Batch nodes.

The Spark Command Menu

At the desktop level, a command menu appears in the menu when you click the Spark button. For example, this is similar to the command menu that appears when you click the Timewarp button. The Spark command menu can have up to six displays that you use to set processing modes and to enter numeric parameter values.

To use a Spark at the desktop level:

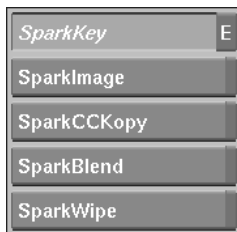
1. Click the Spark button.
The Spark command menu appears.
2. Choose the processing modes and enter the numeric parameter values as appropriate for the selected Spark. These values are specific to each spark.
3. Select the source clips and the destination reel.
The processed clip appears on the destination reel.

The Spark Menu

A Spark menu consists of control buttons and displays, an image window, viewing tools, animation curves and Channel Hierarchy if applicable.

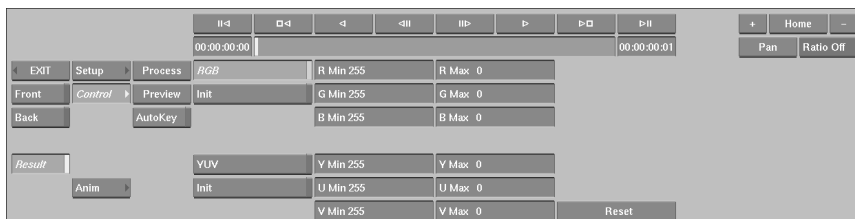
To use a Spark at the module level:

1. Click the Spark button.
2. Select the source clips.
If there is a component menu for this Spark, an E appears on the button.



3. Click the E on the Spark button.
4. Select the destination reel.

The component menu appears.



5. Use the controls in the menu as you would for other components.

Developing Sparks

A Spark Developer's Kit is available from Discreet. For information, send an e-mail message to sparkdev@discreet.com or contact your Discreet sales representative.



Section 6: Power Compositing

Composite,
animate, and
create particle
effects with
Action.

Action: Overview and Setup Options

Fun with layers!

The Action module is a multilayer compositing tool for creating complex visual effects. Use Action to animate clips in 3D and add camera, lighting, shadow, and particle effects.

Summary

In this chapter, you learn about:

- “Action Chapters in This Guide” on page 891
- “About Action” on page 892
- “Action Setup and Processing Options” on page 896

Action Chapters in This Guide

This chapter provides an overview of Action; its main features are detailed in the chapters that follow. These chapters are designed to provide information both by topic and typical workflow, specifically:

- Chapter 43, “Action: Layers and Surfaces,” describes layers in Action and how they are used as composites of a front and matte clip over a common back clip. You learn how to apply layers to surfaces as well as how to blur, crop, slip, and animate a layer’s front and matte clip separately.
- Chapter 44, “Action: Camera and the Scene,” describes how layers, surfaces, and other objects are placed in the scene with respect to the camera position. This chapter includes a special section about using the Action schematic as a way of working with objects and the camera in the scene.
- Chapter 45, “Action: Light and Shadows,” describes how to light the scene and how to add drop shadows. You learn how to work with surface attributes that pertain to lighting such as flip, ambient and diffused lighting, and specular highlights.

- Chapter 46, “Action: 3D Geometry,” shows how to import 3D models into Action, create 3D text, displace surfaces, and work with texture mapping.
- Chapter 47, “Action: The Particle System,” describes how to use particle generators, particle bouncers, and particle manipulators to create effects such as rain, snow, and explosions with 3D models and surfaces.

About Action

In Action, you create effects and animations by manipulating objects in the scene. Objects you work with in Action include surfaces, light sources, axes, particles, shadows, and the camera. The following terms apply to Action and its interface:

- A *layer* consists of front and matte clip composites.
- A *surface* is the combination of a front and matte layer.
- The *scene* is Action’s representation of 3D world space. It is where objects are placed and animated.
- The *schematic* is an icon representation of the scene. It shows all the objects in the scene and their relationship with each other. Icons in the schematic are referred to as *nodes*.
- A *branch* refers to a group of objects where one object parents one or more child objects.
- *Parent-to-child* describes the relationship between two objects where a parent object is connected to a child object. Transformations that you apply to the parent object are applied to its child object.

Opening Action

When you open Action, you load the front and matte clip for the first layer and the back clip used for the background. You load these clips in the order front, back, then matte.

To open Action and load the first layer:

1. In the Effects menu, click Action.



Action button

2. Select the front, back and matte clips.
3. Select the destination.

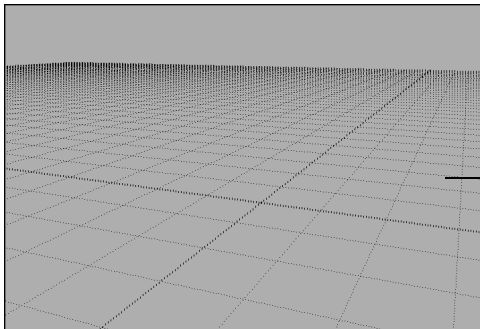
The Action menu appears and the front and matte are loaded in the first layer.

The composite of the front, matte, and back clips appear in the image window and their names are listed in the Layers list. The Total Frames field defaults to the length of the longest clip loaded.

Working with the Camera

With Action, you have the choice of using the automatic camera or the manual camera whose F-Stop, film size, and focal length you can set yourself. You can animate specific camera properties. See “Using the Camera” on page 945.

You can view the scene through the camera as shown in the following figure.



The scene in Action can be thought of as a 3D grid with X, Y, and Z coordinates.

Action Views

In addition to viewing the scene through the camera, you can view the scene in several ways: front, top, side, front layer, matte layer, back layer, context, and schematic views. Front, top and side view are orthographic views used to place and animate objects in the scene, including the camera and its focal point. See “Viewing the Scene” on page 949.

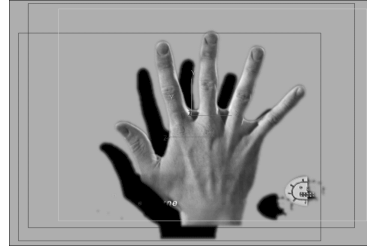
The Schematic view uses nodes to show the objects in the scene, and the relationship between the different objects represented as nodes. Nodes are connected using arrowed and coloured lines. You use axis nodes for animating surfaces and objects, which can be connected to the camera node. You can parent one node to another, creating a *parent-to-child relationship*, yielding complex animations. See “Using the Schematic” on page 955.

Adding Lights and Shadows

After adding a surface to the scene, you can illuminate the surface using light sources. You can animate the position, falloff, and spread of light sources to produce lighting effects.



Use a light source to radiate a cone of light in one direction and illuminate objects in the scene.

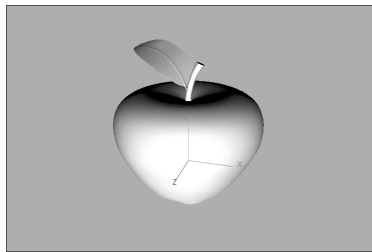


Drop shadows are not cast by lit objects; shadows are cutouts based a surface's matte.

You can also create a drop shadow for surfaces and set lighting options such as shine, transparency, and specular highlights. See “Working with Surface Lighting” on page 974 and “Adding Shadows” on page 978.

3D Models and Text

You can import **3d studio max** files, Wavefront files, Inventor files, and Paint geometry in Action. Also, you can create 3D text with custom bevelling.



An imported 3D model of an apple



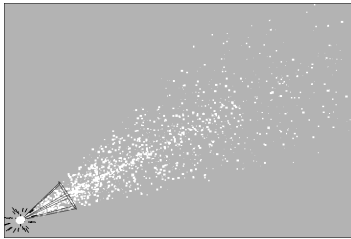
3D text using custom bevelling

You can animate the position, colour, and specular highlight of 3D models and text. You can also apply a texture to 3D models and text. See “Importing 3D Models” on page 983 and “Working with Textures” on page 992.

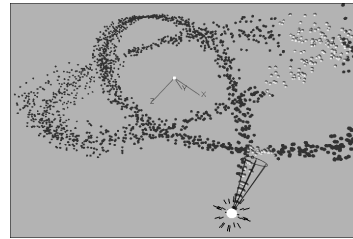
Particle System

Action contains a particle system that can simulate environmental effects such as rain, snow, smoke, and many other user-defined effects.

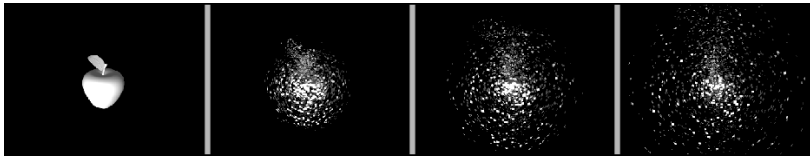
Particle streams are created using a particle generator. Generated particles can be influenced by particle bouncers, used to bounce particles off surfaces, and particle manipulators, used to apply acceleration, rotation, damping, and other transformations. See “Generating Particles” on page 1017, “Working with Particle Bouncers” on page 1047, and “Working with Particle Manipulators” on page 1029, respectively.



Particle generator creating a stream of particles



Particles being influenced by a manipulator



3D model of an apple exploding over four frames

You can also use the particle system to explode surfaces, 3D models, text, or Paint geometry. For more information, see “Exploding Objects” on page 1051.

Accessing Modules through Action

While you work in Action you may want to colour correct or key a layer. Action provides direct access to the Colour Corrector, allowing you to colour correct a layer’s front, matte, or back clip. You can also access the Keyer and Modular Keyer to modify a layer’s matte clip or create a key. See “Using Colour Correction and the Keyer in Action” on page 915 and “Using the Modular Keyer in Action” on page 916.

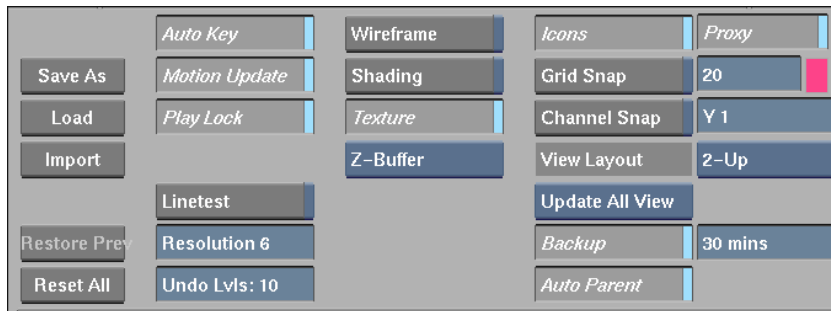
You can access the Stabilizer through Action to apply one- or two-point stabilizing and tracking data to an axis. You can apply four-point tracking data to a bilinear surface. See “One-Point and Two-Point Tracking” on page 798 and “Four-Point Tracking” on page 801.

Action Setup and Processing Options

Use the Action Setup menu to set options for rendering your final Action result, creating animation, controlling motion blur, depth of field, anti-aliasing, and softness, as well as saving, loading, and importing files.

Using the Main Setup Menu

You use the main Action Setup menu to set options for interacting with Action, creating animation, and rendering clips. Click the Setup button to display the Action Setup menu.

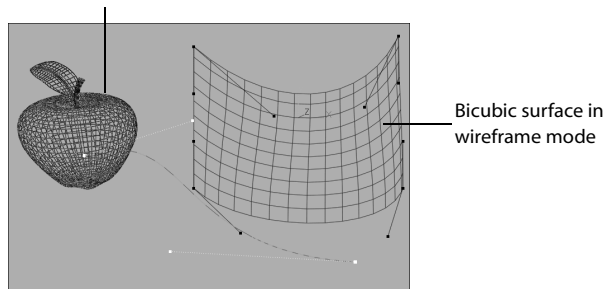


The main Setup controls are described in the following table.

Select:	To:
Save As, Load, Import buttons	Save or load Action setups; import 3D models into Action. See “Saving, Loading, and Importing Setups” on page 899.
Restore Prev button	Revert to the last saved setup. All changes made since the previous Save operation are undone.
Reset All button	Reset all parameters to their default values without deleting objects from the scene. Parent-to-child relationships in the schematic and layers in the Layers list remain intact.
Auto Key button	Set a keyframe automatically when you modify a value in a field or in the Channel Editor. A keyframe consists of a value and the specific frame at which the value is set. When Auto Key is disabled, you must explicitly set and remove keyframes using the Set Key and Rem Key buttons. See “Setting and Removing Keyframes” on page 151.
Motion Update button	Update the position, rotation, colour, and other properties in the scene. This button is enabled by default. When disabled, animated objects do not move, but keep the position of their current value. Disable Motion Update when you want to copy keyframe values from one frame to another. See “Setting and Removing Keyframes” on page 151.

Select:	To:
Play Lock button	Update the animation settings according to the frame or timeline position as you move through the clip while keeping it locked at the current frame. When Play Lock is disabled and you use > or < to play the resulting clip, each frame is loaded and displayed in sequence in the image window.
Linetest button	Process or preview a clip using the resolution value specified in the Resolution field. This is useful if you want to generate an intermediate clip to check your animation. Linetest is ignored when Texture is enabled.
Resolution field	Specify the geometry resolution of surfaces. The lower the value, the better the resolution and the more processing time is required to interact with the image. A value of 1 creates one polygon per pixel on a surface, affording accurate displacement and lighting. See “Setting Surface Resolution” on page 902. Use the value in this field when exploding a surface with a particle generator and when using hardware texture mapping. See “Exploding Objects” on page 1051.
Undo Levels field	Set the number of undoable operations. The Undo and Redo buttons each have a list allowing you to select the most recent operations to undo or redo. Note, you cannot undo a Delete operation for a layer. Setting the undo levels to a large number uses more memory. The default is 10.
Wireframe	Display each surface and 3D model as a wireframe. This is useful if interacting with Action becomes slow because of many surfaces and 3D models. Enable Wireframe to save processing time while setting up the animation.

3D model in wireframe mode



Select:	To:
Shading	<p>Light up scene using added light sources. When Shading is disabled, no lighting effects appear in scene; and surface and 3D models appear flat. Enable Shading for:</p> <ul style="list-style-type: none"> • Light sources • Ambient or diffuse lighting for surfaces • Specular highlights for surfaces and 3D models
Texture	<p>Enable hardware texture mapping, which you can use with any surface resolution. If the image is larger than texture, action creates tiles and does not use a low resolution image.</p> <p>When Texture is disabled, the layer mapping to the surface is done at the specified resolution. See “Setting Surface Resolution” on page 902.</p>
Z-Buffer	<p>Use the distance from the camera eye to determine the order in which objects are rendered when this button is enabled. See “Using Analyze, Z-Sort, and the Z-Buffer” on page 930.</p>
Icons button	<p>Display or hide icons that represent axes, lights, motion paths, and 3D geometry in the scene.</p>
Proxy button	<p>Activate automatic updating of proxies in Schematic view. Action interaction is slower when this button is enabled.</p> <p>When Proxy is disabled, Action updates proxies when you switch views. You can also update proxies by clicking Update.</p>
Grid Snap button and field	<p>Position objects with precision in the scene when this button is enabled. When you move an object in the scene, the object is automatically aligned to the snap grid.</p> <p>To show the grid, select a grid orientation from the Grid box. See “Using a Grid” on page 944.</p> <p>The grid size is set to 50 pixels by default. You can change the grid size by changing the value in the Grid Size field.</p>
Channel Snap button and Y Snap field	<p>Snap keyframe values in the Animation Curves window. When you move a keyframe in this window, it is automatically aligned to the snap grid.</p> <p>The grid size is set to one unit along the X (horizontal) and Y (vertical) axes. To change the grid size along the Y-axis, change the value in the Y Snap field. You cannot change the grid size along the X-axis.</p>
View Layout box	<p>View multiple Action views simultaneously in the image area. Select 2-Up, 3-Up, or 4-Up to set the number of views you want to display. A yellow outline indicates the current view.</p> <p>See “Displaying Multiple Views Simultaneously” on page 950.</p>
View Update box	<p>Select whether you want to update all views interactively or update only the when the selected view is updated while you work with multiple views in the image area.</p>

Select:	To:
Backup button and field	Back up your Action setup automatically. A backup copy of current Action settings is saved in the file named <i>_action.bak</i> in the Action directory. You can set the number of minutes between backups using the Backup field next to the Backup button.
Auto Parent button	Enable automatic parenting in the schematic. Press ALT and drag a node in the schematic to disable Auto Parent temporarily.

Saving, Loading, and Importing Setups

You can save, load, and import Action setups.

To save an Action setup:

1. In the Setup menu, click Save As.
The file browser and Save menu appear.
2. From the Save Action box, select the format for saving the setup.



Select:	To:
All	Save references to all clips in the Layers list. See “Using the Layers List” on page 909.
Raw	Save selected animation channels as a user-readable ASCII file. In the file, each line corresponds to one frame, and each column consists of a frame number and the value of the animation curve at that frame. All saved information starts at frame one. A file with the extension .raw is saved in the <i>.../action</i> directory by default. NOTE: At least one channel must be selected in the Channel Editor.
Text	Save the current text settings, including font, character size, kerning, italics, depth, bevelling curve, and text string properties, all of which can be loaded in another Action session. A file with the extension .atext is saved in the <i>.../action</i> directory by default. NOTE: A 3D text node must be selected in the schematic.

Select:	To:
Preferences	Save the current Action settings as user preferences. A file with the extension <i>.pref</i> is saved in the <i>/usr/discreet/user/effects/user_name/action/pref</i> directory.
Defaults	Save the current Action preferences as Action's new default settings. NOTE: You can restore Action's factory default settings by selecting Factory Defaults in the Load menu.

3. Type a name for the setup file and click Enter.

By default, the setup is saved in the directory */usr/discreet/project/effects/project_name/action*. Using the file browser, you can save setups to the directory of your choice.

To load a setup in Action:

1. In the Setup menu, click Load.

The file browser and Load menu appear.

NOTE: When you access the file browser through Load, you also have the option of removing existing setups.

2. In the Load box, select the format for loading the setup.



Select:	To:
All	Load the clips in the selected setup into their corresponding layers. The layers in the Layers list are replaced with the loaded clips. If a clip cannot be found on the desktop, Action searches for it in the clip library and loads it automatically. If the clip still cannot be found, Action displays the missing clip name in red in the Layers list. A surface that uses a missing clip appears as an outline in the image area and is shown in red in Schematic view.
No Clips	Load a setup without its clips. The current layers in Layers list remain the same.

Select:	To:
Add N+L	Add nodes and layers from the setup file. This option appends the schematic and Layers list from the setup file to the current schematic and Layers list.
Add N	Add only the nodes from the setup file. This option appends the schematic from the setup file to the current schematic.
Raw	Load raw animation data to a selected channel in the Channel Editor.
Text	Load the text setup files. The text settings are loaded into Action's Text menu. To create a 3D text object from loaded settings, click Create in the Text menu. See "Creating 3D Text" on page 985.
Preferences	Load a file containing Action preferences.
Factory Defaults	Load original Action default settings. Selecting this option prompts you to confirm that you want to restore factory defaults and returns you to the Action menu.

3. In the file browser, select the setup you want to load.

To remove an Action setup:

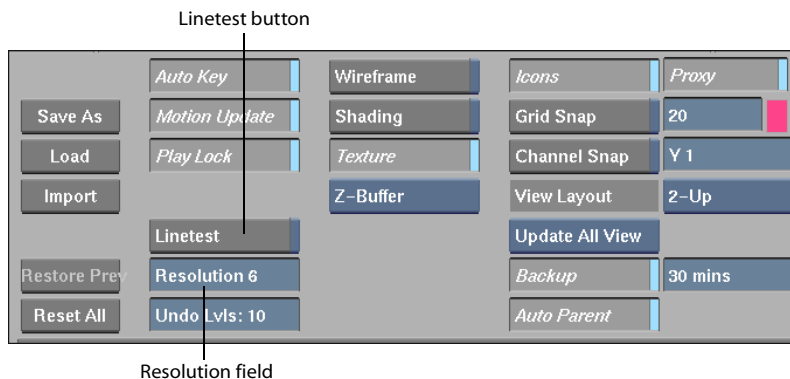
1. In the Setup menu, click Load.
The file browser and Load menu appear.
2. In the file browser, select the setup you want to remove.
3. Click Remove and when you are prompted, click Confirm.

To import a file into Action:

1. In the Setup menu, click Import.
The file browser and Import menu appear.
2. From the Import box, select the file format you want to import.
See "Importing 3D Models" on page 983.
3. In the file browser, select the file you want to import.

Setting Surface Resolution

When you use Action, you work with geometry surface resolution. Surface resolution is set using the Resolution field and Linetest button. You can vary the resolution according to the animation you add to the clip and use Linetest to check it. If you use texture, the system uses full resolution texture even as you interact with **flame**. The image in the image window is displayed at full resolution when you preview or process the clip.



To generate a clip at a lower resolution:

1. In the Setup menu, disable Texture.

Surface layer mapping occurs according to the value in the Resolution field.

2. In the Resolution field, enter a value.

You may want to increase the quality by entering a small value, particularly if you are using lighting, shading, displacement, or particle explosions.

NOTE: To see what happens as you experiment with the Resolution value, add a bicubic surface to the scene and enable Wireframe. Notice the number of surface polygons as you increase and decrease the Resolution value.

3. Enable Linetest.
4. Click Process to render the animation.

The animation is rendered at the resolution that you entered in the Resolution field.

NOTE: Motion blur and anti-aliasing are ignored, but global rendering—either Fields or Frames—is used.

The following examples illustrate processing with Linetest enabled.



Resolution field is set to 2. This is almost the same as processing with Linetest disabled.



Resolution field is set to 10.

Using the Secondary Setup Menu

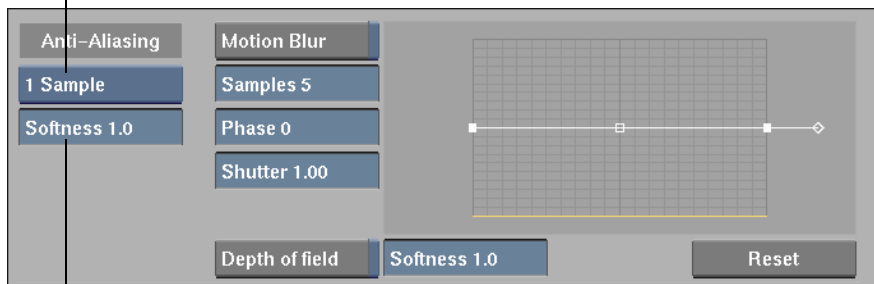
The secondary Setup menu contains advanced Action options that you set to improve rendering and the final output quality of your image.

To display the secondary Setup menu:

1. In Action, click Setup.
2. In the Setup menu, swipe the bar at the bottom the menu.

The secondary Setup menu appears.

Anti-Aliasing box



Anti-Aliasing Softness field

Working with Anti-Aliasing and Softness

The jagged lines that often occur along the edges of diagonal or curved lines when processing high-frequency images such as text are the result of aliasing. Aliasing is caused by insufficient spatial sampling of the image. The process of reducing or removing the effects of aliasing by increasing the sampling rate is called *anti-aliasing*.

Use the Anti-Aliasing box to select the level of sampling. Increasing the number of samples increases the degree of anti-aliasing but also increases the processing time. Values of 4 and 8 give good results. Adjusting the Softness also affects the degree of anti-aliasing. Use a value of 0.5 or 2.0 to get the cleanest image.

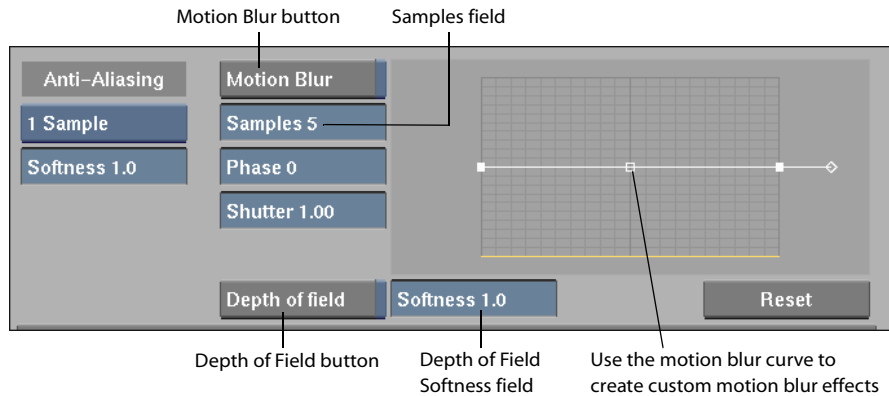
The Anti-Aliasing box and Anti-Aliasing Softness field are used to control the level of anti-aliasing. The best results are obtained when the values are set as indicated in the following table.

Anti-aliasing:	Softness:
1	No effect
2	1
3	2
4	2

When the Anti-Aliasing value is 1, the softness value has no effect on the image. Although the cleanest images are obtained when Softness is set to 0.5 or 2, higher values can also be used for creative effects. However, increasing the level of anti-aliasing also increases the required processing time.

Working with Motion Blur

You can use Action's Motion Blur tool to simulate the blur created by fast-moving objects. Motion Blur is used with both the normal and physical cameras. To use motion blur, enable the Motion Blur button.



Use the following parameters with motion blur.

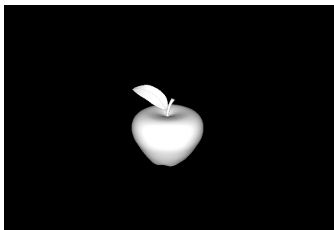
Select:	To:
Shutter field	Control duration of motion blur at each frame, which affects the size of the motion blur. Increasing Shutter value does not increase the processing time.
Samples field	Determine quality of the motion blur produced by the number of samples taken at each frame. Increasing the Samples value causes the processing time to increase linearly and also affects the quality of the depth of field. The number of motion blur samples is multiplied by the number of anti-aliasing samples. To reduce the total number of passes made for each frame, reduce the level of anti-aliasing when motion blur is enabled.

Select:	To:
Phase field	Specify whether the motion blur is based on the movement before or after the current frame. A value of -100 places the motion blur before while a value of 100 places the motion blur after. A value of 0 is centred, which evenly distributes the motion blur. The default value is 0.

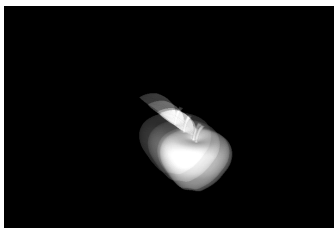
Using the Motion Blur Curve

The motion blur curve controls the sample weight over the scope of the motion blur. The point on the left is the weight of the first sample and the point on the right is the weight of the last sample. By changing the curve, you can create custom motion blur effects such as a Gaussian blur.

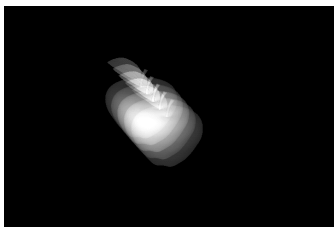
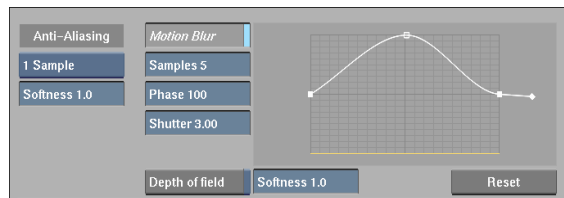
As with the Channel Editor, you can add keyframes to the motion blur curve using the Add mode, move keyframes with Move, and modify the curve's shape using tangent handles.



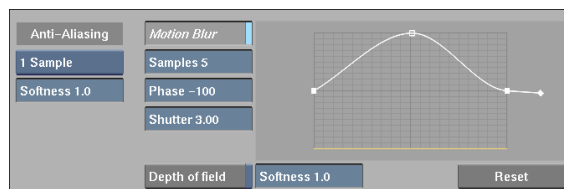
The original animation with Motion Blur disabled.



Motion Blur is enabled using a custom motion blur curve and phase set to 100.

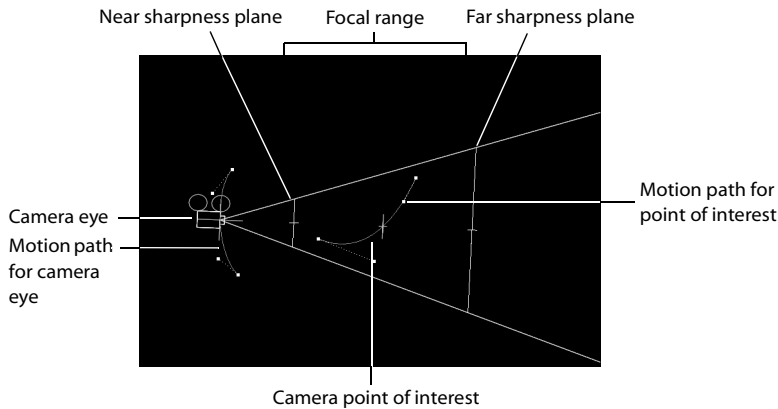


Motion Blur is enabled using the same motion blur curve as above. Phase is set to -100.



Working with Depth of Field

Enable the Depth of Field button to blur objects that are outside the camera's focal range. Objects are blurred on either side of the point of interest. Objects are blurred even more when they are outside the focal range, before the near sharpness plane, or after the far sharpness plane.



Both the physical and normal camera use depth of field, but the physical camera clearly indicates its focal range when in Top, Side, or Front view. With the physical camera, you can change the field of view by **ALT**-dragging the near sharpness plane. This changes the camera only; what you are viewing remains the same size. See “Using the Camera” on page 945.

You can change the amount of blur using the Depth of Field Softness field in the secondary Setup menu.

Action: Layers and Surfaces

Layers galore!

You use surfaces to display layers in the scene and then composite front and matte clips with a common back clip.

Summary

In this chapter, you learn about:

- “Working with Layers” on page 908
- “Using Layers on Surfaces” on page 917
- “Reordering Overlapping Surfaces” on page 928
- “Using Source Nodes” on page 931

About Layers and Surfaces

You should have a working understanding of Action layers and the relationship between layers and surfaces to be successful in using Action:

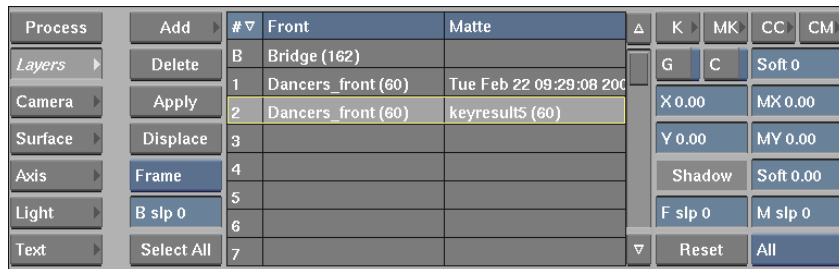
- Each front and matte clip combination that you load from the desktop is called a *layer*.
- You must apply a layer to a surface so that a layer’s clips are added to the scene.
- The clips that you load into Action are listed in the Layers list.
- When you first open Action, you load the front clip and matte clip for the first layer and a common back clip for all layers.

A surface is used to place a layer in the scene. To use a layer that you load from the desktop reels, you must add a surface to the scene for that layer. A surface has the following characteristics:

- A surface is a representation of a layer, which you add to the scene.
- A surface type can be image, bilinear, bicubic, or extended bicubic.
- The same layer can be applied to multiple surfaces. Any cropping, blurring, or recolouring that you apply to a layer is applied to all the surfaces for that layer. For example, if you blur a layer, all of the surfaces using that layer are blurred.

Working with Layers

When you open Action, you load the front and matte clips for the first layer. You can then load as many additional layers as you want from Action. You use the Layers menu in Action to modify the front and matte clips contained in a layer.



The controls in the Layers menu are described in the following table.

Select:	To:
Add	<p>Add more layers by selecting front and matte clips from the desktop reels.</p> <p>You can select multiple layers and click Add to add as many front and matte clips as you have selected layers in the Layers list. Press CTRL and click to select multiple layers.</p>
Delete	<p>Delete a layer from the Layers list.</p> <p>When you Delete a layer you are prompted to confirm its deletion. The Undo operation does not undo a deleted layer.</p>
Apply	Apply a layer to a surface. See “Applying Layers to Surfaces” on page 919.
Displace	Specify the layer to use as the displacement source when you use displacement mapping to create a 3D object from a 2D surface.
Layer Rendering box	Select Field or Frame rendering for each layer. By default, this is the same as the global rendering you set in the Action menu.
Back Slip, Front Slip and Matte Slip fields	<p>Slip the front, matte, or back clip.</p> <p>A positive value begins the clip at the specified frame. A negative value creates a freeze frame effect where the first frame of the clip is held by the specified number of frames.</p> <p>Press ALT while setting either the Front or Matte slip value to set both fields to the same value.</p>
Select All	<p>Select all layers in the Layers list. When you select multiple layers, you can delete and reset multiple layers.</p> <p>Press CTRL and click layers to select a more specific set of layers.</p> <p>You can select multiple layers and clips On or Off. When you select multiple layers, access to the Keyer and Colour Corrector is disabled.</p>

Select:	To:
K button	Load the back clip and the front and matte clips for the selected layer into the Keyer.
MK button	Load the back clip, front clip and matte clips for the selected layer into the Modular Keyer.
CC button	Load the front clip for the selected layer into the Colour Corrector.
CM button	Load the matte clip for the selected layer into the Colour Corrector.
G button	Use Gaussian blur on a selected layer.
C button	Crop a front clip or a matte clip for a selected layer.
Shadow softness	Adjust the softness of a shadow. If the scene contains more than one shadow using the selected layer, all shadows are softened by the same amount you specify in this field. See “Adding Shadows” on page 978.
Reset button and box	Reset layer properties. Use the Reset box to select the properties you want to reset: All, Node, Blur, Crop, Frame, Slip Back, Slip, CC Matte, CC Front, CC Back, and Keyer.

Using the Layers List

The clips that you load for each layer appear in the Layers list. You can display a layer’s clips individually in the image window using the Front and Matte buttons located at the very left of the Action menu. The back clip you select while opening Action appears in the first row of the Layers list and is assigned the letter B in the # column. The second row in the Layer list contains the front and matte clips you selected while opening Action. The length of each clip appears in parentheses beside the clip name.

You can also display a layer’s clips by using the Scene View box.

You can sort the Layers list by list number, front clip name, or matte clip name. As you view a layer’s clips, you can look at the original front, back, or matte clip individually as well as the results of colour correcting and keying matte.

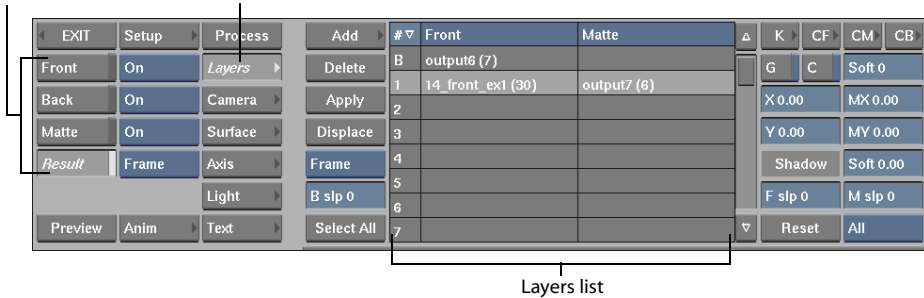
To view a layer's clips:

1. In the Action menu, click Layer.

The Layer menu appears.

Click any of these buttons to view a specific clip for a layer

Layers button



2. To display the front clip in the image window, click Front. To display the matte clip, click Matte.

You can also view the back clip by clicking Back.

3. Click Result to view the overall effect.

Changing a Layer's Clips

You may decide to change the front and matte clips that make up a layer. You can easily replace a layer with any set of front and matte clips from the desktop reels.

To change a layer:

1. Select the layer that you want to change.
2. Click Add.

The desktop reels appear.

3. Select a front clip and a matte clip.

You return to the Layers menu. The names of the new clips replace those of the old clips in the Layers list. The new clips are applied to all the layer's surfaces. See "Working with Layers" on page 908.

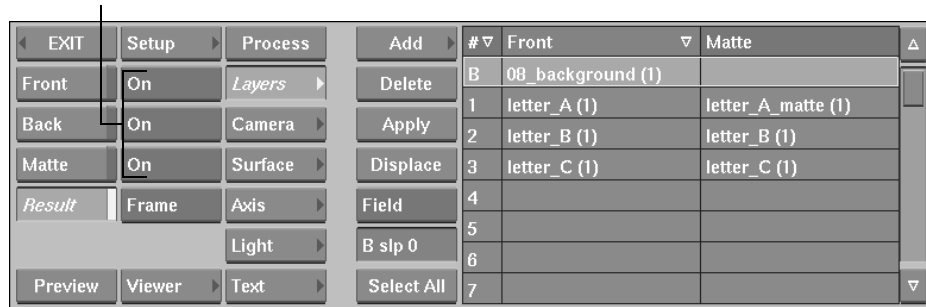
To change the back clip:

1. Select the B layer.
2. Click Add.
The desktop reels appear.
3. Select a clip for the back clip.

Turning Clips On and Off

As you work with layers, you can set properties for either the front, matte, or back clip. To view a clip exclusively, you can turn off the other two clips using the Off option in the Front, Back, or Matte Clip box.

Front Clip, Back Clip, and Matte Clip boxes



To enable a clip, select On from the Clip box. To disable a clip, select Off. You may want to disable a clip in the following situations:

- Disable the back clip to use a solid black background in the animation.
- Disable the matte clip to work with the full-front image. To work with a cutout of the front image, make sure that the matte clip is enabled.

You can also invert the matte clip by selecting Invert in the Matte Clip box. To invert multiple matte clips, **CTRL-click** the matte layers and select Invert in the Matte Clip box.

Locking the Front or Back Clip

Normally when you use any of the image window controls to move through the front or back clip, each frame of the clip is loaded and appears in sequence in the image window. You can change this default setting so that the frame displayed is not updated as you move through the clip, but remains locked at a specific frame.

You lock the back clip in the same way as the front clip, except that you select Lock from the Back Clip box. The matte clip is locked with the front clip by default. However, you can slip the matte so that it is locked at a different frame. Use the Mslp field to set the slip value for the selected matte clip.

To lock the front or matte clip:

1. Click the Front or Matte button to display the clip in the image window (optional).
2. In the timeline, choose the frame at which you want to lock the clip.
This frame becomes the current frame for the clip.
3. In the Front or Matte box, select Lock.
4. Click Result to view the resulting clip in the image window.
The front clip remains locked to the current frame.

Blurring a Clip

Use the X and Y fields to blur the front clip and matte clip for the selected layer. You cannot blur the back clip. The X field controls the amount of blurring on the horizontal (X) axis, and the Y field controls the amount of blurring on the vertical (Y) axis.

NOTE: If you apply the layer to more than one surface, all of these surfaces are blurred.



To blur only the matte clip, use the MX and MY fields. By blurring only the matte clip, you get a softer edge on the surface of the front clip. The MX field controls the amount of blurring on the horizontal (X) axis and the MY field controls the amount of blurring on the vertical (Y) axis.

To blur only the front clip, set the blur using the X and Y values, and then negate the values for the MX and MY fields. For example, to blur only the front clip 10 pixels on both axes, set the X and Y fields to 10 and the MX and MY fields to -10.

Using Gaussian Blur

You can use a Gaussian (G enabled) or a box blur (G disabled). The following is a list of advantages and disadvantages of using both blurs:

- Gaussian blur has rounded, smoother edges, but takes longer to process than a box blur.
- Box blur has rectangular, rougher edges.
- Gaussian blur is better for animating and it can be blurred on a subpixel level (0.00).

To blur a clip:

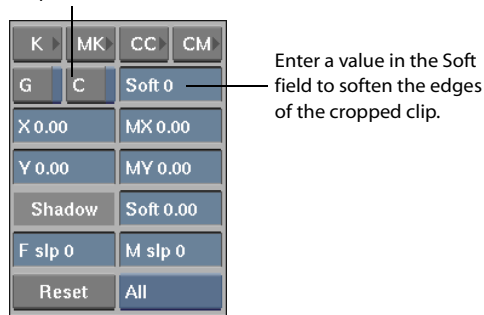
1. Select the layer.
2. If you want to use a Gaussian blur, enable the G button. Disable the G button to use a box blur.
3. Enter the amount of blur in the X and Y fields or the MX and MY fields.

When a box blur or Gaussian blur has been applied to a clip, “(A)” appears next to the clip name in the Layers list.

Cropping a Clip

You can crop a layer using the Crop button. You use the Crop button in the Layers menu to crop the entire layer. Front and matte clip are cropped together. You cannot crop the back clip.

Crop button



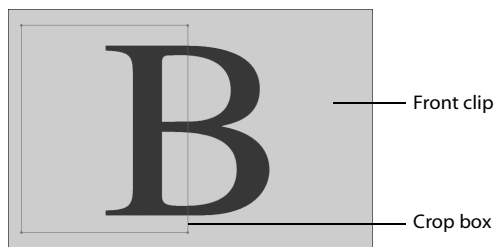
NOTE: If you applied the layer to more than one surface, all the surfaces are cropped.

You can animate a crop by changing the size and shape of crop box size at different keyframes. You can also animate the softness of the crop box. See Chapter 9, “Animation.”

To crop a clip:

1. In the Layers list, select the layer containing the clip you want to crop.
2. Click C.
3. To the left of the Layers menu, click Front or Matte.

The clip appears in the image window with a red outline on image, which indicates crop box.



HINT: You can use two views to view the crop interactively on both the front and the matte clips. See “Displaying Multiple Views Simultaneously” on page 950.

4. Set the corners of the crop box by dragging the four corners of the red outline.
5. Click Result.

The cropped layer appears in the image window, and the front and matte clip share the same crop.

Deleting Layers

In Action, you can delete layers that are applied to surfaces. When you delete a layer, the surface becomes invisible and the layer is empty. You can then apply another layer to the surface, add a new front and matte clip layer, or delete the surface.

The system prompts you to confirm the deletion operation. Click anywhere in the menu to cancel the delete operation. You cannot undo a deleted layer operation.

To delete a layer:

1. In the Layers list, select the layer that you want to delete.
Hold **CTRL** and click to select multiple layers. Alternatively, hold **SHIFT** and click to select a consecutive range of layers.
2. Click Delete.

Click Delete to remove the selected layer

Add ▾	# ▾	Front	Matte	Δ	K ▾	MK	CC	CM ▾
Delete	B	Bridge (162)			G	C	Soft 0	
Apply	1	Dancers_front (60)	Tue Feb 22 09:29:08 200		X 0.00		MX 0.00	
Displace	2	Dancers_front (60)	keyresult5 (60)		Y 0.00		MY 0.00	
Frame	3				Shadow		Soft 0.00	
B slp 0	4				F slp 0		M slp 0	
	5							
	6							
Select All	7			▽	Reset		All	

Click Select All to delete all layers

The selected layer is deleted.

Using Colour Correction and the Keyer in Action

You can access the Keyer and the Colour Corrector directly from Action. You can key any layer or colour correct any front, matte, or back clip without having to exit to the desktop.

When you enter the Colour Corrector or Keyer from Action, you can view the result of your colour correction or key as it would appear in Action using the Context button. The Context view is interactive: as you make changes, the Action result is updated in the image window.

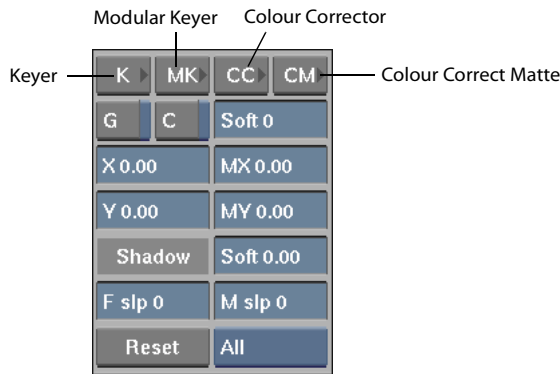
As you use more layers in Action, interactivity in the Colour Corrector and Keyer may be slower than when you access the modules exclusively from the desktop. However, accessing these modules through Action is more efficient for checking your changes against the other Action layers.

HINT: Use Result view while keying or colour correcting. In this way, the number of Action layers has no impact on system performance. You can tweak the keying and colour correction setup using Context view.

While you work in the Colour Corrector or Keyer, you do not have to click Process because the modifications are automatically applied to the layer in Action. If you enter the Colour Corrector from Action, scrub the timeline to view the Result.

To access other modules from Action:

1. Select the layer containing the clip you want to colour correct or key.
2. Click the corresponding module button.



3. As you work in the module, click Context to preview the result.
4. Click Exit to return to Action.

The colour-corrected clip appears with a “(C)” in the Layers list, and a keyed layer appears with a “(K)”.

Using the Modular Keyer in Action

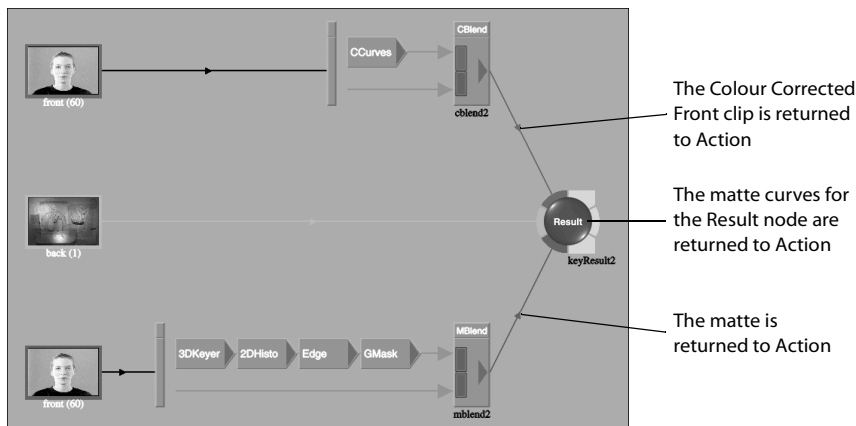
There are some minor differences and things to keep in mind when you access the Modular Keyer from Action instead of from the desktop. This section describes features specific to using the Modular Keyer in Action. For complete information on using the Modular Keyer, see Chapter 34, “The Modular Keyer.”

When you load a layer into the Modular Keyer, the front and matte clips for the selected layer are loaded with the back clip and appear at the beginning of the processing pipeline. The front clip is used as the front clip and the matte clip is used as the Key In clip for the processing pipeline.

The back clip is used as the back clip in the processing pipeline and is loaded as a reference only. Any modifications you make to the back clip in the Modular Keyer are not used when you return to Action.

To view the results of your Modular Keyer work as it would appear in Action, select Action from the Alt View box.

The following example shows a layer loaded into the Modular Keyer from Action.



All branches in the processing pipeline are connected to the Result node. When you return to Action from the Modular Keyer, the following information is used:

- The colour corrected front clip. (Attached to the red tab of the Result node.)
- The matte clip. (Attached to the blue tab of the Result node).
- The matte curves for the Result node. (To access the matte curves, click the Result node).

NOTE: The back clip is for reference only. Any modifications you make to the back clip, such as colour correction, are not applied in Action.

Using Layers on Surfaces

When you apply a layer to a surface, you either have to add a new surface or apply a layer to an existing surface. You can then add displacement and lighting effects, or change the surface's properties such as its shape, transparency, and specular highlight.

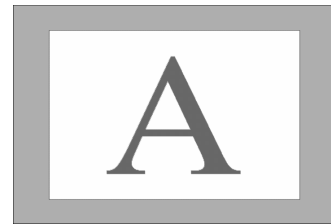
You control the position of the surface using axis, rotation, scale, shear, and other attributes related to its placement. See “Adding Objects to the Scene” on page 940.

Action provides you with the choice of different surface types.

Image

An image surface is flat. You cannot change its shape in the same way as bilinear and bicubic surfaces because an image does not have vertices. You can, however, scale and shear an image using its axis. See “Manipulating an Axis” on page 941.

This is the simplest surface. It is added to the scene automatically the first time you enter Action with a Front, Back, and Matte.

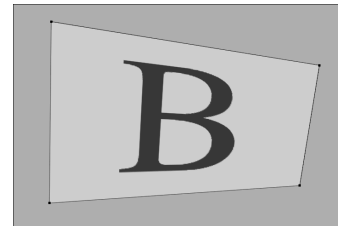


Flat surface

Bilinear

A bilinear surface has four vertices: one for each corner. The vertices are joined using linear interpolation (straight lines).

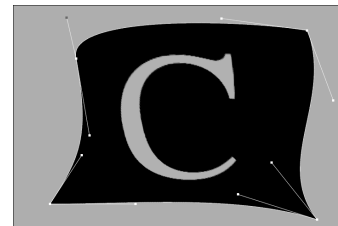
You can animate the shape of a bilinear surface by changing the position of the corners. See “Changing the Shape of a Surface” on page 925.



Bilinear Surface

Bicubic

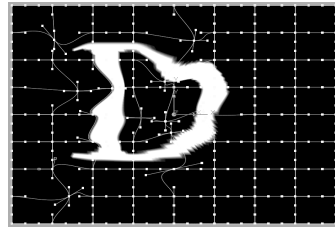
A bicubic surface has four vertices: one for each corner. The vertices are joined using bicubic interpolation (curved lines). Each corner has two additional tangent handles used to adjust the curve of the line between points. You can animate the shape of a bicubic surface by changing the position of the corners and moving the tangent handles to adjust the curve between corners. See “Changing the Shape of a Surface” on page 925.



Bicubic Surface

Extended Bicubic

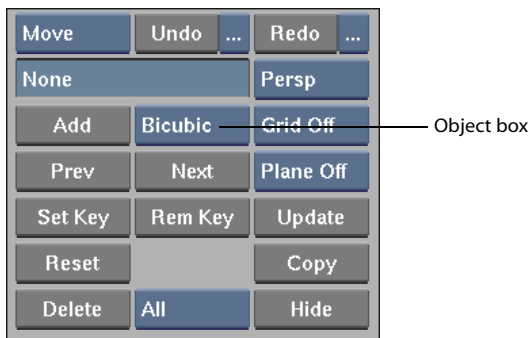
An extended bicubic surface has four vertices like a bicubic surface, but the sections of the extended bicubic surface can be subdivided to increase the number of vertices. The vertices are joined using bicubic interpolation (curved lines). You can animate the shape of a bicubic surface by changing the position of the corners and moving the tangent handles to adjust the curve between corners. See “Extended Bicubics” on page 1002.



Extended Bicubic Surface

To add a surface:

1. In the Layers list, select the layer containing the front and matte that you want to add to the scene.
2. In the Object box, select a surface type.



3. Click Add.

The surface is added to the scene with its own axis. The selected layer in the Layers list is automatically applied to the surface. For example, the following figure shows what happens

when an image surface is added to the scene with Layer1 selected. The contents of Layer1 are also shown.

00025

A

25

00024

A

25

00023

A

25

00022

A

25

00021

A

25

000

25

Front clip stored in Layer1

00025

A

25

00024

A

25

00023

A

25

00022

A

25

00021

A

25

000

25



When the surface is added to the scene, Layer1 is applied to the surface. This creates a link between the layer and the surface.

Notice that when Layer1 is applied to image surface, the front and matte clip are combined. You can turn off the matte to show the entire front clip. See “Turning Clips On and Off” on page 911.

NOTE: The first time you open Action during a session, an image surface is added to the scene using the first layer. You must add a surface for each additional layer before its clips can be used in the scene.

Applying Layers to Surfaces

You must select a surface before you can apply a layer. You cannot apply the back clip to a surface. The back clip is used as the background in Action. You use the Apply button to change the layer used by a surface. This lets you apply a new layer to an existing surface.

To apply a layer to a surface:

1. Select a surface by clicking one of the surface edges in the scene or changing to Schematic view and clicking the surface icon.
2. In the Layers list, select the layer to which you want to apply a surface.

3. Click Apply.

Apply button

Add ▸	# ▾	Front	Matte
Delete	B	output6 (7)	
Apply	1	14_front_ex1 (30)	output7 (6)
Displace	2		
Frame	3		
B slp 0	4		
Select All	5		
	6		
	7		

The new layer replaces the layer for the selected surface.

To add multiple surfaces at a time:

1. In the Layers list, select the layers for which you want to apply the same type of surface.
2. In the Object box, select a surface.

Add button

Move	Undo ...	Redo ...
None		Persp
Add	Bicubic	Grid Off
Prev	Next	Plane Off
Set Key	Rem Key	Update
Reset		Copy
Delete	All	Hide

Object box

3. Click Add.

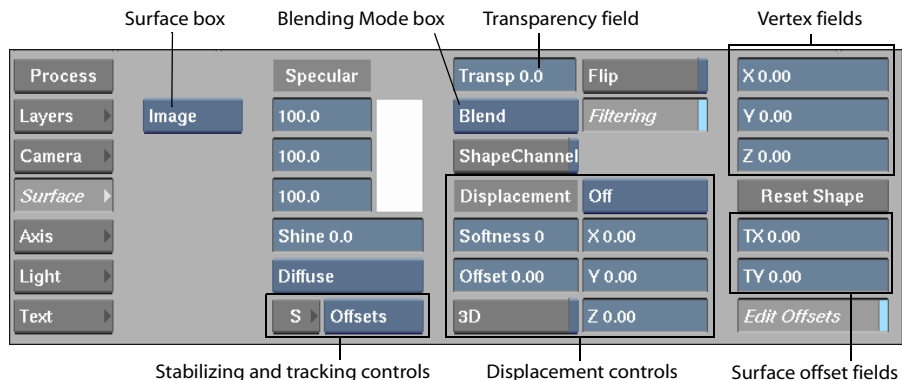
The layers appear in the scene.

Working with Surfaces

As you work with a surface you can change its shape, position, or transparency, as well as apply lighting effects. You can also apply four-point tracking data to a bilinear surface. The Surface menu includes many submenus that you access using the Surface box. You use the Image submenu to adjust the shape, position, and material properties of a surface, as well as to turn on ambient lighting and use displacement mapping.

To modify surface image properties:

1. In Action, click Surface.



2. In the Surface menu, select Image from the Surface box.

Surface Image controls are described in the following table.

Select:	To:
Specular highlight fields	Set highlights of a surface. Click colour pot to select the colour of the surface highlights. See "Adjusting Specular Highlights" on page 975.
Shine field	Add a shine to specular highlights. There are no specular highlights when Shine is set to 0.
Lighting box	Set Ambient or Diffuse lighting so that surface can reflect incidental light. See "Applying Ambient and Diffuse Lighting" on page 976.
Stabilizing and Tracking controls	Apply stabilizing data to a surface or surface offsets. See "Tracking and Stabilizing" on page 789.
Transparency field	Set surface transparency. See "Changing Surface Transparency" on page 922.
Blend Mode box	Set surface blending mode. See "Using Surface Blending Modes" on page 922.
Shape Channel button	Specify whether you want to use the Shape channel or Surface vertex channels in the Channel Editor. This button appears when a bilinear, bicubic, or extended bicubic is selected. See "Using the Shape Channel or Vertex Channels" on page 926.

Select:	To:
Displacement controls	Create a 3D model using a 2D surface. See “Displacement Mapping” on page 999.
Flip button	Flip the selected surface normals. This is used to light the back side of a surface, and has no incidence on the layer or the orientation of the image that is displayed by the surface.
Filtering button	Enable or disable bilinear filtering. See “Filtering” on page 924.
Vertex position fields	Verify the coordinates of the currently selected vertex or tangent. You can modify these values to alter the shape of a surface. See “Changing the Shape of a Surface” on page 925.
Reset Shape button	Reset surface shape. See “Resetting the Surface Shape” on page 926.
Surface offset fields	Offset a surface along X-axis or Y-axis. See “Offsetting a Surface” on page 924.
Edit Offsets button	Edit tracker offsets. See “Editing Tracker Offsets” on page 925.

Changing Surface Transparency

By adjusting the transparency of the surface, you can make it fade in or out of view, or simulate a transparent material such as glass. When the Transp field is set to 100, the surface is completely transparent. When the value is set to 0, there is no transparency in the surface.

Using Surface Blending Modes

Use the Blend Mode box to select how the front and matte clips or front and back clips are combined. For example, use Screen for blending fire or a lightning bolt that was originally shot on a black background and you want to composite the fire or lightning with a different background.

As another example, use Simple Add to add the values of a surface to whatever is behind it while maintaining control over surface transparency. Basically, experiment with these blending modes to create your own unique effect. The surface blending modes are described in the following table.

Select:	To:
Add	Compensate for the soft or anti-aliased edge on an object in a front and matte clip layer. This is useful for 3D imagery and whose front is rendered over a black background.
Blend	Punch the matte through the front. This blends the edge of the front clip and adds additional softness to the layer. This is the default setting.

Select:	To:
Max	Compare the RGB channels of each pixel of the front clip and the back clip individually and return the larger of the two values.
Min	Compare the RGB channels of each pixel of the front clip and the back clip individually and return the smaller of the two values.
Multiply	Multiply the RGB channel values of corresponding pixels of the front clip and the matte clip, and normalize the result by dividing by 255 in 8-bit mode, or 4095 in 12-bit mode. The resulting RGB channel values are assigned to corresponding pixels in the generated clip. Hardware limitations prevent transparency from working properly with Multiply.
Negate	Create a negative-like result. Try Negate with a soft white matte with a white front clip. This mode does not use the matte or transparency in its blend.
Screen	Multiply the inverse of the matte clip's colours with the colours of the front clip. The resulting colour is always a lighter colour. Screen does not work with transparency.
Simple Add	Punch the matte through the front using more softness than Blend mode. Simple Add mode is similar to Add but includes transparency.
Spotlight	Create a slide projector effect. This mode does not use the matte or transparency and decreases system performance.
Spotlight Blend	Create an effect where black areas of the front disappear. This mode does not use the matte or transparency; it works well with a clip on which an object is surrounded by black.
Subtract	Subtract the RGB channel values of the pixels from the matte clip from the RGB channel values of the pixels from the front clip and assign the result to the RGB channel values of the pixel in the result. If an RGB channel value from the matte clip is larger than the corresponding channel value in the front clip, yielding a negative result, that result is clamped at 0 (black).

NOTE: When using Blending modes, surface transparency varies according to the selected mode. You can still create a transparent surface, but you may have to access the Keyer and alter the back matte keyer curve or change the Diffuse r,g,b channels to 0,0,0. See “Using Colour Correction and the Keyer in Action” on page 915.

Offsetting a Surface

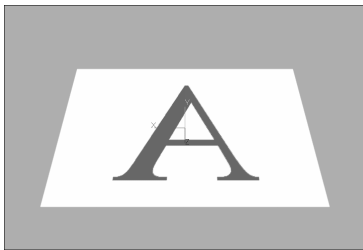
You use the TX and TY fields to offset a surface along the X-axis or Y-axis. All rotations, scaling, and shearing applied to a surface are applied about its axis. The location of the axis is indicated by the axis icon in the scene. See “Manipulating an Axis” on page 941.

By default, a surface’s axis is at the centre of the surface. However, you may want to apply transformations about the centre and left-most point of the surface for example. To do this, you can offset the surface using the TX and TY fields.

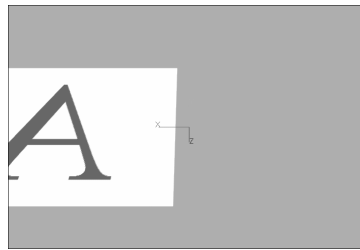
To offset a surface:

1. Select the surface you want to offset.
2. In the Surface box, select Image.
3. Change the TX and TY fields accordingly.

The following figure illustrates the difference between rotating a surface that has not been offset and a surface offset using the TX field.



Rotated image with no offset. Notice that the axis is in the centre of the image.



Rotated image with an offset along the X-axis. Notice that the axis is offset from the image centre.

Filtering

Use the Filtering button to enable or disable bilinear filtering of the image. If you disable Filtering and zoom in on the image, you may notice that some edges are very hard. When Filtering is enabled, a smooth filter is applied to the image.

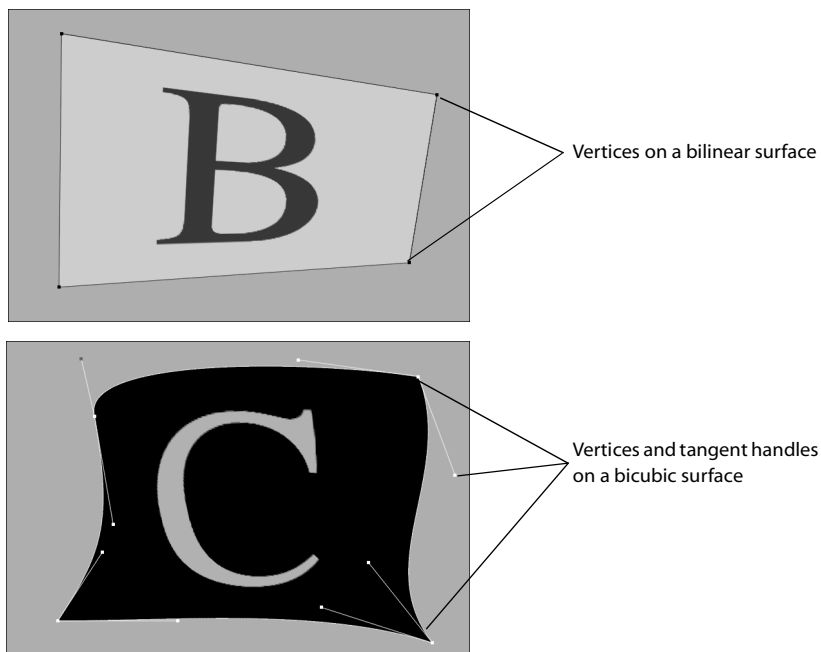
Since filtering may soften an image, sometimes you will want to turn off filtering if you are doing straight front, back, matte compositing, and when no scaling or animation is required. In this way, turning filtering off ensures you maintain as much of the original picture as possible. However, with filtering off, no subpixel interpolation is possible, and if you are animating the movement of an image, it may appear “steppy.”

Editing Tracker Offsets

Enable the Edit Offsets button to change the way the image is applied to the surface. When you enable this button, an offset point appears for each of the surface's four corners, and you edit the offsets to match the corners of your square. View the result by disabling Edit Offsets. Editing tracker offsets only works with bilinear surfaces. See “Tracking and Stabilizing” on page 789.

Changing the Shape of a Surface

You use the vertices—or handles—on bilinear and bicubic surfaces to change a surface's shape. While, you cannot change the shape of an image surface, you can move, rotate, shear, and scale a surface using the Axis menu. See “Manipulating an Axis” on page 941.



To move a handle on a bilinear or bicubic surface, click the handle that you want to edit. The selected handle appears in red. Use the cursor to drag the handle to its new position.

Each corner has two tangent handles. Lengthen or move the tangents to change the shape of the bicubic between corners.

Resetting the Surface Shape

To reset the surface handles to their default position, click the Reset Shape button in the Surface Image submenu. The handles are reset in the current frame only, and a shape key is added at the current frame if Auto Key is enabled.

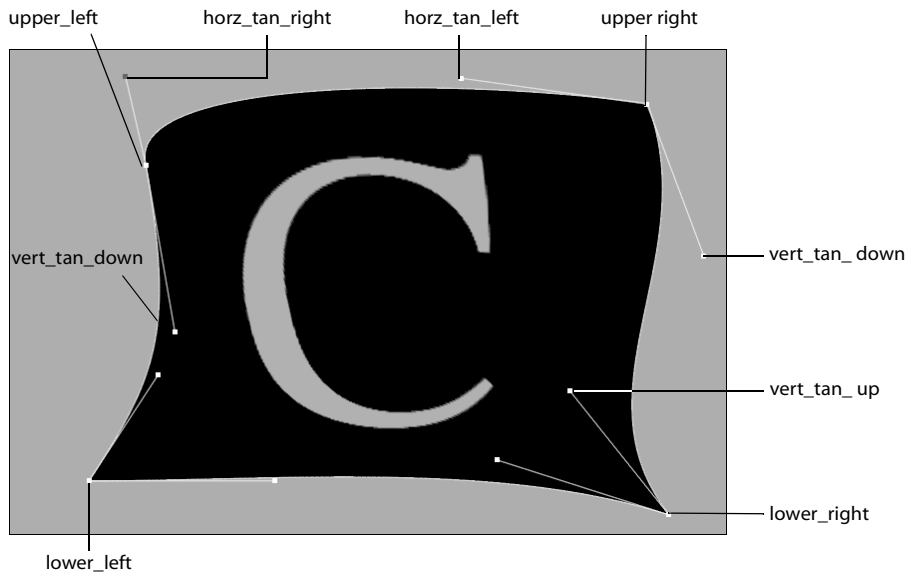


Using the Shape Channel or Vertex Channels

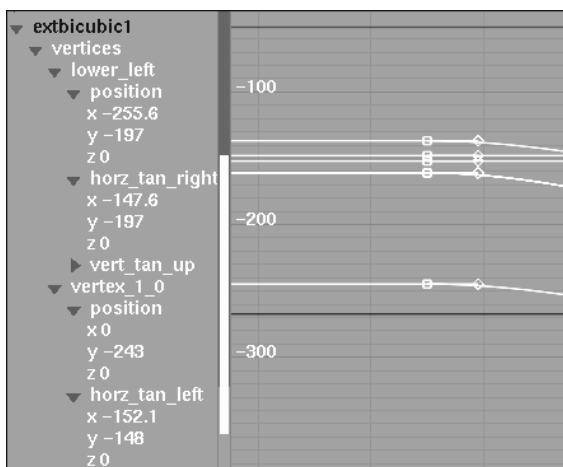
In the Channel Editor, you can select whether you want to create animation for bilinear, bicubic or extended bicubic surfaces using the Shape channel or individual Vertex channels. The Shape channel shows when the shape of the surface changes during the animation. Each time you move a surface handle, a shape key is added at the current frame provided Auto Key is enabled in the Setup menu.

Vertex channels correspond to the vertices—or handles—that appear on the four corners of bilinear, bicubic, and extended bicubic surfaces. The channel names for each corner are: upper_left, upper_right, lower_left, and lower_right, and are listed in the channel hierarchy each with a position x,y,z.

Each corner vertex has two tangent handles—the tangent handles for upper_left are named horz_tan_right and vert_tan_down.



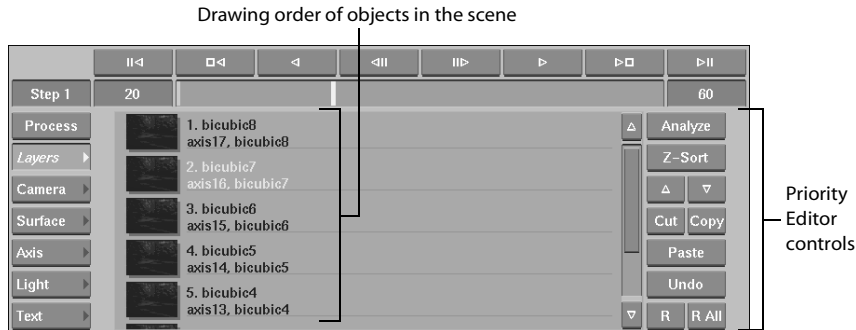
Extended bicubics have additional vertex channels that appear in the Channel Editor for subdivided vertex channels. These channels appear only when you create a keyframe for the channel and its value changes. These vertex channels are named vertex_0.1, vertex_1.0, vertex_2.1 and so on, according to their position on the surface. Click the vertex on the surface to highlight its channel in the channel hierarchy. See “Extended Bicubics” on page 1002.



Reordering Overlapping Surfaces

When you add a surface (image, bilinear, or bicubic) or geometry (3D text or model), the surface or geometry appears in front of all other objects in the scene. These overlapping objects create a stack and an order of priority as one object is drawn in front of the other.

You can change the drawing order of surfaces, 3D models, and other objects using the Priority Editor. Swipe the bar at the bottom of the Layers menu to access the Priority Editor.



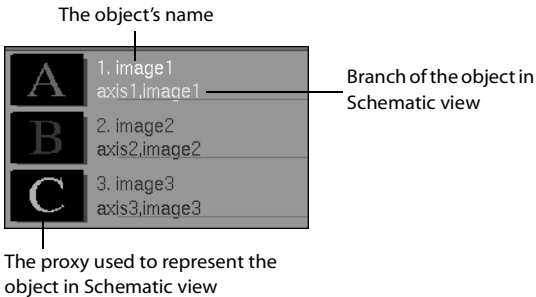
The objects in the scene are shown in their drawing order: highest priority at the top of the menu to the lowest priority at the bottom. The lowest level object is drawn first, the second to lowest level object is drawn next, and so on until the top of the list.

To change the drawing order of objects:

1. In the Priority Editor, select the object.
The selected object name is highlighted in yellow.
2. Click the Down arrow to move the selected object one position lower in the stack, or click the Up arrow to move the selected object one position higher in the stack.
When you move an object in the stack, a keyframe is added to the timeline.
3. Click Preview to ensure that the drawing order is correct.
Occasionally, a matte may obscure another object if the drawing order is incorrect. If this happens, move the surface up and down in the stack until the drawing order is corrected.

Using the Priority Editor

Each entry in the Priority Editor indicates the object's name, the order of the object in the stack, and the location of the object in its branch. There is also a proxy showing what the object looks like in Schematic view.



Use the Priority Editor controls to analyze the scene for changes in rendering priority, to change the rendering priority, and to cut, copy, and paste priority information from one frame to another. The Priority Editor controls are described in the following table.

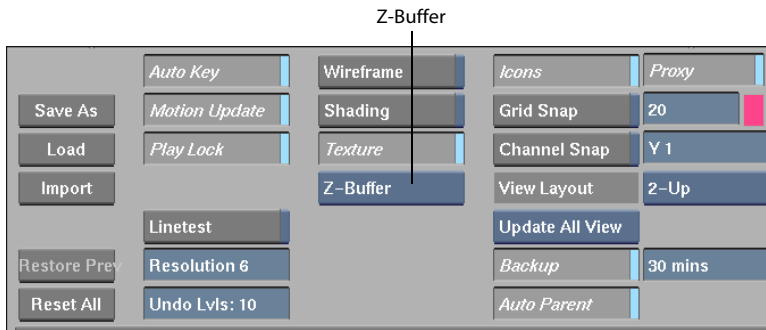
Select:	To:
Analyze and Z-Sort buttons	Analyze the entire scene (Analyze) or the current frame (Z-Sort). A mark appears in the timeline if there is a change in drawing order. See "Using Analyze, Z-Sort, and the Z-Buffer" on page 930.
Up and Down Arrows	Change the priority of the selected object. Moving an object in the stack, place a mark in the timeline to indicate a change in priority. The Down arrow moves the selected object one position lower in the stack, behind the next lower object. The Up arrow moves the selected object one position higher in the stack, in front of the next higher object.
Cut, Copy, and Paste buttons	Cut, copy, and paste priority information between frames. Note that a mark indicates when a change in priority occurs. If you copy and paste a mark that does not change the priority, no mark appears on timeline.
Undo	Undo the last operation. You can undo Cut, Copy, Paste, Reset, and Reset All, but <i>not</i> Z-Sort and Analyze. Use the Undo List to view a list of recent operations and revert to a prior state. Select an item in the list to return to that state. All actions that occurred after the selected item are undone.
Reset and Reset All buttons	Reset priority information for the current frame or for all frames. The timeline is reset to the original priority before any changes were made.

Using Analyze, Z-Sort, and the Z-Buffer

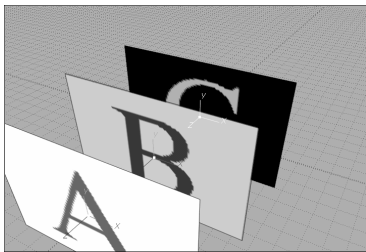
When you use Analyze or Z-Sort, the objects in the scene are compared using their position in the scene and not the individual pixels of a shape or model. The Priority Editor uses the value of the distance between the camera and the nearest and farthest points on the objects in the scene to determine the sorting order.

Because the Priority Editor is axis based and not pixel based, you cannot properly order objects that intersect because of rotation, scale, or shear. The Z-buffer uses the Z-value of each pixel for the sorting order.

The Priority Editor is used in conjunction with the Z-buffer/Shadow Mix box in the Setup menu.



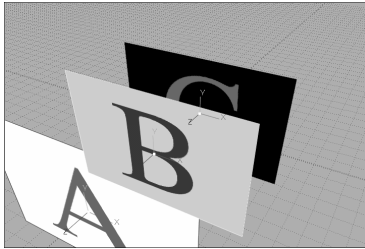
When Z-buffer is on, objects are arranged according to their distance from the camera eye. Since the camera is pointed towards the Z-axis by default, objects are arranged according to their location on the Z-axis. For example, Image A is at Z position 200, Image B is at Z position 100, and Image C is at -50. When Z-buffer is on, Action draws these images as shown in the following figure.



If you move the camera, objects are sorted according to the axis the camera is pointed towards.

When Z-buffer is off, the distance of objects from the camera eye is not considered. Objects are drawn in the order shown in the Priority Editor. To follow the previous example, you can turn Z-

buffer off and change the priority of Image B so that it is drawn on top of both Image A and Image C. The Z position of these images is therefore ignored. You cannot do this when Z-buffer is on.

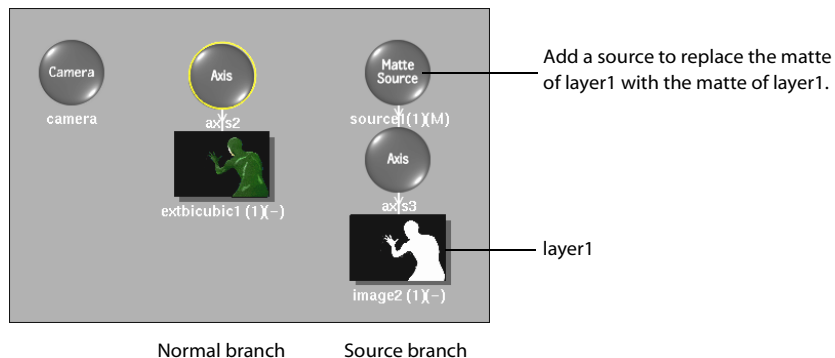


Shadow Mix, like Z-buffer, arranges objects according to their distance from the camera eye. Use Shadow Mix so that each shadow is rendered in the correct Z order with its corresponding surface.

Using Source Nodes

Use source nodes to separate front and matte clips in a layer and then apply separate transformations to the front and matte clips. For example, if you apply a layer to a surface and you want to create an effect where the matte moves into the scene, you add a source that isolates the matte clip. A matte source lets you animate the matte's position separately from the front. You can also use source nodes to apply several matte clips to a single front clip.

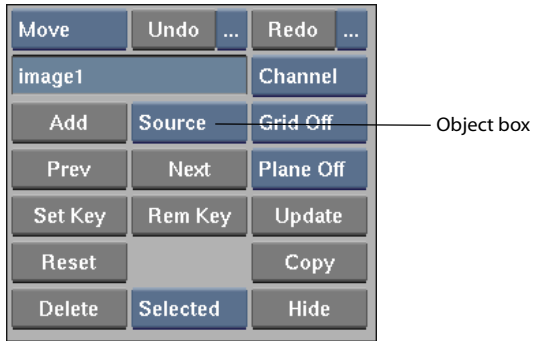
You create a matte source to add transformations to a layer's matte, and create a front source to do the same for the front clip. When you create a matte source, a new branch appears in the Schematic view showing the matte clip is separated from its layer, ultimately replacing the matte with itself. This method of using source nodes lets you animate a layer's matte separately.



To view the contents of the Source node in the image window, select the Source node and use Result view. These objects only add transformations to the clips in the Layers menu. For example, in the preceding figure, axis2 and image2 only affect the matte of layer1.

To create matte source:

1. In the Layers list, select a layer.
2. In the Object box, select Source and click Add.



A source is created in Schematic view that parents an axis and a surface.

3. Select the axis or surface parented by the source and create the additional animation.
The changes applied to the axis or surface connected to a source are applied to the layer's matte.

For example, if you change the axis's scale to 80%, then the layer's matte is scaled 80% when used with the layer's front. In addition, if you replace the surface by a bilinear or bicubic, you can create complex animations where the matte behaves like a page turn leading into the scene.

4. Select the source in Schematic view and switch to Perspective view to view the source's result.

When a source is selected, only its result is shown in the image window.

5. Select an axis or surface in a normal schematic to view the result of how changing the matte affects the scene.

Transforming the Front Clip

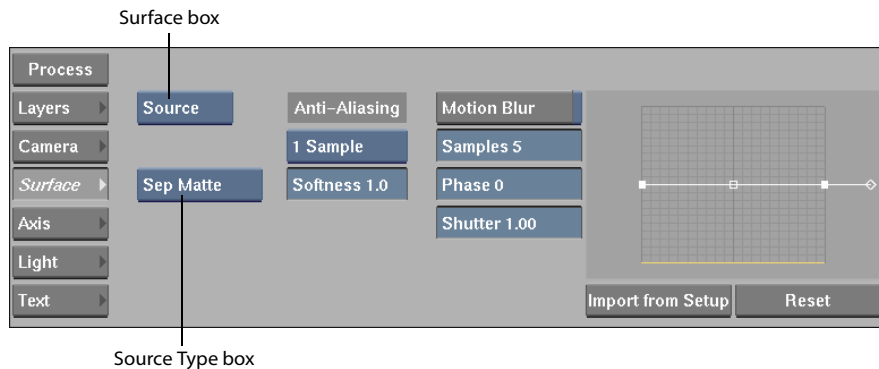
If you want to add transformations to a layer's front clip, you must create a special branch that begins with a Front source.

To add transformations to a layer's front:

1. Select the layer and click Surface.
2. In the Surface box, select Source and click Add.

A source is created in Schematic view that parents an axis and a surface. By default a matte source that separates out the matte clip is automatically added to the scene. You must change this to a front source to separate the front clip.

3. With the source selected, in the Surface menu, select Source from the Surface box.



4. From the Source Type box, select Sep Front.
The source is switched to separate the front clip. Use the Source menu to switch the selected source and separate a layer's front, a layer's matte, or create a custom source. See "Replacing the Front or Matte Clip" on page 934.
5. Select the axis or surface parented by the source and create the additional animation.
The changes applied to the axis or surface connected to a source are applied to the layer's front.
6. Select the source in Schematic view and switch to Perspective view to view its result.
When a source is selected, only its result is shown in the image window.
7. In the Schematic view, select an axis or surface outside the Source branch and return to Perspective view again to see how changing the front affects the scene.

Replacing the Front or Matte Clip

With source nodes, you can also replace a layer's front and matte clips with the front, matte, or a combination of the front and matte clips from another layer.

To replace a layer's front or matte:

1. Select the layer.
2. Select Source and click Add.
A source node that parents an axis and image is added to Schematic view.
3. Select the source node in Schematic view.
4. In the Surface menu, select Source.



5. Select the type of source in the Source Type box.

Select:	To:
Sep Front	Set the source to replace the front clip of the selected layer with the front clip from another layer. If you select this type of source, go to step 8.
Sep Matte	Set the source to replace the matte clip of selected layer with the matte clip from another layer. If you select this type of source, go to step 8.
Custom	Create a custom source. Use this to combine the front and matte clips of a layer, to replace a front clip with a matte clip, or other combinations. Steps 6 and 7 include instructions on using custom source nodes.

6. If you select Custom as the Source Type in the last step, select the clip to replace in the Replace box.

Select:	To:
Replace Front	Replace the front clip of the applied layer.
Replace Matte	Replace the matte clip of the applied layer.

7. If you select Custom, select the clip to use from the image parented by the source.

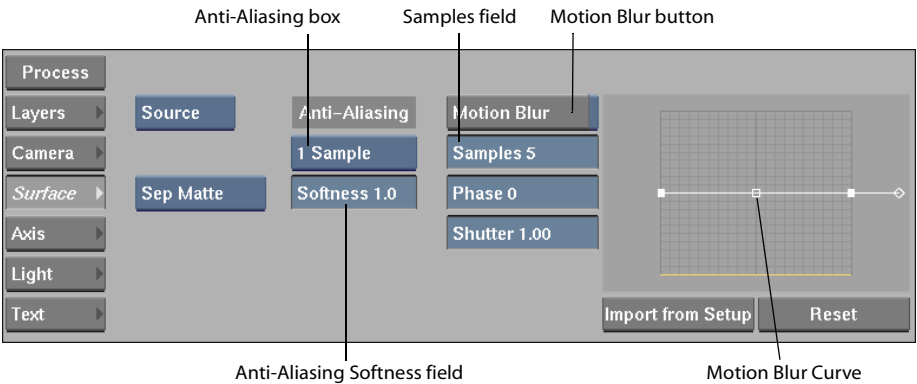
For example, if you select Matte, only the matte of source's child image is rendered and used.

Select:	To:
Front	Use the front clip from the source's child image.
Matte	Use the matte clip from the source's child image.
F+M	Use both the front and matte clip from the source's child image. The front and matte clips are combined before they are applied.

8. Select the image parented by the source and apply the layer containing the replacement clips.

Using Motion Blur with Source Nodes

You can set motion blur and anti-aliasing specifically for either the front or matte clip of a layer. Use motion blur with Sources to simulate the blur created by fast-moving objects by blurring the motion of a front or a matte layer.



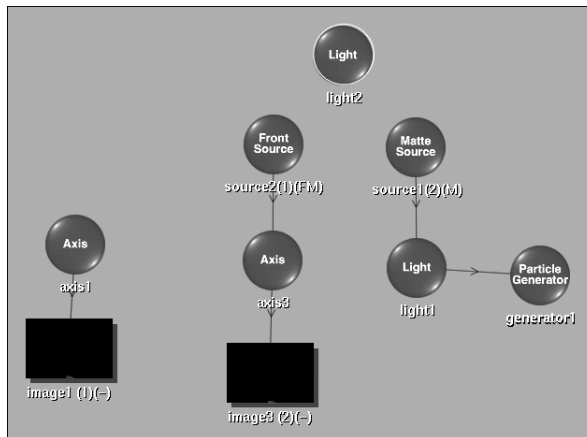
The Source motion blur controls are described in the following table.

Select:	To:
Anti-Aliasing box	Select level of anti-aliasing samples. Values of 4 and 8 give good results.
Anti-Aliasing Softness field	Adjust the degree of anti-aliasing. Use a value of 0.5 or 2.0 to get the cleanest image.
Motion Blur button	Apply a motion blur effect to the selected clip. To create this effect, the software takes samples of previous and subsequent frames and displays them over the current frame. You set the number of sample, the transparency (weight) of the samples, and the number of frames over which the samples are taken.

Select:	To:
Samples field	Determine the quality of motion blur. Increasing the number of samples improves the quality of the motion blur. Increasing the number of samples causes the processing time to increase linearly. The number of motion blur samples is multiplied by the number of anti-aliasing samples. To reduce the total number of passes made for each frame, reduce the level of anti-aliasing when Motion Blur is enabled.
Phase	Specify whether the motion blur is based on the movement before or after the current frame. A value of -100 places the motion blur before the current frame while a value 100 places the motion blur after.
Shutter field	Set size of motion blur. It is essentially the number of frames that the shutter is open. Increasing this value does not increase processing time.
Motion Blur curve	Set the weight distribution (or transparency) of the samples.
Import from Setup button	Use the anti-aliasing and motion blur values in the Setup menu. See "Using the Secondary Setup Menu" on page 903.
Reset button	Reset the anti-aliasing and motion blur settings to their default values.

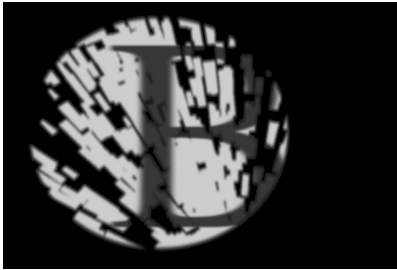
Replacing Clips with Other Effects

A source does not have to parent a surface. Geometry, text, particle systems, and other effects can be parented by a source when you want to create effects that are applied in conjunction with a matte. For example, the following schematic combines the matte from one layer with a particle generator.



The first source (source1) replaces the matte clip of layer2 with a particle generator. The second source (source2) replaces the matte clip of layer1 with a combination of the front and matte clip (particle generator) of layer2.

In the main branch, the front clip in layer1 (the letter B) is drawn using a circular matte combined with a particle generator. The circular matte was loaded as the front clip of layer2 and is combined with the particle generator in the second source (source2). The following figure shows the result of the schematic shown in the previous figure.



Parenting and Redrawing Source Nodes

A source cannot be parented by other objects in Schematic view. If you attempt to parent a source, the connection is refused. If a source is unparented and then reparented, the surfaces below it may not be rendered properly. Click the Update button to redraw the surfaces in the scene. Update is also used to redraw particle streams.

NOTE: Be careful when you unparent or delete a source. Surfaces or other objects that were parented by the source are added to the scene.

Applying Effects with Source Nodes

Cropping and other layer-based effects that modify a layer's clips must be used on the replacement layer and not on the original layer. Otherwise, the effect is overridden.

For example, if the matte of layer1 is being replaced by the matte of layer2, cropping must be done on the matte of layer2. Any cropping, blurring, etc., applied to the matte of layer1 is lost when it is replaced by the matte of layer2. Note that the front of layer1 can still be cropped or blurred as usual.

[illegible]

Action: Camera and the Scene

Changing your perspective

In Action, you view the scene through the camera eye. You can control how the camera views surfaces and animate the camera position in the scene.

Summary

In this chapter, you learn about:

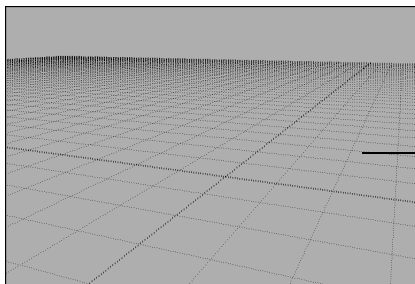
- “Working in the Scene” on page 939
- “Using the Camera” on page 945
- “Using the Schematic” on page 955
- “Animating the Scene Using the Channel Editor” on page 965

About the Camera and the Scene

In Action, the scene is actually what you see through the camera lens. Typically, you work with the camera to frame and animate the view to achieve the effect you want in the final result of your clip.

Working in the Scene

In Action, the scene is a representation of 3D space with its own axis system, called *world space*. World space has three directions: X (left/right), Y (up/down), and Z (in/out).



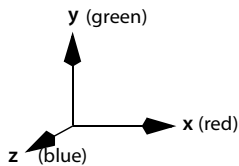
The scene in Action can be thought of as a 3D grid with X, Y, and Z coordinates.

You use the X,Y, and Z coordinate system to place each surface in the scene. You also use this coordinate system to rotate, scale, and shear surfaces. You move, rotate, and animate objects directly in the scene and use the camera to record the scene. The part of the scene that the camera, or frustum, looks at is what gets rendered.

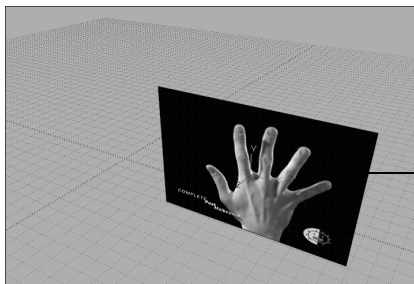
You can view the scene through the camera eye or other scene views. Use the orthographic views to place objects in the scene and animate the position of the camera. The schematic is an alternative view that uses nodes to represent axes, surfaces, and other objects in the scene. Connecting arrows in the schematic show the relationships between the objects.

Adding Objects to the Scene

When you add certain objects to the scene, they are added with their own axes. Each axis is used to place its respective object in the scene. An axis is represented by the following icon.



All transformations that you apply to the selected axis are applied to the objects connected to the axis. For example, if the position of a surface's axis is set to 500, 100, 0, then its surface is placed at 500 on the X-axis, 100 on the Y-axis, and 0 on the Z-axis.



Surface is placed
in the scene at
500, 100, 0

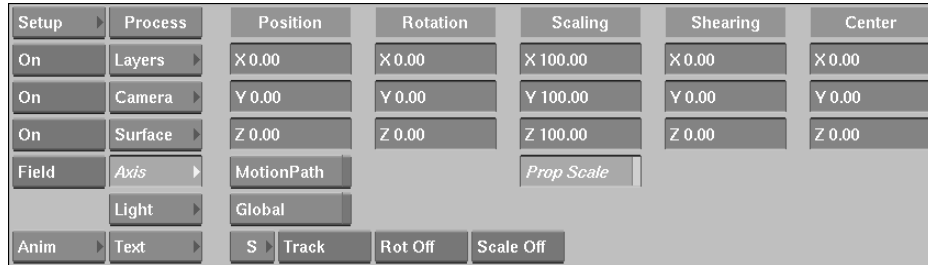
The connection between an axis and its object is called a *parent-to-child relationship*. Any changes in position applied to the parent object are equally applied to the child. See “Working with Branches in the Schematic” on page 963.

The following objects are added with axes:

- All surfaces: Image, Bilinear, Bicubic, and Extended bicubic
- Imported 3D models, text, or paint geometry
- Shadows
- Textures
- Deformations

Manipulating an Axis

You use the controls in the Axis menu to position, rotate, scale, and shear an axis. To display the Axis menu, click the Axis button in the Action menu.



The controls in the Axis menu are described in the following table.

Use:	To:
Position X, Y, and Z fields	Translate the selected axis.
Rotation X, Y, and Z fields	Rotate the selected axis.
Scaling X, Y, and Z fields	Change the size of the axis.
Prop Scale button	Scale the X, Y, and Z axes proportionally.
Shearing X, Y, and Z fields	Shear the axis.
Center X, Y, and Z fields	Offset an axis relative to its children.
Motion Path	Animate the position of the axis using a spline drawn in the scene. Disable Motion Path to animate the position of the axis using explicit animation. See “Using Explicit Animation” on page 148.
Global	Ignore transformations from more parent axes. Enable Global when you want to create a bouncing particle stream. See Chapter 47, “Action: The Particle System.”
Stabilizer and tracking controls	Apply stabilizing data to an axis. See “Tracking” and “Stabilizing” on page 806.

Selecting an Axis

You can select an axis in any of the following ways:

- Click directly on the axis in the scene.
- Go to Schematic view and click the node for the axis.
- Display the Channel Editor and select the Axis folder or one of its channels.
- Use the Prev and Next buttons in the Axis menu to select the previous or next axis. You can also use these buttons in the Layers menu to switch between the objects in the scene.

Moving and Rotating Axes in the Scene

You can move and rotate an axis directly in the scene using the Move and Rotate options in the Edit Mode box. You use Move and Rotate when viewing the scene in Perspective, Top, Side, or Front view. See “Viewing the Scene” on page 949.

The selected mode remains in effect until you select a different mode.

Edit Mode box



Moving an Axis in the Scene

You can move an axis in the scene, as well as objects that do not have exclusive axes such as lights, particle generators, and manipulators. You can also change the X, Y, and Z position gesturally by dragging any of the position fields to move the axis.

To move an axis directly:

1. In the Edit Mode box, select Move.
2. Select the axis you want to move and drag it to a new position.

If the Axis menu is displayed while you move an axis, you can see the Position X, Y, and Z fields update after the axis is placed in its new position.

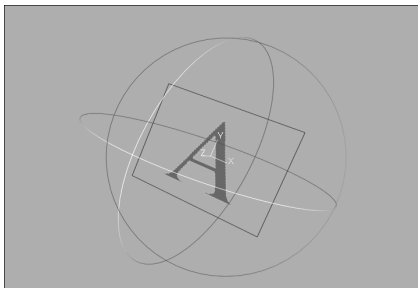
Rotating an Axis in the Scene

The following steps show how to rotate an axis in the scene. You can also use this procedure to rotate objects that are not added with their own axes, such as lights, particle generators, and manipulators.

To rotate an axis directly:

1. In the Edit Mode box, select Rotate.
2. Select the axis or the object you want to rotate.

Action displays a trackball in the scene. The trackball consists of three concentric rings. Each ring is used to rotate the object and its axis around one of the three axes.

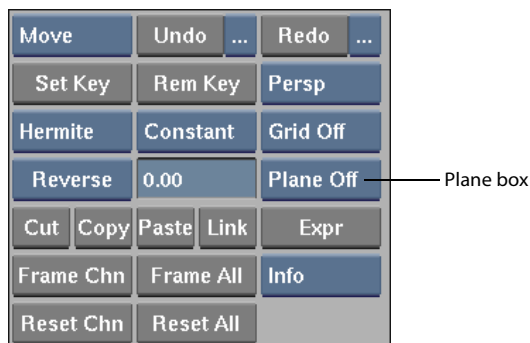


3. Click and drag one of the trackball rings.

The axis and its object rotate in the image window. If the Axis menu is displayed while you rotate, you can see the Rotation fields update after the axis is rotated to its new position. You can also drag any of the Rotation fields to rotate the axis and its object.

Changing the Plane

When you move an object in the scene, the object moves on a 2D plane. By default, an axis is moved gesturally on all three planes. You can change the orientation of the plane by selecting one of the options in the Plane box.



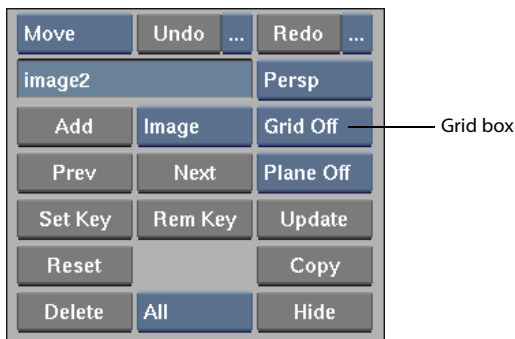
For example, if the camera is pointed toward the Y plane and you want to move an axis along the X and Z planes, you can change the orientation to PlaneXZ.

Select:	To:
Plane XY	Move the object on the X or Y plane, but not on the Z plane.
Plane XZ	Move the object on the X or Z plane, but not on the Y plane.
Plane YZ	Move the object on the Y or Z plane, but not on the X plane.
Plane Off	Move the object on the X, Y, or Z plane. The plane is oriented to face the camera.

NOTE: The different planes can only be used in Perspective view. You see the change in the orientation of the plane only when you move the camera away from its default position.

Using a Grid

You can display a grid in the scene to position objects in the scene more accurately. Use the Grid box to select the grid orientation.



The grid orientation options are described in the following table.

Select:	To:
Grid Off	Disable the grid.
Grid XY	Use a grid constructed on the X and Y planes. This is the default setting.
Grid XZ	Use a grid constructed on the X and Z planes. The XZ grid is visible only when the camera is moved from its default position.
Grid YZ	Use a grid constructed on the Y and Z planes. The YZ grid is visible only when the camera is moved from its default position.

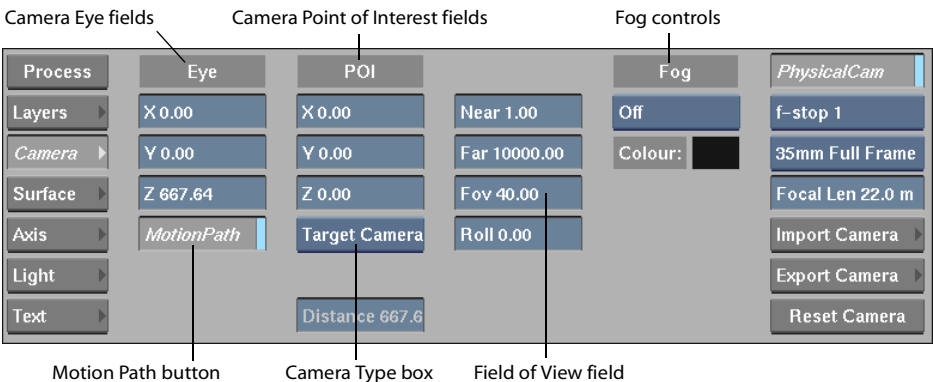
Use a snap grid to align objects in the scene by enabling the Snap button in the Setup menu. See “Using the Main Setup Menu” on page 896.

NOTE: The Action grid is independent from the general grid in the Grids and Guides menu. See “Grid and Guides Menu” on page 66.

Using the Camera

The camera is used to “record” the scene in Action. By default, the camera consists of two objects: the camera eye and the point of interest.

Use the Camera menu to control the position of the camera eye, point of interest, and the near and far clipping planes. You can control camera roll and the field of view angle. You can also set the camera using a simulation of a physical camera. To display the Camera menu, click the Camera button in the Action menu.



The Camera controls are described in the following table.

Select:	To:
Camera EyeX, Y, and Z fields	Edit the position of the camera eye on the horizontal, vertical, and perpendicular (X, Y, and Z) axes.
Motion Path button	Animation the camera eye on a motion path. See “The Camera Eye and Point of Interest” on page 948.
Camera point of interest X, Y, and Z fields	Edit the position of the point of interest on the horizontal, vertical, and perpendicular (X, Y, and Z) axes.
Camera Type box	Select Free or Target Camera. A Free Camera views the scene in the direction that you aim the camera. Free cameras are easier to use because you do not have to manipulate the point of interest. You can simply animate the camera rotation or camera tilt as though it were on a tripod. A Target Camera ensures the camera is specifically aimed at a target object in the scene because you specify the point of interest. Use the Roll field in conjunction with the Target Camera. Use the Distance field in conjunction with the Free Camera.
Near and Far	Edit the position of the near and far clipping planes. See “The Clipping Planes” on page 948.


Select:	To:
Field of View field	Specify the camera field of view, measured in degrees. When PhysicalCam is disabled, use the field of view angle in the Y direction to adjust the width of the camera frustum. When PhysicalCam is enabled, use the focal length for the same purpose.
Roll field	Set the amount of camera roll. This field is available with the Target Camera.
Fog box and colour pot	Create a foggy atmosphere in the scene. Use the colour pot to select the fog colour. See “Applying Fog to the Scene” on page 954.
Physical Cam button	Activate the physical camera, which simulates a manual camera. When enabled, the F-Stop, Film Size, and Focal Length fields appear. You use these fields to adjust the field of view for the camera.
F-Stop	Adjust the depth of field. F-stop is the ratio between the focal length of the lens and the diameter of the aperture. For example, the F-stop value for a 100mm lens with a 50mm full aperture is 2 (or f/2). Large F-stop values correspond to smaller apertures. The depth of field for your physical camera increases as you increase the F-stop value. In Action, you can select standard F-stops.
Film Size	As the film size increases, the focal length gets larger and the focal point moves further away from the camera eye. Changing the film size also updates the focal length. In Action, you can select standard film sizes from the Film Size box.
Focal Length	Adjust the field of view. Changes you make to the Focal Length automatically update the field of view (based on the film size) and vice versa. As the focal length increases, the field of view decreases. As the focal length decreases, the field of view increases.
Import Camera button	Import 3d studio max camera path information. Select the camera path file that contains the data you want. Typically this data is an ASCII representation of the scene—.ase. The fields Eye X, Y, and Z; Poi X, Y, and Z; Fov, Roll, Near and Far are updated according to the data contained in the imported file.
Export Camera button	Export camera path information from Action to 3d studio max format. Exported data includes Eye X, Y, and Z; Poi X, Y, and Z; Fov; Roll, Target or Free Camera; motion path and explicit keyframe camera animation.
Reset Camera button	Reset the Camera menu to its default settings.

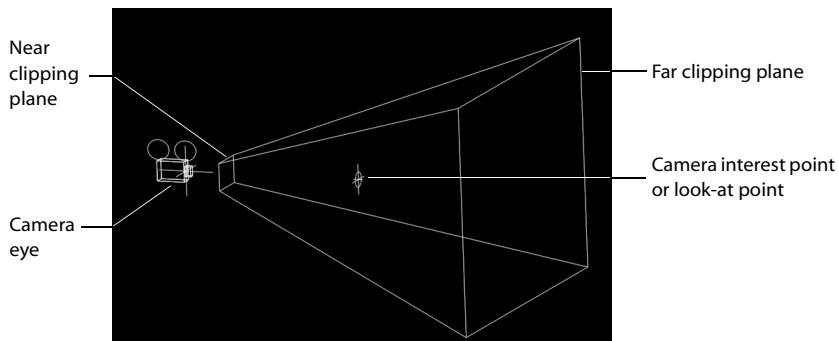
The Frustum

The volume of space viewed by the camera eye is called the frustum. The frustum is in effect a viewing pyramid. The camera eye is located at the apex of the pyramid, and the far clipping plane forms the base. The pyramid may be truncated by the near clipping plane.

If you place a surface within the frustum, it is visible in the final animation. If the surface is located outside the scope of the frustum, it will not be visible at that frame in the animation.

To see the camera and frustum:

1. In the Action menu, click Result.
2. In the Scene View box, select Side.
3. In the Image Window controls, click  to zoom out from the scene.
4. Select Pan in the Edit Mode box and pan around the scene until you see the camera eye icon. Alternatively, you can use Orbit mode to pan around the scene in circular motion.
5. Go to the Camera menu and drag the Roll field until you see the four sides of the frustum (the pyramid shape below):



You can modify the frustum by:

- Changing the position of the near clipping planes to alter the depth of the frustum.
- Changing the position of either the camera eye or the camera's point of interest to alter the orientation of the frustum.
- If the Physical Cam button is disabled, enter a value in the Fov field to adjust the width of the camera frustum.
- If the Physical Cam button is enabled, enter a value in the Focal Length field to narrow or widen the frustum. You can also alter the depth of the frustum using only the near clipping plane.

The Clipping Planes

The camera frustum is determined by six clipping planes: the left, right, top, bottom, near, and far clipping planes.

The frustum and field of view can be narrowed or widened by changing the left, right, top, and bottom clipping planes. This is achieved by modifying the camera's focal length. Increasing the value of the focal length narrows the frustum and field of view. Decreasing the value widens the frustum and field of view.

The depth of the frustum is affected by the near and far clipping planes. The values for these channels are expressed in units relative to the position of the camera eye.

The value in the Near field corresponds to the position of the near clipping plane. The default value is 1. Any object between the camera eye and the near clipping plane is outside the camera frustum and does not get processed in the final result.

The value in the Far field corresponds to the position of the far clipping plane. The default value is 10000. Any object positioned behind the far clipping plane is outside the camera frustum and does not get processed in the final result.

The Camera Eye and Point of Interest

Objects in the scene can be recorded from an arbitrary position as determined by the orientation of the camera eye in world space. To change the position of the camera eye, modify the EyeX, EyeY, and EyeZ fields in the Camera menu.

You can also animate the camera eye on a motion path. Enable the Motion Path button and drag the camera icon while viewing the scene in Top, Side, or Front view.

The camera point of interest is the point in world space at which the camera eye is directed. The point of interest is always at the center of the camera's frustum; changing the position of the point of interest causes the orientation of the frustum to change. You can take advantage of the relationship between the point of interest and the frustum to make the camera follow a moving object. To do this, animate the point of interest while keeping the camera eye in a fixed position.

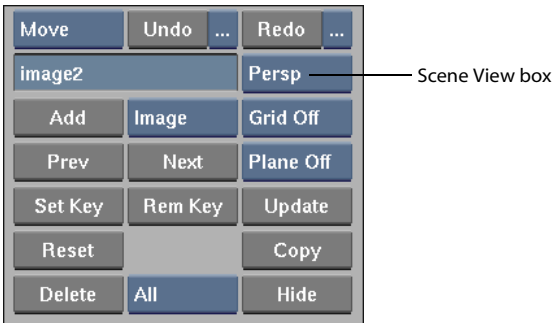
To change the point of interest, modify the PoiX, PoiY, and PoiZ fields in the Camera menu. You can also move the point of interest by dragging while viewing the scene in Top, Side, or Front view. You can animate the point of interest using a motion path, by enabling the Motion Path button.

NOTE: You can only animate the point of interest with a Target Camera.

Viewing the Scene

The scene appears in the image window and you can view the scene from various angles and use multiple views of these angles simultaneously. Changing your view of the scene and having the ability to view the scene from multiple angles at one time are convenient for setting motion paths, light sources, camera angles, and animation keyframes more accurately.

You use the Scene View box to change the view.

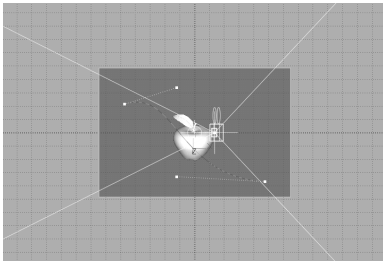


Using Perspective and Orthographic Views

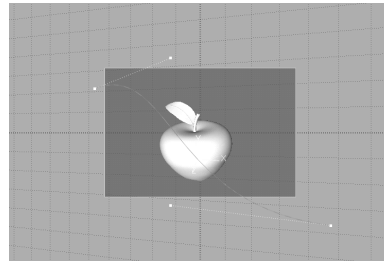
You can view the scene from Perspective view and three different orthographic views. In Perspective view, an object becomes smaller as it moves farther away from the camera. In orthographic view, an object remains the same size, regardless of its distance from the camera. Orthographic views are more helpful for aligning objects.

Select:	To:
Persp	View the scene in perspective. This is the scene as viewed by the camera eye. In other words, your field of vision in world space is equivalent to the viewing frustum of the camera. The size of objects depends on their distance from the camera eye.
Top	View the scene as if you are positioned on the positive Y-axis. This is an orthographic view; there is no perspective deformation.
Side	View the scene from the side, as if you are positioned on the positive X-axis. This is an orthographic view; there is no perspective deformation.
Front	View the scene as if your line of vision is directed into the camera eye. This is an orthographic view; there is no perspective deformation.

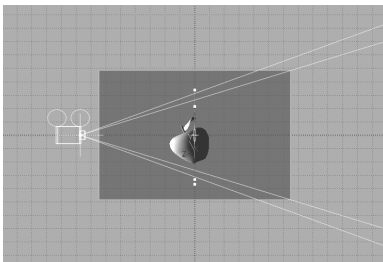
The following figures illustrate the different angles by which the scene can be viewed. The scene in this example contains a grey back clip and the 3D model of an apple.



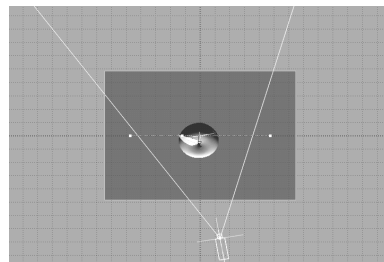
Front view



Perspective view



Side view



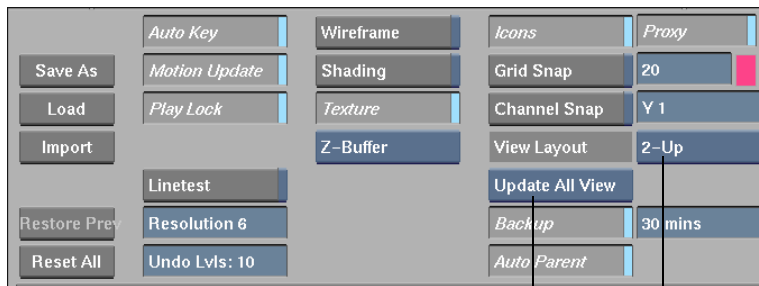
Top view

In addition to perspective and the three orthographic views, you can also select a Schematic view, which uses nodes to represent the objects in the scene and arrows to illustrate the relationships between objects. See “Using the Schematic” on page 955.

Displaying Multiple Views Simultaneously

You can display up to four views at a time in the image window including a view of the Channel Editor. Multiple views are convenient for setting channel values, working in Schematic view, and previewing your result in different views all at the same time.

You use the View Layout box in the Setup menu to set the number of views to display in the image window.



View Update box

View Layout box

To set the number of views to display:

1. In Action, click Setup.
2. In the View Layout box, select 2-Up, 3-Up, or 4-Up.

Hot Key: For View Layout:**ALT-1** 1-Up**ALT-2** 2-Up**ALT-3** 3-Up**ALT-4** 4-Up

The image window displays the views according to your selection. The current view is outlined in yellow.

View names appear in for each view in a multiview window.

NOTE: Channel view has no title.

3. In the View Update box, select Update Sel View to update only the currently selected view; or select Update All View to update all views simultaneously.

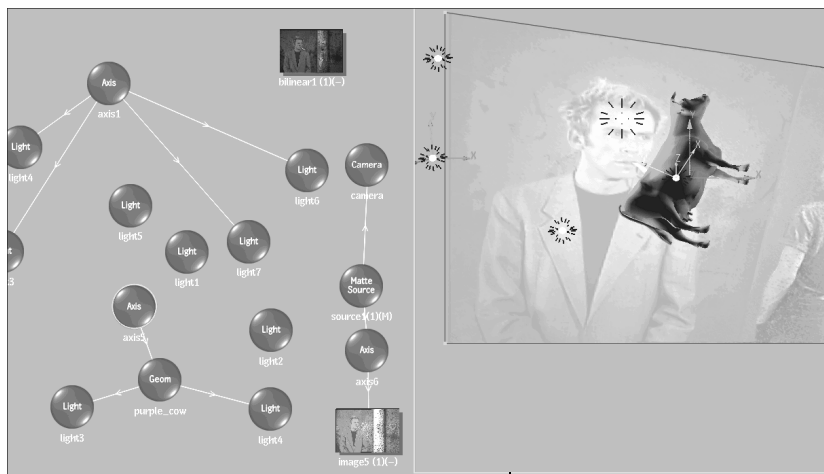
To set the content of a view in the image window:

1. In the image window, select the view whose contents you would like to change.

The selected view is outlined in yellow.

2. Click Anim, and in the Animation menu, select a view from the Scene View box.

The following example illustrates a 2-Up view layout using Schematic and Perspective views.



The current selection is outlined in yellow.

Using Layer Views and Context View

When working with multiple views, you can also display individual clip layers in a view. These views are convenient for viewing the result of slipping and cropping individual layers. See “Back Slip, Front Slip and Matte Slip fields” on page 908.

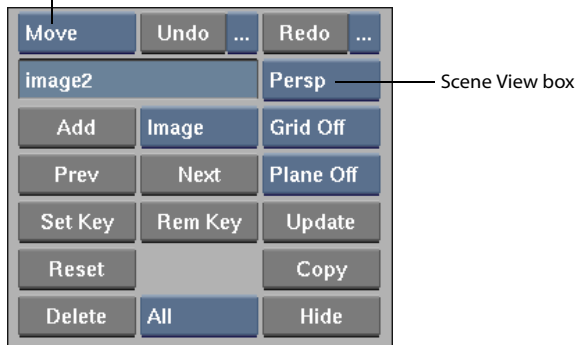
Select:	To:
Layer Back	View the back layer for the selected layer in the schematic or Layers list.
Layer Front	View the front layer for the selected layer in the schematic or Layers list.
Layer Matte	View the matte layer for the selected layer in the schematic or Layers list.
Context	View the context that is set in Batch when accessing Action through Batch. Action uses Context1 as it is set in Batch. For example, if the Batch context is set on the result of an Optics node for which the current Action node is an input, changes you make it Action appear in context of the Optics node. See “Viewing Clips In Context” on page 582.

Modifying the Camera in the Scene

You can modify the camera directly in the scene using the Pan, Zoom, and Orbit options in the Edit Mode box. A mode remains in effect until you select a different mode. These modes may be used in other views in Action.

NOTE: Pan, Zoom, and Orbit modes do not apply to Schematic view, Layer Back, Layer Front, and Layer Matte, and Context views.

Edit Mode box



Panning the Camera

Use the Pan option to pan through the scene or to pan the camera. In Perspective view, the camera eye and point of interest pan. In Top, Side, or Front view, you scroll through the scene without affecting the camera.

To use Pan:

1. In the Edit Mode box, select Pan.
2. Place the cursor in the image window and drag in any direction.

Orbiting the Point of Interest

Use the Orbit option to rotate the camera eye around the point of interest and change the orientation of the camera's frustum. Use Orbit in the following views.

Select:	To:
Perspective	Rotate the camera around the point of interest in the same direction of the dragging direction.
Top	View the scene from above.
Side	View the scene from the side.
Front	View the scene from the front.

To use Orbit:

1. In the Edit Mode box, select Orbit.
2. Place the cursor in the image window.
The cursor changes to a black arrow.
3. Drag the cursor.
The camera eye rotates around the point of interest.

Zooming In and Out

Use the Zoom option to move the camera eye toward or away from point of interest. While viewing scene in Perspective view, zoom in or out from point of interest to move camera eye closer to or farther from point of interest. In Top, Side, or Front view, you can enlarge or reduce scene in image window without affecting the camera. Zooming has no effect in Schematic view.

To use Zoom:

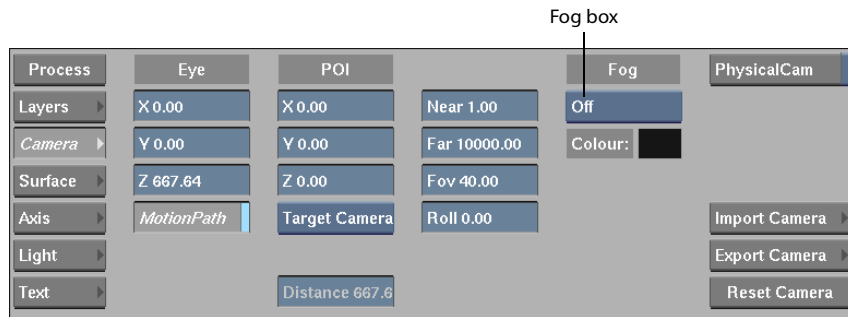
1. In the Edit Mode box, select Zoom.
2. Place the cursor in the image window.
The cursor changes to a magnifying glass.
3. To zoom in, drag the cursor to the left. To zoom out, drag the cursor to the right.

Applying Fog to the Scene

Use fog in the scene to create visual effects such as mist, fog, haze, and murky water. The fog effect causes objects to fade as their distance from the camera increases. The further away from the camera an object is the foggier the scene appears.

The fog effect is based on depth-cueing, which means the intensity of the fog varies along the Z-axis, and the specified colour gradually increases or decreases its intensity.

You control the density and colour of the fog using the fog controls in the Camera menu.



Try combining your fog effect with other effects such as transparency to create a more genuine look for non-solid substances such as water. You can also use expressions to improve the quality of fog in a scene so that it looks more natural. See “Working with Expressions in the Channel Editor” on page 165.

To add fog to the scene:

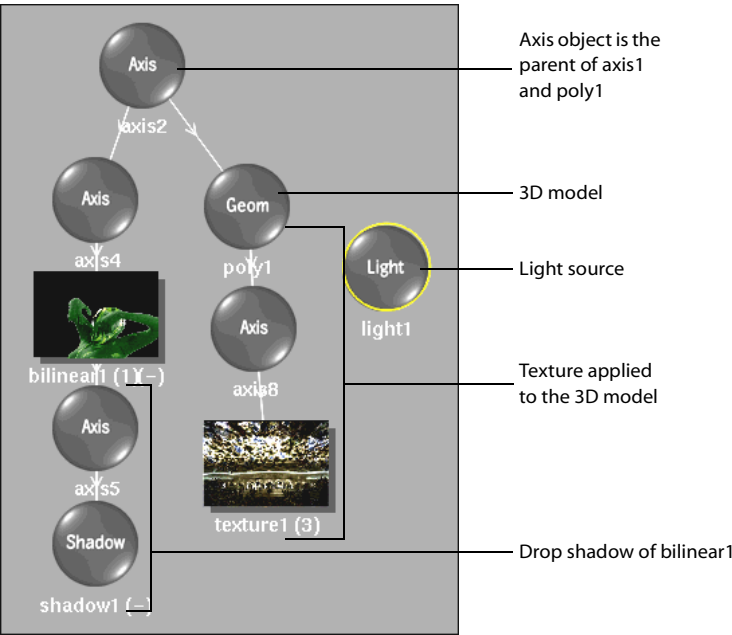
1. In the Camera menu, select the a fog type from the Fog box.

Type	Description
Linear	Specifies the distance at which fog should start and end.
Exp	$\text{Fog} = e^{-(\text{density} * z)}$ <p>where z represents the range or distance from the camera. The range should lie within the realm of the near and far clipping planes. The minimum value is 1.</p>
Exp2	$\text{Fog} = e^{-(\text{density} * z)^2}$ <p>The exponential types provide more depth, more natural looking fog as well as appearing much smoother.</p>

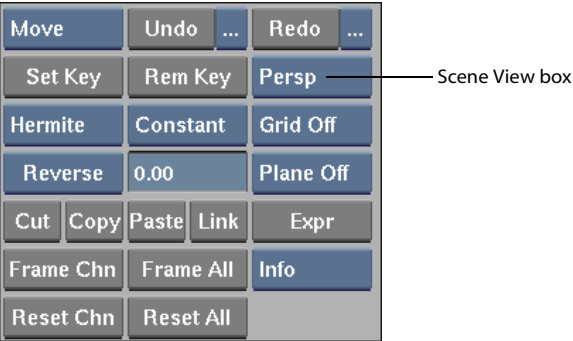
2. For Linear fog, specify the distance at which the fog should start and end in the Start and End fields. For Exp or Exp2 fog, specify the range in the Range field.
3. To select the colour of the fog, click the colour pot to display the colour picker.
Typically, the fog colour should match the background colour of the image in the scene.

Using the Schematic

The Schematic view uses nodes to represent all Action objects in the scene—including the camera—and shows the relationship between objects such as an image and its axis. Use the schematic to control every object in the scene, set parent-to-child relationships among nodes, as well as select objects more easily.



To view the schematic, select Schematic in the Scene View box, or press the ~ key. Pressing the ~ key a second time returns to the previous view.



Working with Nodes

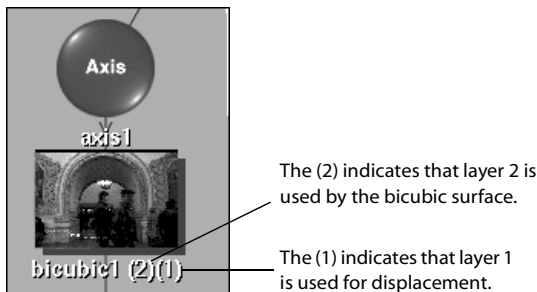
In Schematic view, a node exists for each object in the scene. For example, the camera, axes, shadows, and lights are represented by circles. Surfaces, projectors, and textures are represented by rectangles. Each node displays the name of the object it represents.

You work with nodes to create parent-to-child relationships between objects and set up complex animations. See “Working with Branches in the Schematic” on page 963.

Surface Nodes

A surface node is the general term used for nodes that represent image, bilinear, bicubic, and extended bicubic surfaces. See “Adding Objects” on page 957.

When you add one of these surface nodes to the schematic, the node name is shown with two numbers in parentheses. These numbers indicate the layer to which the surface belongs and the layer used as a displacement source. For example, a bicubic object labelled (2)(1) shows that the bicubic uses the clips from Layer2 and is displaced using the clips from Layer 1.



For more information on the relationship between layers and surfaces, see “Working with Layers” on page 908 and “Using Layers on Surfaces” on page 917. For more information on displacement, see “Displacement Mapping” on page 999.

Camera Node

The camera node appears in Schematic view by default and you can link it to an image. Use the camera node to rotate the camera about its own axis, and parent other nodes including Shadow, Texture, and Geom nodes.

Shadow and Texture Nodes

Shadow and texture nodes each have a single number in parentheses that indicates the layer used for the shadow or texture. For example, a shadow labelled (2) shows that the shadow uses the matte from Layer2.

Geom Nodes

Geom nodes are added to the scene when you import a 3D model, geometry from Paint, or when you create 3D text.

Source Nodes

Source nodes are used as part of an advanced schematic structure that separates a layer’s matte and front so that each clip can be animated individually. You can also use sources to create complex compositing effects such as nesting. See “Using Source Nodes” on page 931.

Projector Nodes

Projector nodes are used to project images in the scene. The effect is similar to that of a slide projector. See “Projecting Textures” on page 996.

Adding Objects

You add objects to the scene using the Schematic controls. An object can be an axis, a surface, a texture, a shadow, or a light. The Schematic controls include the Add button and Object box in the Animation menu.



To add an object to the scene:

- 1. If you are adding an image, a bilinear, a bicubic surface, or an extended bicubic, select the layer that you want to use from the Layers list.
- 2. Select the type of object that you want to add from the Object box.

Select:	To:
Axis	Add an axis. See “Manipulating an Axis” on page 941.
Bicubic	Add a surface with four corners joined using bicubic bezier interpolation. See “Working with Layers” on page 908.
Bilinear	Add a surface with four corners joined using linear interpolation. See “Working with Layers” on page 908.
Deform	Add a deformation mesh to a selected image or 3D model. See “3D Deformations” on page 1011.

Select:	To:
ExtBicubic	Add a surface with four corners joined using bicubic bezier interpolation. You can subdivide Extended Bicubic surfaces to obtain additional tangents. See “Extended Bicubics” on page 1002.
Image	Add a surface without adjustable corners. See “Working with Layers” on page 908.
Light	Add a light source. See “About Light Sources” on page 967.
PartBnc	Add a particle bouncer. See “Working with Particle Bouncers” on page 1047.
PartGen	Add a particle generator. See “Generating Particles” on page 1017.
PartMan	Add a particle manipulator (referred to as a particle animator in the schematic). See “Working with Particle Manipulators” on page 1029.
Projector	Add a projected clip to the scene. See “Projecting Textures” on page 996.
Shadow	Add a shadow to the selected surface. See “Adding Shadows” on page 978.
Source	Add a source node. See “Using Source Nodes” on page 931.
Texture	Add a texture to the selected geometry (Geom) or particle generator (PartGen). See “Working with Textures” on page 992.

NOTE: Only one camera can exist in the Action scene, and it appears in the schematic by default.

3. Click Add.

The object is added to the scene and a node representing the object is added to the schematic.

Selecting Objects

Before you can edit or animate an object in the scene, you must select it. You can select an object in the following ways:

- In the scene, click an object. For example, to select an image, bilinear, or bicubic surface, click one of the surface’s edges.
- In Schematic view, click a node representing the object.
- In the Channel Editor, select an object’s name or one of its channels.

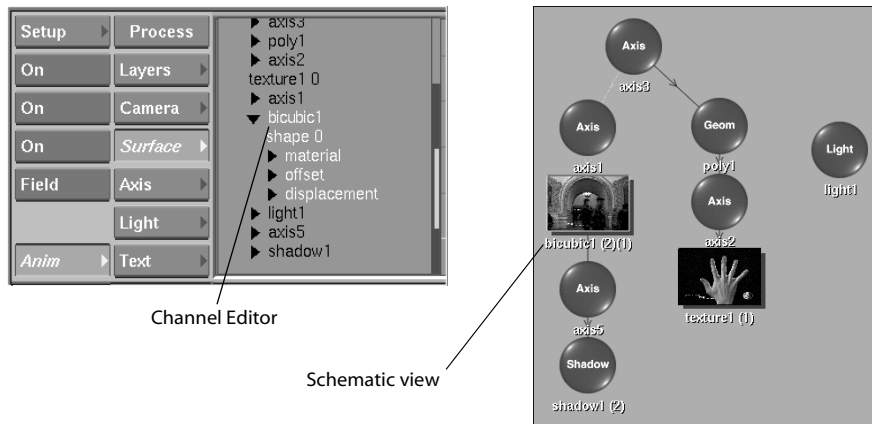
You can also use the Prev and Next buttons to select the next or previous object depending on the current menu.

In this menu:	Use the Prev and Next buttons to select:
Layer and Camera	Previous or next object.
Surface	Previous or next surface.
Axis	Previous or next axis.
Light	Previous or next light source.
Text	Previous or next text object.

Naming Objects

Each object that you add to the scene is automatically assigned a default name. The default name identifies the object and the order in which it was added. For example, image surfaces are named image1, image2, image3, and so on. Similarly, bilinear surfaces are named bilinear1, bilinear2, bilinear3, and so on.

An object's name appears beneath its node in Schematic view and as a folder in Channel Editor.

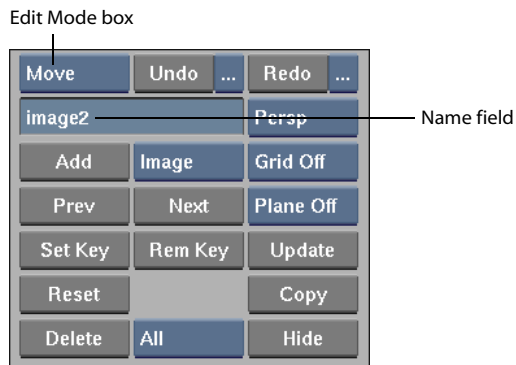


You can assign more meaningful names to objects. If you rename an object, the Schematic view updates to show the new name. The object is also renamed in the Channel Editor.

To change the name of an object:

1. Select the object in the scene.

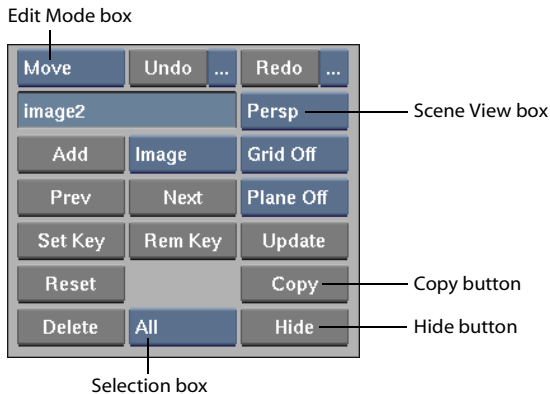
The current name for the object appears in the Name field.



2. Click the Name field and type the new name.

Modifying Objects in the Schematic

While in Schematic view, use the schematic controls to move, parent, unparent, copy, hide, or delete objects. Schematic controls include the Edit Mode box, and the Copy, Hide, and Delete buttons.



Moving Objects

Use the Move mode to move nodes in the schematic. Note that moving the nodes in Schematic view has no effect on the relationships between objects, nor does it affect the position of the objects in the scene. If Auto Parent is enabled in the Setup menu, Move mode acts as both Move and Parent.

To move a node in the schematic:

1. In the Edit Mode box, select Move.
2. Click the node you want to move and drag it to a new position.

To move an object and its children, press and hold the **ALT** key while dragging the parent.

Deleting Objects

You can remove nodes from the schematic.

To delete an object:

1. In the Edit Mode box, select Delete.
2. Click the node corresponding to the object that you want to delete.

The object is removed from the scene. You can also delete nodes using the Delete button.

To delete objects using the Delete button:

1. In the Selection box, select an option.

Select:	To:
Selected	Delete the currently selected object. For example, this option deletes a surface without deleting its parent axis, or an axis without deleting its surface.
Branch	Delete the selected object and all its children. To avoid deleting objects by mistake, use the Schematic view to determine which objects you are deleting.
All	Delete all objects in the scene.

2. Click Delete.

Parenting Objects

You use the Parent and Cut options in the Edit Mode box to change the parent-to-child relationship between nodes in the schematic. Parent mode makes an object the parent of another object. Cut mode removes the parent-to-child relationship between objects. Alternatively, when Auto Parent is enabled in the Setup menu, you can automatically parent objects in the schematic.

Use these options when you create branches. See “Working with Branches in the Schematic” on page 963.

Copying Objects

Use the Copy button to copy an object or a branch (object and its children).

To copy a single object or branch:

1. In the schematic, select the object you want to copy.
2. From the Selection box, select an option.

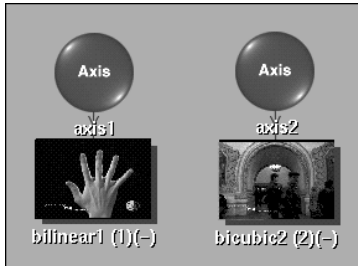
Select:	To:
Selected	Copy the currently selected object. For example, this option copies a surface without copying its parent axis, or an axis without copying its surface.
Branch	Copy the selected object and all of its children. To avoid copying objects by mistake, use the Schematic view to determine which objects you are copying.
All	Copy all objects in the scene.

3. Click Copy.

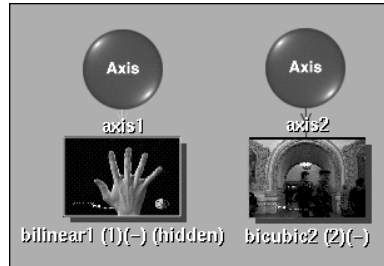
Hiding Objects

Use the Hide button to hide or unhide selected objects or branches. For example, when you are working on a specific surface, you may want to temporarily remove other objects from the scene without deleting them.

Hidden objects are marked “(hidden)” in Schematic view and do not appear in the scene. For example, in the *Before* schematic, both bilinear1 and bicubic2 are shown in the scene. In the *After* schematic, bilinear1 is hidden and does not appear in the scene.



Before: Bilinear1 is not hidden. It appears in the scene (below).



After: Bilinear1 is hidden. It does not appear in the scene (below).



To hide an object:

1. In the schematic, select the object you want to hide.
2. In the Selection box, select Selected.
3. Click Hide.

To hide a branch:

1. In the schematic, select the parent of the branch you want to hide.
2. In the Selection box, select Branch.
3. Click Hide.

To unhide an object or branch:

1. Select the hidden object or branch.
2. In the Selection box, select Selected or Branch.

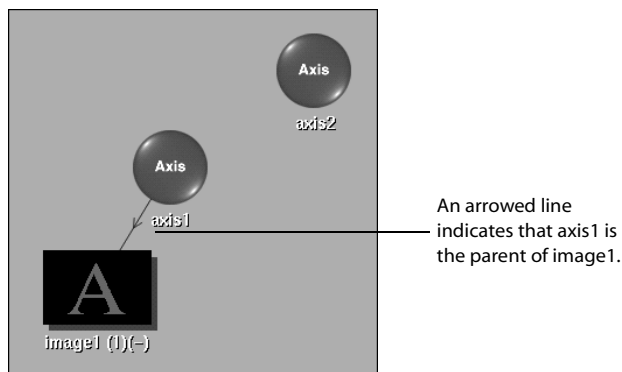
Use Selected if you are un hiding an object and Branch if you are un hiding a branch.

Working with Branches in the Schematic

In Action, you can create complex animations where movements applied to one object are passed down to all connecting objects.

The relationship between objects is referred to as a *parent-to-child* relationship where the *parent* object passes its animations down to its child *object*. The structure of one parent and one or more child objects is referred to as a *branch*.

When you add certain objects to the scene, a parent-to-child relationship is created automatically. For example, when you add a surface to the scene, it is automatically parented by an axis.



Parenting an Axis

You can add an axis to the scene by itself, then make it the parent of another object. Use this method of parenting additional axes to create complex animations.

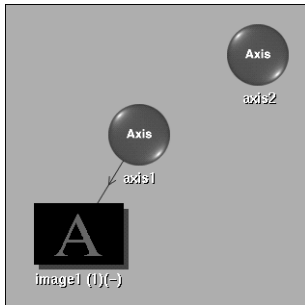
For example, you can create a cube of surfaces by parenting three additional axes to the same surface. Each axis that is parented to a surface places an additional surface in the scene. By changing the position and rotation of each axis, you can create a cube. If you parent the axes by another axis, you can control the position, rotation, scale, and shear of the cube.

To create a branch:

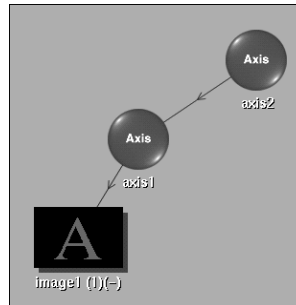
1. Add an axis (axis2) to the scene. See “Adding Objects to the Scene” on page 940.
2. In the Scene View box, select Schematic view. The Schematic view should be similar to the *Before* figure.
3. In the Setup menu, enable Auto Parent, and then in the schematic, drag the cursor from axis2 to axis1. Alternatively, in the Edit Mode box, select Parent and drag the cursor from the edge of the parent node to the node that will be its child.

NOTE: Press **ALT** and drag a node in the schematic to disable Auto Parent temporarily.

Axis2 becomes the parent of axis1, as shown in the *After* figure.



Before: The schematic view shows axis1 as the parent of image1



After: Axis2 is made the parent of Axis1 using Parent mode

Any transformations applied to axis2 are applied to axis1 and its surface (image1). If axis1 has any transformations, they are added to the transformations from axis2. For example, if axis2 is set to 500, 100, 0 and axis1 is set to -50, 20, -30, the positions are accumulated and applied to the surface. In this case, Image1 is positioned at 450, 120, -30.

Overriding Transformations

You can override the transformations passed from a parent to a child by enabling the Global button in the Axis menu. The Global button is used most often when bouncing or manipulating a particle stream. See Chapter 47, “Action: The Particle System.”

Removing Branches

You can use the Cut or Parent mode to undo the relationship between objects or you can simply drag the cursor across the line that joins the nodes.

To cut a parent-to-child relationship:

1. In the Edit Mode box, select Parent or Cut.
2. In the Scene View box, select Schematic.
3. Drag the cursor across the line that joins the two axes.

The connecting line is deleted and the relationship between the two axes no longer exists.

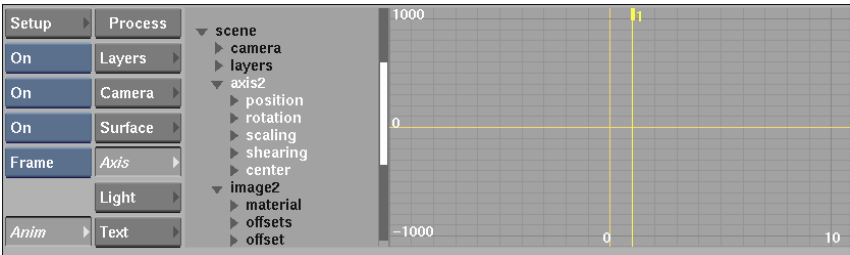
Animating the Scene Using the Channel Editor

Use the Channel Editor to animate the layer, axis, surface, light, and camera properties of every object in the Action scene.

To open the Channel Editor and display Action’s channels:

- 1. In Action, click Anim.
- 2. In the Animation menu, select Channel in the Scene View box.

The top level folder in the hierarchy is the Scene, which provides the overall view of the animation. See “The Channel Hierarchy” on page 130.



The Scene folder contains folders of objects found in the Action scene. Initially these are the Camera, Layers, Axis, and Image folders. If you add an object to the scene, the Channel Editor adds a folder to the Channel Hierarchy for the new object. For example, if you add a light, a Light folder is added in the Channel Hierarchy.

The following table describes some of the Action folders listed in the Channel Editor. For complete information and instructions about using the Channel Editor and animating objects in the scene, see Chapter 9, “Animation,” on page 127.

Select:	To:
Camera	Animate the camera position and point of interest. It also contains channels for animating camera roll, field of view, and the near and far clipping planes. When Free Camera is selected, the point of interest channels are replaced by rotation channels in the Channel Editor.
Layers	Animate layer properties such as blur, crop, shadow softness. A Layer folder is listed for each layer in the scene.
Axis	Animate axis properties such as the position, rotation, scaling, and shearing.
Image, Bilinear, Bicubic, or Extended bicubic	Animate surface properties such as material, offset, and displacement. The Material folder contains a shininess channel and folders for the specular highlight, diffuse, and ambient lighting.
Shadow	Animate the shadow colour and shadow transparency.
Light	Animate light properties such as intensity, falloff, spread, position, rotation, and colour.

[illegible]

45

Action: Light and Shadows

Lighten up!

In Action, objects are lit up in the scene according to the number, position, direction, and colour of light sources, as well as the rotation and spread of each light source.

Summary

In this chapter, you learn about:

- “The Light Menu” on page 969
- “Selective Lighting” on page 971
- “Working with Surface Lighting” on page 974
- “Adding Shadows” on page 978

About Light Sources

In Action, you can add any number of light sources to the scene, and each one can be controlled individually. Using the Light menu, you control light sources and set lighting properties such as position, orientation, spread, falloff, and colour.

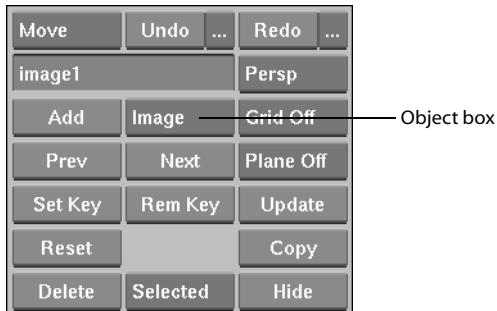
By default, the light you add to the scene is applied to all surfaces. However, you can also apply a light source to specific surfaces using the Selective Lighting feature.

Adding a Light Source

Use the Object box to add lights to the scene.

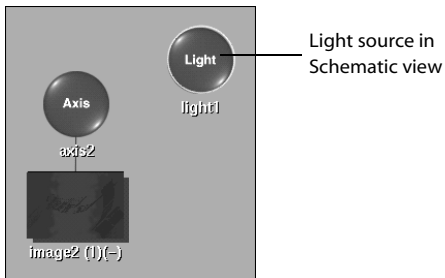
To add a light to the scene:

1. In the Object box, select Light.



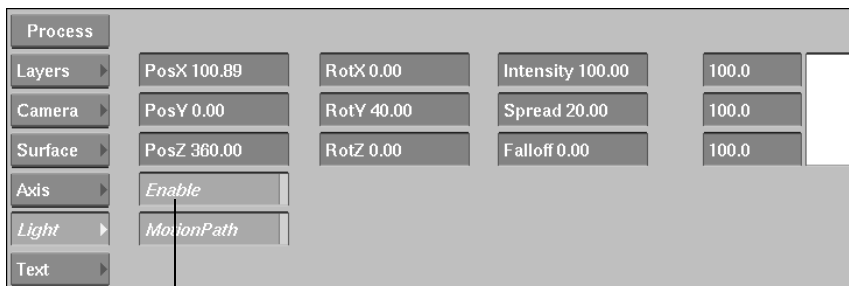
2. Click Add.

Action adds a light to the scene. An icon representing the light source is added to schematic.



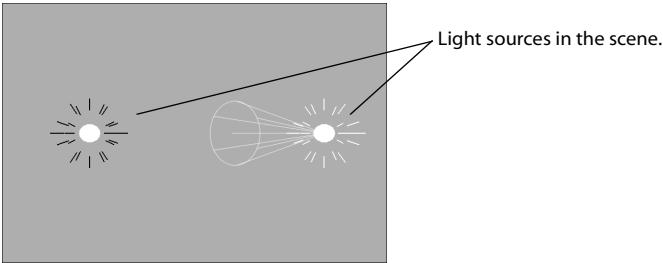
Unlike many objects, a light is added without an axis. To set the position and rotation of a light source, use the Position and Rotation fields in the Light menu.

3. With the new light source selected, click Light to open the Light menu.



Click Enable to enable the light source

4. In the Light menu, click Enable to activate the light source.
When Enable is activated, the light icon appears in the scene.



All light sources initially appear at the same X, Y, and Z position in the scene (0, 0, 0). If you add two light sources, for example, you need to move one light source in order to see the other.

Selecting and Manipulating a Light Source

You can select a light source in the following ways:

- Click the light source in the scene.
- In Schematic view, click the icon corresponding to the light source.
- Select a Light folder in the Channel Editor.
- Use the Prev and Next buttons to select the previous or next light source.

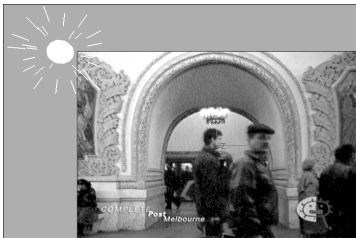
The Light Menu

The values for the currently selected light source are shown in the Light menu. When the light is selected, you can change its properties and animate its position. Click Light to display the Light menu.



The Light controls are described in the following table.

Select:	To:
PosX, PosY, and PosZ fields	Place the light source in the scene.
Enable button	Turn the selected light source on or off.
Motion Path button	Animate the position of the light using a spline drawn in the scene. Disable Motion Path button to animate position of light using explicit animation. See "Using Explicit Animation" on page 148.
RotX, RotY, and RotZ fields	Rotate the light source.
Intensity field	Adjust the intensity of the light.
Spread field	Use a light source as either a point light or a spotlight. A point light radiates light uniformly in all directions. A spotlight radiates a cone of light centred along the spotlight direction. Use Spread field to change spread angle. A spread of 90 or less creates a spotlight.



Falloff field	Adjust the amount of falloff around the edge of the light source. This value also changes the size of the specular highlight. A lower falloff value creates a larger specular highlight.
Colour	Change the colour of the light source by entering values in the red, green, and blue channel fields or using the colour picker. See "The Colour Picker" on page 57.

Selective Lighting

When you add a light source to a scene, the light is applied to all surfaces. You may want a light source to only illuminate an individual or specific group of surfaces. To do this, use the selective lighting option in Schematic view. Use the Lighting option to parent a light source to the surfaces you want to illuminate. Selected surfaces have blue dotted lines with an arrow.

The following procedure uses these three clips:



image1

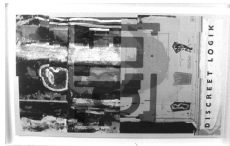


image2



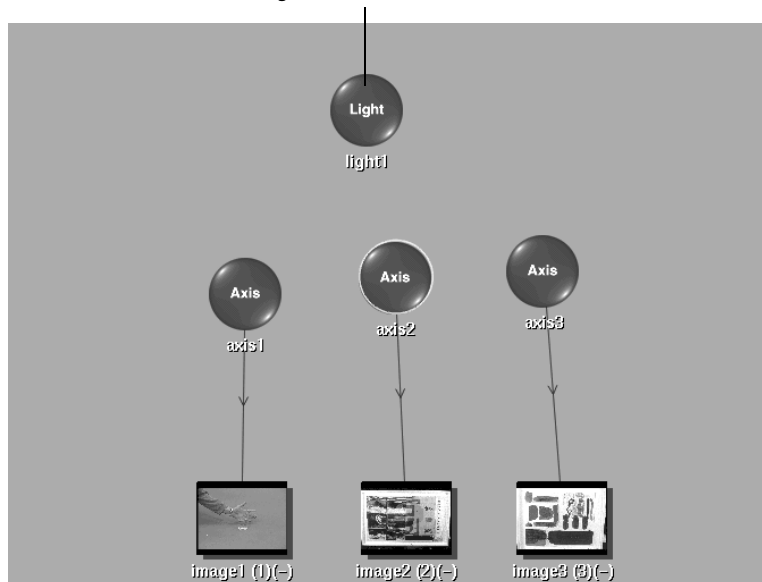
image3

To use selective lighting:

1. Add multiple surfaces to the scene.
2. Add a light to the scene.

All surfaces are illuminated.

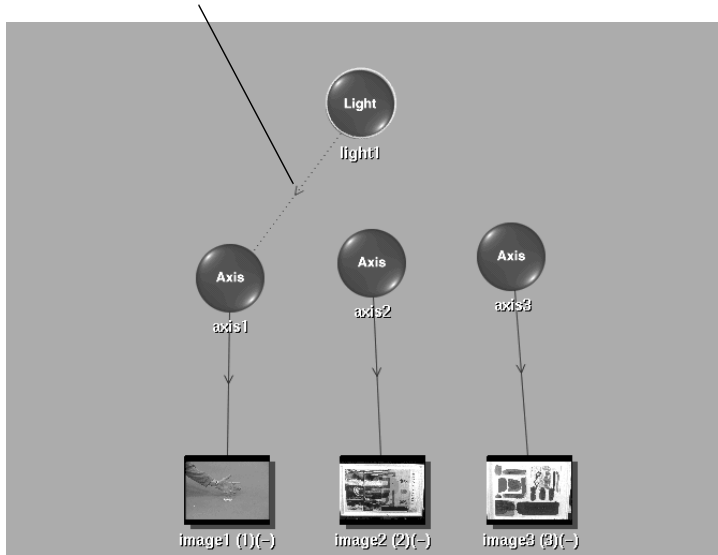
This light illuminates all surfaces.



3. Select Lighting from the Edit Mode box.
4. Click the light node, and drag it to an axis or image you want illuminated.

The selected object is connected to the light source by a blue dotted line with an arrow, and only the selected surfaces are illuminated.

Only the branch that has Axis1 as a parent is illuminated.



NOTE: You can remove this link by clicking and dragging over the parent line from the light source to the axis or image.

Excluding Surfaces from a Light Source

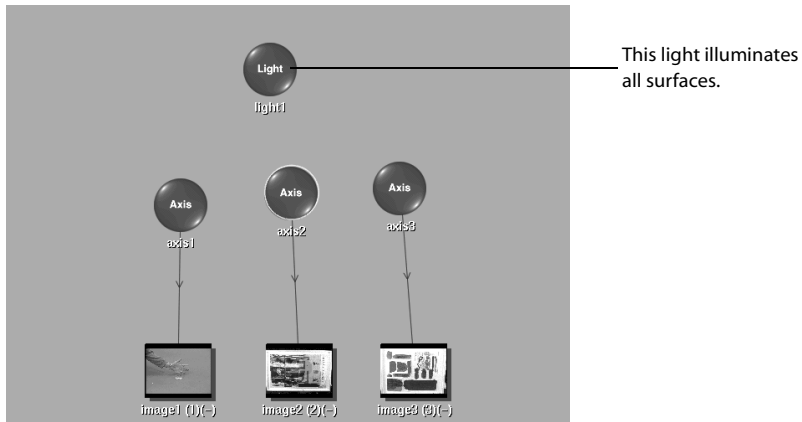
Using selective lighting, you can also omit a surface from being illuminated. For example, if you have two images parented to the same global axis and you only want to illuminate the first surface, you can parent the light source to the global axis and then exclude the surface you do not want illuminated.

The following procedure uses the same three images as the previous one.

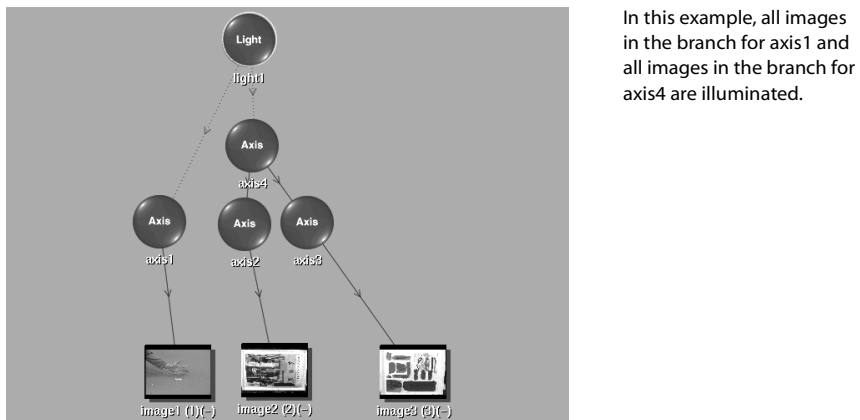
To exclude a surface from a light source:

1. Add three surfaces to the scene.
2. Parent two surfaces to the same axis.
3. Add a light source.

All surfaces are illuminated.

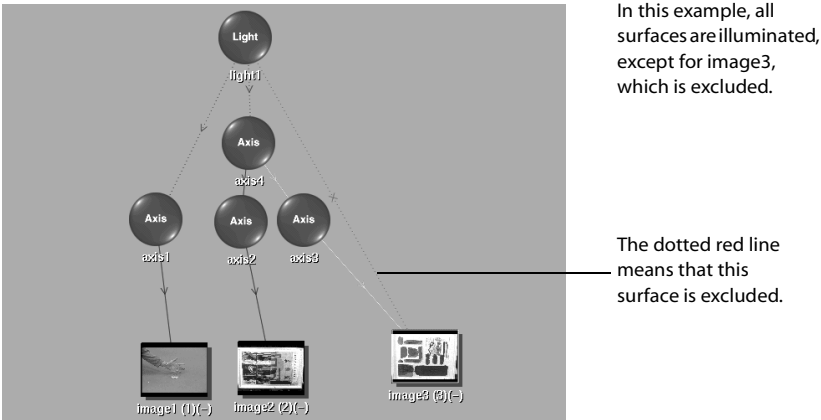


4. Select Lighting from the Edit Mode box.
5. Parent the light source to the axis for the first surface and the global axis.
All surfaces in the selected branches are illuminated.



6. Exclude a surface by holding the **ALT** key while clicking and dragging from the light source to the surface you do not want illuminated.

Excluded surfaces are connected to the light source by a red dotted line with an “X”, and they are not illuminated.



Working with Surface Lighting

After you set up the parent-to-child relationships between lights and surfaces, you can set specific surface lighting properties. You can control a surface’s lighting and adjust the surface’s specular highlight in the Surface Image menu.

Process

Layers

Camera

Surface

Axis

Light

Text

Image

Specular

100.0

100.0

100.0

Shine 0.0

Diffuse

S

Track

Specular Highlight controls

Transp 0.0

Blend

Displacement

Softness 0

Offset 0.00

3D

Flip button

Flip

Filtering

Off

X 0.00

Y 0.00

Z 0.00

Reset Shape

TX 0.00

TY 0.00

Z 0.00

Lighting box

The Surface Image controls are described in the following table.

Select:	To:
Specular highlight fields	Change the specular colour. In descending order, fields represent red, green, and blue channel values. Alternatively, you can click the colour bar next to the fields and use the colour picker to pick specular colour.
	NOTE: The specular highlight is visible only if Shading is enabled in the Setup menu and if shine is greater than 0.

Select:	To:
Shine field	Change the intensity of the specular highlight. A small Shine value produces an intense highlight while a large Shine value produces a dim highlight.
Lighting box	Apply diffuse or ambient lighting to a surface. To use diffuse or ambient lighting, enable Shading in the Setup menu.
Flip button	Flip the normals of the surface so that light is applied to the opposite side of the surface.

For information on a surface's transparency, offset, and other material properties, see "Working with Surfaces" on page 921. For information on using 3D displacement on a surface, see "Displacement Mapping" on page 999. For information on using four-point tracking, see "Four-Point Tracking" on page 801.

Adjusting Specular Highlights

A specular highlight is a reflection of a light source. The position of the specular highlight depends on the position and number of the light sources surrounding a surface and the angle of the camera.

Use the Shine field to change the intensity of the specular highlight. When the Shine value is set to 0, the specular highlight is disabled. To change the size of the specular highlight, use the Falloff field in the Light menu.



This surface is lit using a Falloff of 30 and a Spread of 27. The surface's Shine is set to 10.



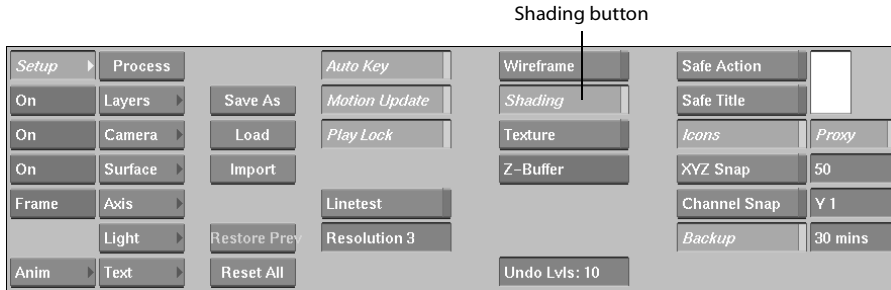
This surface is lit using a Falloff of 20 and a Spread of 27. The surface's Shine is set to 1.

By default, the specular highlight is the same colour as the light source. However, you can change the colour of the specular highlight by changing the specular colour values.

The specular colour is the colour of light that is reflected by the surface. For example, if the specular colour is red and the light source is white, then the specular highlight is red. If the specular colour is yellow and the light source is red, then the highlight is orange.

To use a specular highlight:

1. Add and position a light source in the scene.
2. In the Setup menu, enable Shading.



3. In the Surface menu, select Image to open the Surface Image menu.
4. Using the specular highlight field, set the specular colour.
5. Set a value in the Shine field.

Applying Ambient and Diffuse Lighting

You set how a surface reflects incidental light by applying ambient or diffuse lighting. With ambient lighting, all parts of a surface that are not directly illuminated receive indirect light. Diffuse lighting reflects light equally in all directions, producing a flat reflection on the object. The actual colour of the reflection depends on both the colour value of each pixel and the colour of the incidental light.

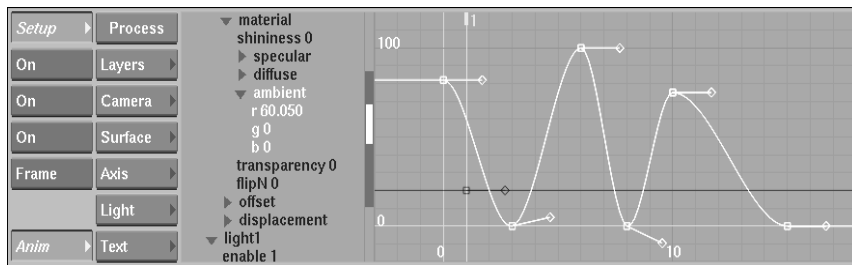


When you turn shading on, you do not have to enable a light source to see the lighting effect: a default infinite light source supplies ambient light at 20% intensity. The infinite light source is located behind the camera eye and cannot be moved. As soon as you add a light source, the infinite light source is replaced by the new light source.

To use ambient lighting:

1. In the Setup menu, enable Shading and disable Texture.
2. In the scene, select the surface to receive ambient lighting.
3. In the Surface menu, select Image from the Surface box.
4. In the Lighting box, select Ambient.
5. Optional: Adjust the colour of the ambient light using the ambient RGB channels in the surface's material folder of the Channel Editor.

To display the RGB channels, click Anim to display the Channel Editor. Expand the surface's folder (it should already be selected), expand its Material folder, then expand the Ambient folder.



With diffuse lighting, the intensity of the reflection depends on the orientation of the light source relative to the surface; it is greatest where the incident light strikes the object perpendicular to its surface. The intensity of the reflection is independent of the camera eye position.

To use diffuse lighting:

1. In the Setup menu, enable Shading and disable Texture.
2. In the scene, select the surface to receive diffuse lighting.
3. In the Surface menu, select Image from the Surface box.
4. In the Lighting box, select Diffuse.
5. Optional: Adjust the colour of the diffuse light using the diffuse RGB channels in the surface's material folder in the Channel Editor.

Flipping a Surface's Normals

When you flip a surface's normals, light is applied to the opposite side of a surface. Use this feature to create a two-sided shaded surface.

To create a two-sided surface:

1. Add a surface and a light source.
2. In the Setup menu, enable Shading.
3. Copy the surface and its axis using the branch option in Schematic view. See “Copying Objects” on page 961.
4. Select the surface of the copied branch and in the Surface menu select Image from the Surface box.
5. Enable Flip.

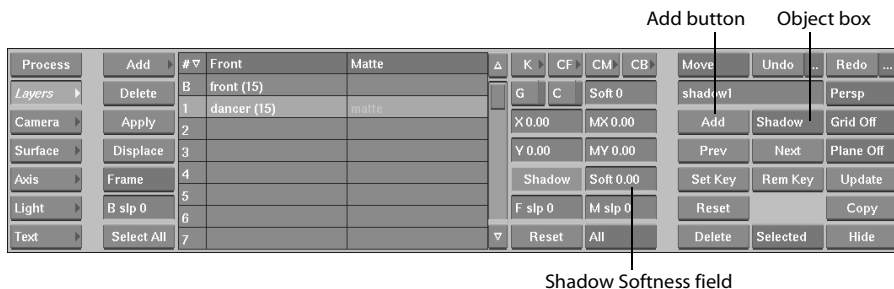
To control both surfaces, parent them by a new axis and use this axis to rotate, scale, shear, and move both surfaces. There may be a priority problem causing one surface to be drawn over the other. To correct this problem, you could use the Priority Editor to animate the drawing priority of surfaces, or change the Z position of one surface by one pixel.

Adding Shadows

In Action, shadows are not cast by lit objects; shadows are cutouts based on a surface's matte. This type of shadow is referred to as a *drop shadow*. A drop shadow can be fully opaque or slightly transparent to simulate a real shadow.

To add a drop shadow:

1. Select the surface that you want to use.
2. In the Object box, select Shadow.
3. Click Add.



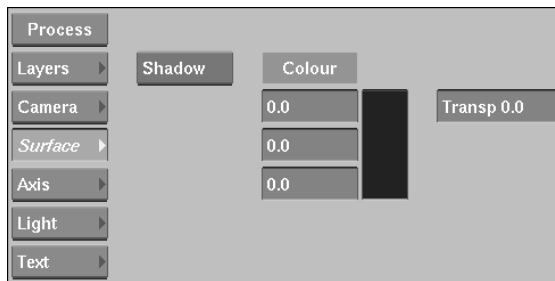
Adjusting the Shadow Softness

You can adjust the softness of a shadow using the Shadow Softness field. Use a value of 0 to have a distinct shadow. Increase the value to soften the shadow.

Shadow softness is a property of the layer. If you add more than one shadow for the same layer, all shadows are softened by the same amount.

Adjusting the Shadow Colour and Transparency

Shadow colour and transparency controls are found in the Shadow menu of the Surface menu. To display the Shadow menu, select Shadow from the Surface box.



You can change the colour of the shadow using either of the following methods:

- Enter colour values in the three fields in the Shadow menu. These fields specify, in descending order, the red, green, and blue channel values.
- Click the colour bar next to the fields. The colour picker appears. Use the colour picker to pick the shadow colour. See “The Colour Picker” on page 57.

You can also adjust the transparency of the drop shadow using the Transp field. When this field is set to 100, the shadow is completely transparent. When set to 0, the shadow is completely opaque.

Moving the Shadow

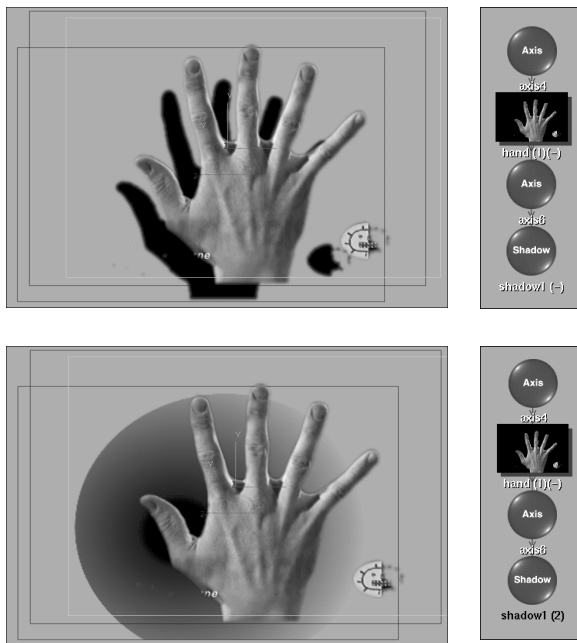
The shadow has its own parent axis. It can be moved, rotated, scaled, and sheared independently of its parent surface. Because the shadow is also the child of the surface, moving the surface axis also moves the shadow.

Changing the Layer Used for the Shadow

You can generate some interesting effects by changing the layer used to generate the shadow.

To change a layer's shadow:

1. Add a shadow to an image surface of Layer1. This shadow, named Shadow1, is the matte of the hand.
2. Select Layer2 in the Layers list.
3. Select Shadow1 in Schematic view.
4. Click Apply. The matte of Layer2 is applied to Shadow1, so Shadow1 becomes the gradient.

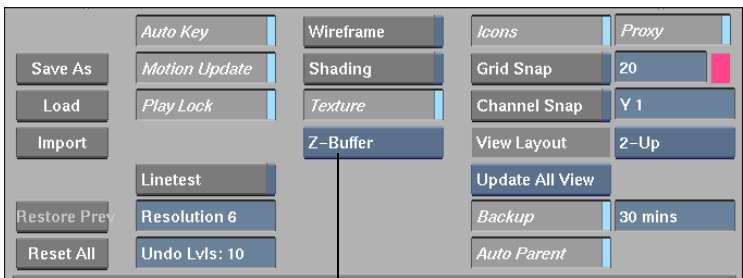


The new layer is used for all shadows generated by that shadow node. You can reset this override by selecting Node in the Reset box and clicking R. See “Reset button and box” on page 909.

The number of the matte that is applied to the shadow is shown with the name of the shadow in Schematic view. Suppose that you add a shadow to a surface of Layer1 in the Layers list. If the default layer is used, (—) appears next to the name of the shadow node. If you apply Layer2 to the shadow, (2) appears next to the name of the shadow node indicating that Layer2 has been applied to the shadow.

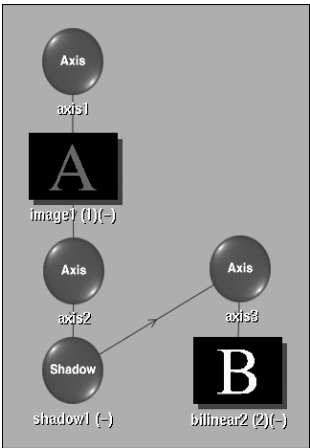
Using Shadow Mix

Use Shadow Mix in the Z-Buffer box to render shadows and surfaces in hierarchical order, as shown in Schematic view. This option makes it possible to have the shadow of one layer on top of the shadow of another layer.



Z-Buffer box

For example, in the following figure, Image1 (a surface of Layer1) is the parent of its shadow, and the shadow is the parent of Bilinear2 (a surface of Layer2). If you preview this setup with Z-Buffer on, Bilinear2 is rendered in front of the shadow. But, if you render with Shadow Mix on, the shadow is rendered in front of Bilinear2.





Setup rendered with Z-Buffer on



Setup rendered with Shadow Mix on

If a surface with a shadow is rotated until the shadow covers its own surface, blending problems will occur. This cannot be corrected with the Priority Editor or with Shadow Mix because the shadow is hierarchically lower than its surface.

To fix blending problems:

1. Copy the surface/shadow branch.
2. Hide the surface in one of the branches.
3. Hide the shadow in the other branch.
4. Change the priority of the shadow and surface until the blending problem no longer occurs.
See “Reordering Overlapping Surfaces” on page 928.

46

Action: 3D Geometry

The “Z” factor

Action offers a 3D representation of world space. Use this space to work with 3D models.

Summary

In this chapter, you learn about:

- “Importing 3D Models” on page 983
- “Creating 3D Text” on page 985
- “Working with 3D Geometry” on page 989
- “Working with Textures” on page 992
- “Displacement Mapping” on page 999
- “Extended Bicubics” on page 1002
- “Using the Magnet” on page 1006
- “3D Deformations” on page 1011

About 3D Geometry

In Action, 3D Geometry consists of 3D models and 3D text. A powerful feature of Action is its ability to import 3D models created in other software applications. You can animate 3D models and 3D text with textures and layers in the scene to produce a variety of effects.

Importing 3D Models

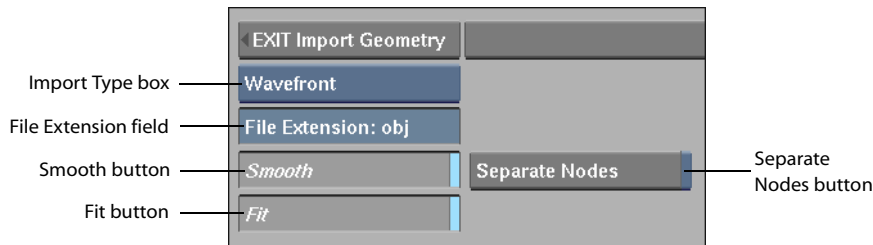
In Action, you can import 3D polygon objects such as **3d studio max**® files, Wavefront files, Inventor files, and Paint geometry. **3d studio max** files contain object data, specifically, texture and materials.

Wavefront files are a set of output files that contain descriptions of an object. You can import geometry data files in Wavefront format to composite a composite, and you can export keyframe animation data for use with products that render Wavefront.

The Inventor file format supports complete descriptions of 3D scenes with rendered polygon objects, lighting, materials, ambient properties, and realism effects. It also defines a standard file format for exchanging 3D data among applications. **flame** only reads the polygon geometry.

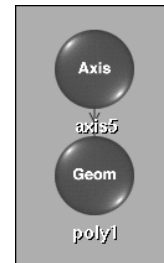
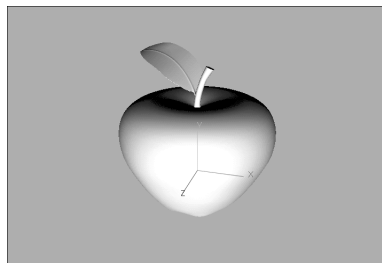
Paint geometry files are created by **flame**'s Paint module. If you want to work with polygon geometry in Action, import Paint geometry. Action ignores its animation and attributes, such as its colour, outline, and gradient. For example, in Paint, if you create a blue polygon, animate its scale, and save it as geometry, it is imported in Action as a white polygon with no animation.

You import 3D models into Action through the Import Geometry menu. To access the Import Geometry menu, click Setup in Action and then click Import.



The Import Geometry controls are described in the following table.

Select:	To:
Import Type box	Select the 3D model type you want to import: Wavefront, Inventor, 3DStudio, or Paint. When you select Paint as the import type, the subsequent controls described in this table do not apply.
File Extension field	Specify a particular file extension for Wavefront, Inventor, or 3DStudio files. By default, Wavefront files have <i>.obj</i> , Inventor files have <i>.iv</i> , and 3D Studio files have <i>.3ds</i> as their file extensions.
Smooth button	Force Action to build the normals for the 3D model. Enable this button if you are importing polygons that do not have normals.
Separate Nodes button	Create individual nodes for all 3D models contained in a file. When this button is disabled, the 3D model is added to the scene with its own axis.



Select:	To:
Fit button	Scale the imported model to fit into the current frame. When disabled, the imported model maintains the same size in which it was created using a third-party application.
Rotate Axis button	Rotate the imported 3D model by 90° on the X-axis so that it is compatible with Action's coordinate system. This button appears only when 3DStudio is selected as the import format.

To import a 3D model:

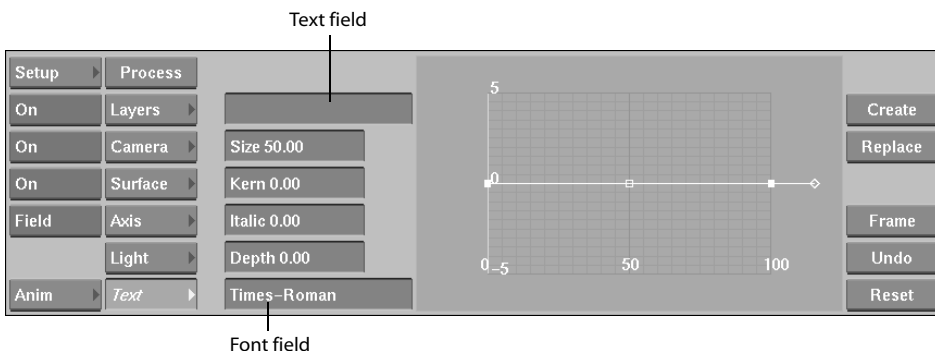
1. Click Setup, and in the Setup menu, click Import.
The Import Geometry menu and file browser appear.
2. In the Import Type box, select the import format.
3. If the file you want to import has a file extension different than the one specified, type a file extension.
4. Enable Smooth or Fit as needed.
5. Select the file to import from the file browser.
The 3D model is added to the scene.

You can change the 3D model's colour, specular highlight, shine, and other material properties. See "Working with 3D Geometry" on page 989.

Creating 3D Text

You create 3D text using the Text menu in Action. 3D text strings have the same properties as 3D imported objects. In addition, with 3D text, you specify typical text properties such as font, font size, kerning, and italics. In 3D space, text has depth and can be rotated on the X-, Y-, or Z-axis.

You can create 3D text strings using Type 1, TrueType, or Asian fonts. You set 3D text properties using the 3D Text Editor. To access the 3D Text Editor, click Text.



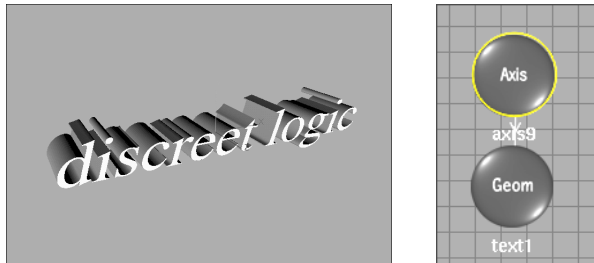
The 3D Text Editor controls are described in the following table.

Select:	To:
Text field	Type the characters that make up the text string. When keyboard preference is enabled, clicking this field displays the on-screen keyboard representing the character set for the selected font. Enable Up ASCII to access the rest of the character set. See “Setting the Keyboard On or Off” on page 119.
Size field	Specify font size for the characters in the text string.
Kern field	Set kerning for the characters in the text string.
Italic field	Italicize the characters in the text string.
Depth field	To extrude the characters in the text string, making it three dimensional.
Bevel curve	Apply a bevel to the depth of the text string by manipulating the Bevel curve. You can move and add points to the curve, as well as adjust the tangent handles to produce different effects with the text string.
Font field	Access the font library and select a font for the text string. In the font library you can select from Standard, TrueType, and Asian fonts. By default, Action uses the font specified for the TextDefaultFont token in the <i>init.cfg</i> configuration file. For example: TextDefaultFontTimes-Roman See “Using New Fonts” on page 878 and the <i>flame Installation Guide</i> .
Create button	Place the contents of the Text field in the scene.
Replace button	Apply changes to the 3D text string.
Frame button	Reset the Bevel curve viewer to show the whole curve.
Undo button	Undo Bevel curve operations.
Reset button	Reset the Bevel curve.

To create 3D text:

1. In Action, click Text.
2. Enter a text string in the Text field.
3. To change the font, click the Font field to select a new font from the font library.
4. Set font size, kerning, italics, and depth in the corresponding fields.
5. Click Create.

The 3D text string appears in the scene, and a Geom object with its parent axis appear in schematic.

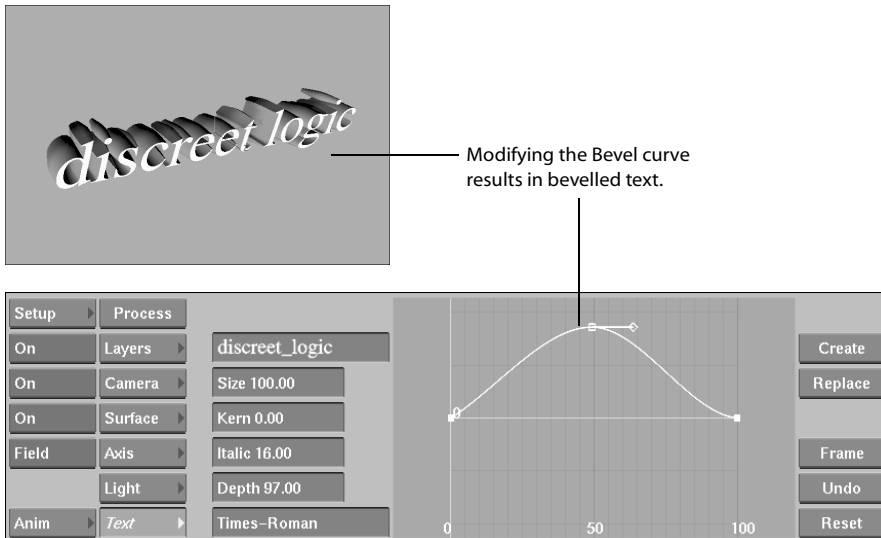


6. Click the Geom node to provide a descriptive for it.

Creating Bevelled Text

Use the Bevel curve to add a bevelled edge to your 3D text. Use the options in the Edit Mode box to add, select, delete, or move keyframes on the Bevel curve. The Bevel curve behaves in much the same way as an animation curve in the Channel Editor.

Experiment with different curves to create different effects.



You can save Bevel curve settings as well as font, character size, kerning, italics, and depth. See “Saving, Loading, and Importing Setups” on page 899.

Changing 3D Text Properties

As you work with objects in the scene, you can edit 3D text properties by first selecting the Geom object in the schematic.

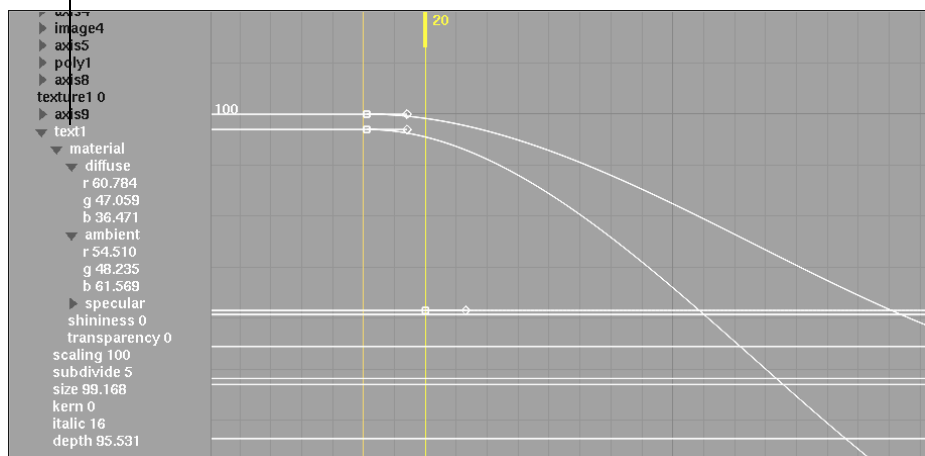
To edit 3D text parameters:

1. In Schematic view, select the Geom object representing the 3D text string.
2. In the Text menu, modify the 3D text string or its parameters accordingly.
3. Click Replace to apply your changes.

Animating 3D Text

With 3D text strings, you can animate the 3D text channels and geometry channels in the Channel Editor. However, neither the text string nor its Bevel curve can be animated. The 3D text animation channels are contained in the text folder as shown in the following figure. See “Animation” on page 127.

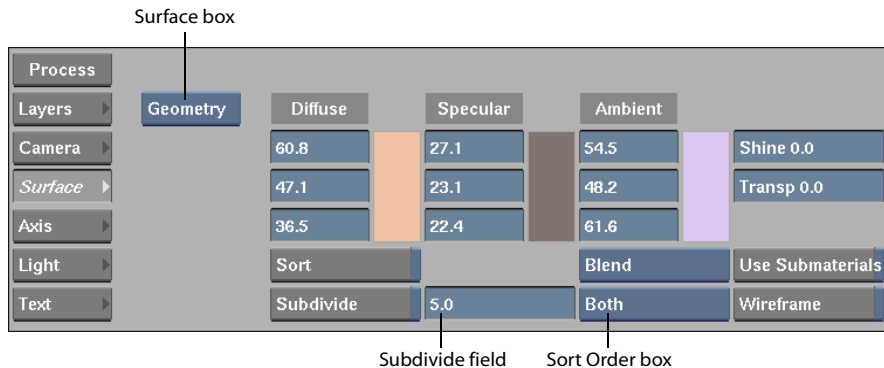
This is a default name. If you rename the Geom for the 3D text, its new name appears here.



You can create 3D text deformation effects by applying a Deform mesh to the 3D text string. See the *flame Tutorial* for specific 3D text lessons.

Working with 3D Geometry

You can change and animate the colour, specular highlight, shine, transparency, and other parameters of 3D models and 3D text. You set these parameters through the Surface Geometry menu. To access this menu, select Geometry in the Surface box of the Surface menu.



The Surface Geometry controls are described in the following table.

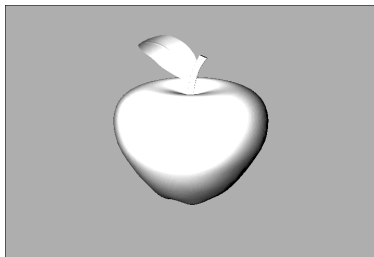
Select:	To:
Diffuse	<p>Modify the colour and illumination of the entire 3D model. Diffuse light mixes with the colour of the light sources used to illuminate the 3D model. The diffuse colour may also mix with the ambient colour and the colour of the specular highlight.</p> <p>Change the colour of the 3D model by entering values in the Diffuse red, green, and blue channel fields or using the colour picker. See “The Colour Picker” on page 57.</p>
Specular	<p>Set the colour of light reflected by the 3D model’s surface. To enable the specular highlight, the Shine value must be larger than zero. Change the specular colour by entering values in the Specular red, green, and blue channel fields or using the colour picker.</p> <p>For example, if the specular colour is red and the light source is white, the specular highlight is also red. If the specular colour is yellow and the light source is red, the highlight is orange.</p>
Ambient	<p>Set colour to the area of the 3D model that is not illuminated by a direct light source.</p> <p>The edge of the ambient area mixes with the specular highlight colour and the diffuse colour. Change the Ambient colour by entering values in the Ambient red, green, and blue channel fields or using the colour picker.</p>
Shine	<p>Set the intensity of the specular highlight. When this value is zero, the specular highlight is disabled.</p> <p>Shine affects both size and intensity.</p>
Transp field	Increase or decrease the transparency of the 3D model or text.

Select:	To:
Sort button	Sort the drawing priority of the 3D model normals either back to front or front to back. See "Sort Order box" in this table.
Subdivide button and field	Create high quality shading for polygon models. Use the Subdivide field to enter the maximum polygon side length.
Add/Blend box	Use a regular blend to composite the object in the scene. Alternatively, you can use additive blend to add the colours of the object to the colours of what lies behind it. You may have to change transparency to see the effect.
Sort Order box	Determine how the 3D model or text is drawn according to its normals. Use the options as follows: <ul style="list-style-type: none"> • Both Draw the polygons that are both facing and opposite the camera. • Front Draw the polygons facing the camera last. • Back Draw the polygons opposite the camera last.
Use Submaterials	Use materials imported with 3d studio max models only.
Wireframe	View your model or 3D text as a wireframe outline only. When you render the 3D model or text with Wireframe enabled, it retains its light, shading, and texture attributes.

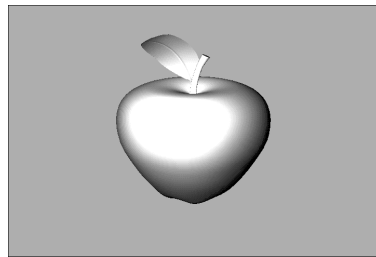
To use a specular highlight with a 3D model:

1. Add and position a light source in the scene.
2. In the scene, select the 3D model to which you want to add the highlight.
3. In the Setup menu, enable Shading and disable Texture.
4. In the Surface menu, select Geometry from the Surface box.
5. In the Shine field, set a value.

A large Shine value produces a dimmer highlight while a low Shine value produces an intense highlight. When the Shine field is set to zero, the 3D model does not have a highlight. The following example shows the same 3D model using two different Shine values.



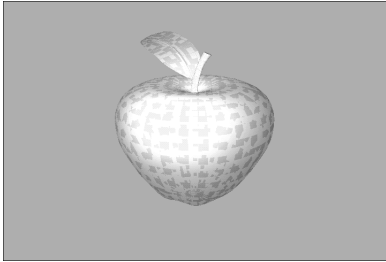
Shine is set to 1. The specular highlight is intense.



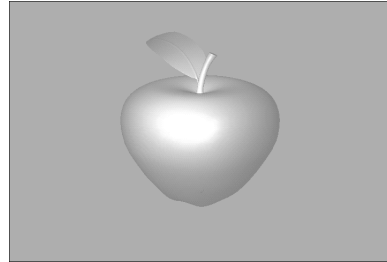
Shine is set to 20. The specular highlight is dimmer.

Using Transparency and Sort

When using transparency with 3D models, you may sometimes see the back polygons, giving the model a shattered or broken look. This happens when the drawing priority of the normals is not sorted properly. Enable Sort and select an option from the Sort Order box to sort the drawing priority back to front or front to back.



Drawing order of the 3D model's polygons is incorrect. Back polygons are drawn through when the model is transparent.

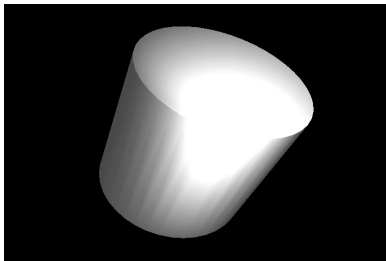


Select Front to sort the polygons front to back. The transparency is drawn correctly.

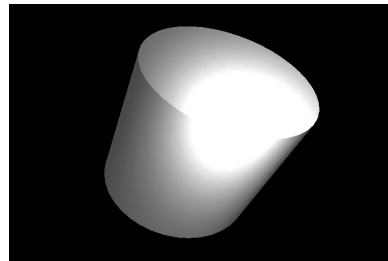
Using Subdivide

The Subdivide feature is useful for creating precise highlights and spotlights. The effect of Subdivide is apparent only after you process the clip or when you click Preview in the Action menu.

The polygons in the 3D model are subdivided according to the value you specify in the Subdivide field, resulting in smoother surfaces on the model.



Spotlight on a polygon model rendered with Subdivide off.



Spotlight on a polygon model rendered with Subdivide set to 2.

NOTE: The smaller the number in the Subdivide field, the slower the rendering.

You can animate the Subdivide channel in the Channel Editor; however, expect a longer rendering time when the Subdivide value changes over several keyframes.

Working with Textures

You can apply a texture to a 3D model or 3D text. You can move, rotate, scale, and animate the texture independently from the 3D model or 3D text. Since the texture and its axis are parented by the geometry node, animating the geometry's axis also animates the texture, which has the effect of keeping the texture properly in place on the model. A texture uses the specular highlight, diffuse colour, and shine that is set by its parent.

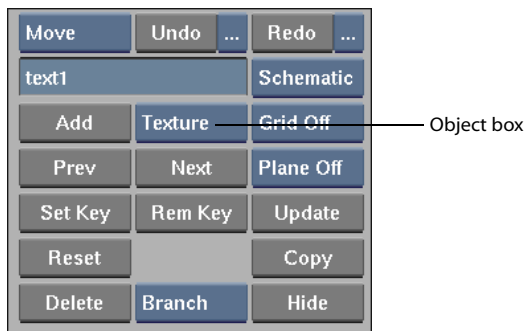
You can project a texture in 3D space using a Projector node, to which you can apply blend and additive effects. See “Projecting Textures” on page 996.

To apply a texture:

1. In the schematic, select the 3D model or 3D text to which you want to apply the texture.
2. In the Layers menu, select the layer you want to use for the texture.

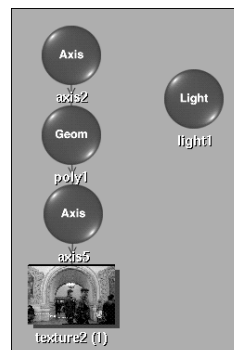
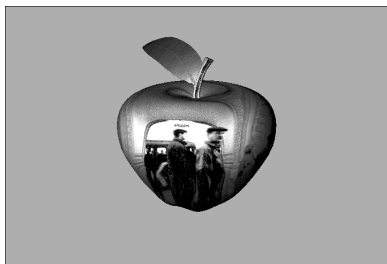
If the texture does not use a matte, you should turn off the matte clip; otherwise, the texture will not render properly. See “Turning Clips On and Off” on page 911.

3. In the Object box, select Texture.



4. Click Add.

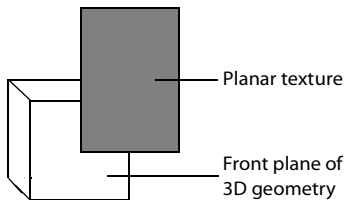
The texture object is added to the schematic with its own parent axis. The new axis is the child of the selected 3D model.



5. In the Surface menu, select Texture from the Surface box.
The Surface Texture menu appears.
6. Select the texture and set texture properties as described in the next section.

Understanding Texture Mapping

When you apply a texture, Action lines up the lower left corner of the texture (coordinates 0, 0) with the centre point of the front plane of the geometry. For example, the following figure shows how Action initially aligns a planar texture to a cube.

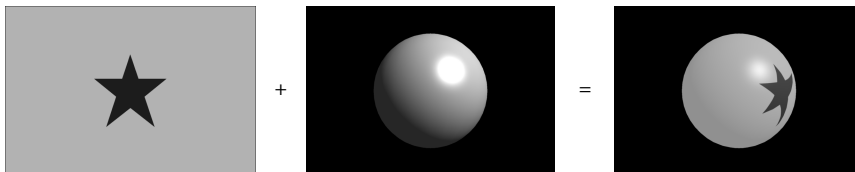


Because a texture is added with its own axis, you can use this axis to offset the texture from the centre of the geometry using the PosX and PosY fields. You can also rotate and scale the texture.

The initial placement of the texture is most evident when you use planar texture mapping. In Action, you can map textures to 3D models using reflection, planar, wrap, or cylindrical mapping. You select the mapping mode in the Mapping box of the Surface Texture menu.

Reflection

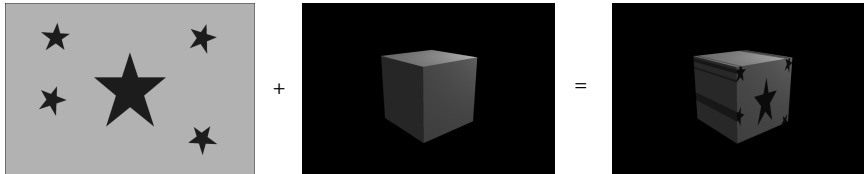
Reflection mapping mode applies the 2D texture as though the 3D model is reflecting the texture. This type of mapping uses the surface normals of the geometry to generate texture values and the texture is reflected onto the 3D model.



Reflection mapping sometimes distorts or scales the texture, since it is wrapped around the entire object. If you move the 3D model, the texture slides on the object.

Planar

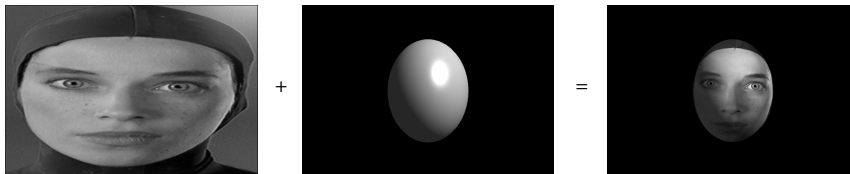
Planar mapping mode applies the texture without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the texture values. Planar mapping positions the lower left corner of the texture at the 3D model's axis.



When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the “sides” of the object.

Wrap

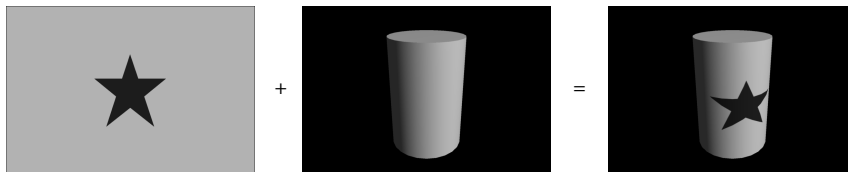
Wrap mapping mode completely envelops the 3D model with the texture according to the object's texture coordinates. To use this option, you must import a model that has its own texture coordinates.



NOTE: If the 3D model has no texture coordinates and you choose wrap mapping mode, the texture reverts to planar mapping mode.

Cylindrical

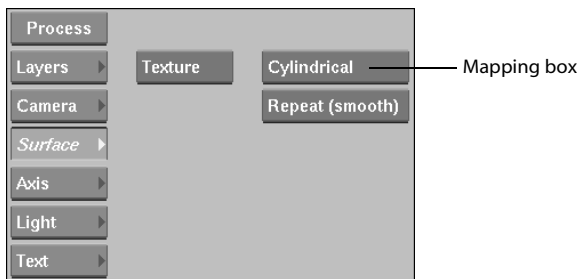
Cylindrical mapping mode wraps the texture around the 3D model in a cylindrical manner, starting from the lower left corner of the texture. Use cylindrical mapping for models of cans, bottles, or other circular and semicircular bodies.



Cylindrical texture mapping is oriented counterclockwise—left-to-right—when viewed from the top of the model.

To change texture mapping:

1. Select the texture to which you want to apply different texture mapping.
2. In the Setup menu, enable Shading.
Enable or disable Shading depending on the method of texture mapping you are using. When Shading is enabled, normals are used. You must enable Shading when using Reflection mapping because it also uses normals.
3. In the Surface menu, select Texture from the Surface box.



4. In the Mapping box, select the texture mapping.
5. Select the repeat mode for the texture.

Select:	To:
No Repeat	Use the texture once, without repeating it over the surface.
Repeat (fast)	Repeat the texture to fill the surface. If the surface is larger than the texture, or you scale down the size of the texture, the texture will repeat to cover the entire surface. The fast option will give a slightly lower quality result, but with faster processing time.
Repeat (smooth)	Repeat the texture to fill the surface. If the surface is larger than the texture, or you scale down the size of the texture, the texture will repeat to cover the entire surface. The smooth option will give better results but with slightly longer processing time.

Projecting Textures

Like a slide projector, textures can be projected onto 3D models and 3D text to create an effect where a texture is animated on an object in 3D space. The advantage of using a Projector node is that you can cast images on arbitrary surfaces. You can create a spotlight texture and project it onto a 3D model, so that you can see “into” it.

The texture projection properties are located in the Surface Projector menu. To access the Surface Projector menu, select Projector from the Surface box.

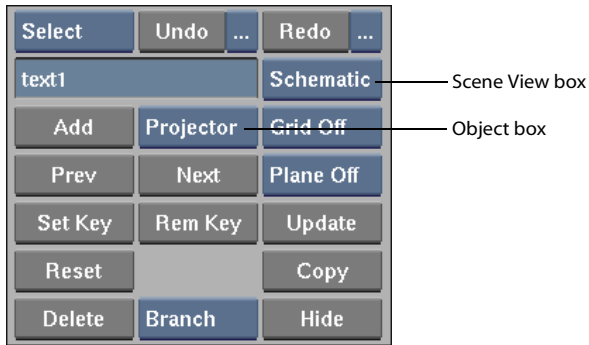


The Surface Projector controls are described in the following table.

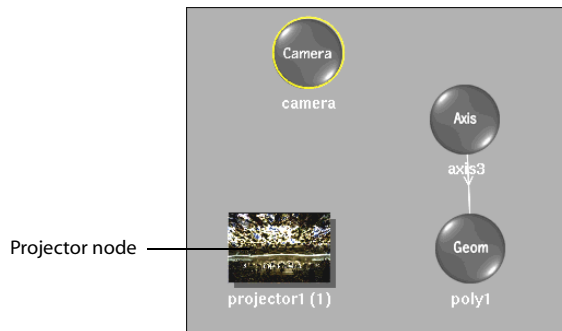
Select:	To:
PosX, PosY, and PosZ fields	Position the projector in 3D space. You can also set the texture projector position by dragging the projector in the scene.
RotX, RotY, and RotZ	Rotate the projector in 3D space.
Enable button	Activate or deactivate the projected texture effect in the scene. When this button is disabled, the Projector node remains in the schematic and the projector icon is reduced in the image area.
Transparency field	Modify the transparency of the projected texture effect. Transparency values are 0 to 100.
FOV field	Adjust the projector's field of view. FOV values are 1.00 to 180.00.
Aspect field	Specify the ratio of height to width for the projected image. Aspect values are 0.10 to 100.00.
RGB colour fields	Specify RGB colour values or pick a colour with the colour picker.
Projection Blend Mode box	Select the projected texture effect. See “Using Projection Blend Modes” on page 998.
Motion Path button	Enable motion path animation for the projector.

To project a texture:

1. Go to the Layers menu and in the Layers list, select the layer that you want to project.
2. In the Object box, select Projector and click Add.

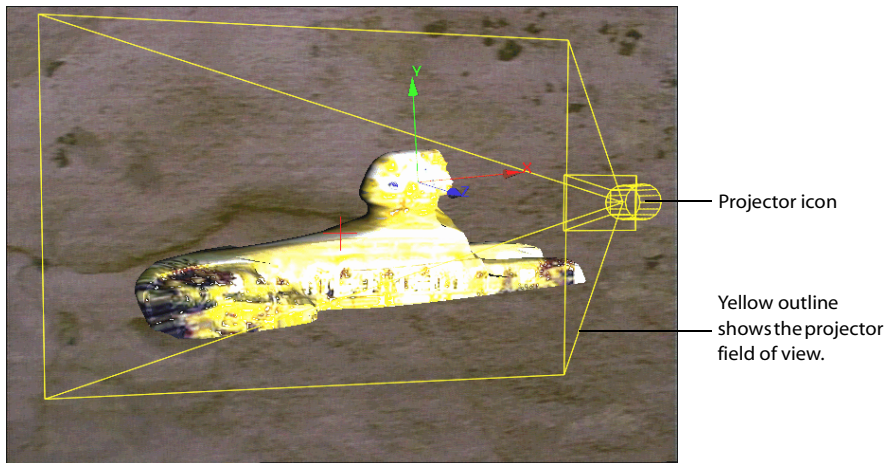


The Projector node appears in the schematic and a projector icon appears in the scene.



3. Double-click the Projector node to display the Projector menu. Alternatively, you can select Projector from the Surface box in the Surface menu.
4. Switch to Perspective view to manipulate the projector in the scene.

The following example illustrates a projected layer on a 3D model using Spotlight projection blend mode. See “Using Projection Blend Modes” on page 998.



Using Projection Blend Modes

You use projection blend modes to modify how you want to blend the projected image in the scene as described in the following table.

Select:	To:
Add	Compensate for the soft or anti-aliased edge on an object in a front and matte clip layer.
Blend	Punch the matte through the front. This blends the edge of the front clip and adds additional softness to the layer. This is the default setting.
Max	Compare the RGB channels of each pixel of the front clip and the back clip individually and return the larger of the two values.
Min	Compare the RGB channels of each pixel of the front clip and the back clip individually and return the smaller of the two values.
Multiply	Multiply the RGB channel values of corresponding pixels of the front clip and the matte clip, and normalize the result by dividing by 255 in 8-bit mode, or 4095 in 12-bit mode. The resulting RGB channel values are assigned to the corresponding pixels in the generated clip. Hardware limitations prevent transparency from working properly with Multiply.
Negate	Create a negative-like result. Try Negate with a soft white matte with a white front clip. This mode does not use the matte or transparency in its blend.

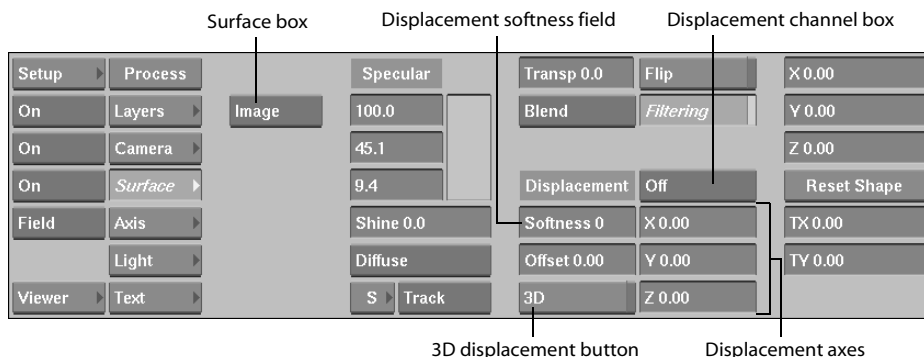
Select:	To:
Screen	Multiply the inverse of the matte clip's colours with the colours of the front clip. The resulting colour is always a lighter colour. Screen does not work with transparency. When using a matte, set the diffuse r, g, b channels to 0, 0, 0 in the Channel Editor for the image surface you are blending.
Simple Add	Punch the matte through the front using more softness than Blend mode. Simple Add mode is similar to Add but includes transparency.
Spotlight	Create a slide projector effect. This mode does not use the matte or transparency and decreases system performance.
Spotlight Blend	Create an effect where black areas of the front disappear. This mode does not use the matte or transparency; it works well with a clip on which an object is surrounded by black.
Subtract	Subtract the RGB channel values of the pixels from the matte clip from the RGB channel values of the pixels from the front clip and assign the result to the RGB channel values of the pixel in the result. If an RGB channel value from matte clip is larger than corresponding channel value in front clip, yielding a negative result, that result is clamped at 0 (black).

Displacement Mapping

Use displacement mapping to create a 3D model from a 2D surface. The values of a selected colour channel in the displacement source clip are used to create a displacement map. When the displacement map is applied to the surface, the pixels of the surface are displaced along the positive or negative X, Y, and/or Z axes. By default, the layer for the selected surface is used as the displacement source.

You can also apply deformations to a selected area of the surface by using the options in the Extended Bicubics menu. See “Extended Bicubics” on page 1002.

Use the Surface Image menu to set the displacement for the selected surface. To access the Surface Image menu, select Image from the Surface box.



The Surface Image displacement controls are described in the following table.

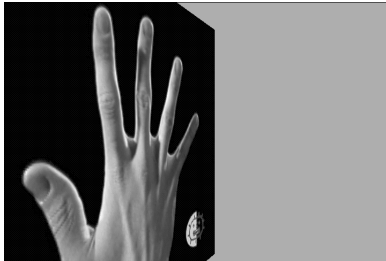
Select:	To:
Displacement channel box	Calculate the displacement map by selecting any of the colour channels: Use R, Use G, Use B, Use Y, Use U, or Use V.
Displacement axes	Specify the amount of displacement in pixel units along the X, Y, and Z axes. Use positive values for displacement on the positive axis, and negative values for displacement on the negative axis.
Displacement softness field	Round off, or soften, the spikes that result from colour values in the image that vary from pixel to pixel in the displacement map. Softness rounds the edges of the displacement. The larger the softness, the smoother the displacement. Softness also affects rendering; the larger the softness, the longer it takes to render. An example of using softness is provided in the following procedure.
Offset field	Apply an offset to the displacement of X and Y.
3D displacement button	Displace bilinear and bicubic surfaces according to their normals. For image surfaces, disable 3D to displace in the X, Y, and Z directions.

To use displacement mapping:

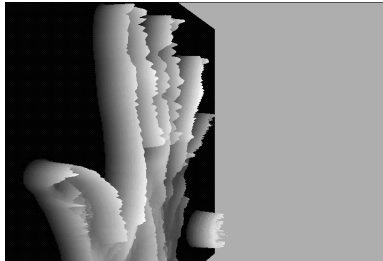
1. In the scene or schematic, select the surface you want to displace.
2. In the Surface menu, select Image from the Surface box.
3. In the Displacement channel box, select a colour. For example, select Use Y to base the displacement on the luminance of the displacement source.
4. In the X, Y, and Z fields, enter displacement values.
5. In the Softness field, set the softness of the displacement.
6. In the Setup menu, adjust the resolution in the Resolution field for a more accurate displacement.

Using a high resolution value results in an inaccurate displacement. A low value yields very good results, but takes longer to process. Typically, you compromise by specifying a high enough value for Resolution while adding softness to make the rough edges smooth.

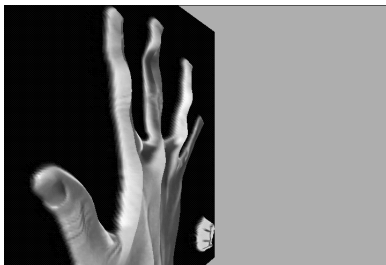
The following figure illustrates a possible use of displacement mapping and shows the difference between displacing with and without softness.



The original image.



Z-axis displacement (60) using the luminance channel (Y) and a softness of 0.



Z displacement (60) using the luminance channel (Y) and a softness of 12.

Using a Different Layer for the Displacement Source

You can specify a different layer as the displacement source.

To use Layer2 as the displacement source for a surface of Layer1:

1. Go to the Layers menu and in the Layers list, select Layer2.
2. Go the schematic and select the surface to which you want to apply displacement mapping.

Notice that in Schematic view, the number and symbol next to the name of the selected surface node is (1)(-). (1) indicates the layer for the surface is Layer1. (-) indicates the default layer is used as the displacement source. The default layer is the layer applied to the surface (also Layer1).

3. In the Layers menu, click Displace.

In Schematic view, the two numbers next to the name of the selected surface node change to (1)(2). (2) indicates Layer2 is used as the displacement source.

4. You can reset this override by selecting Node from the Reset box, then clicking Reset and Confirm.



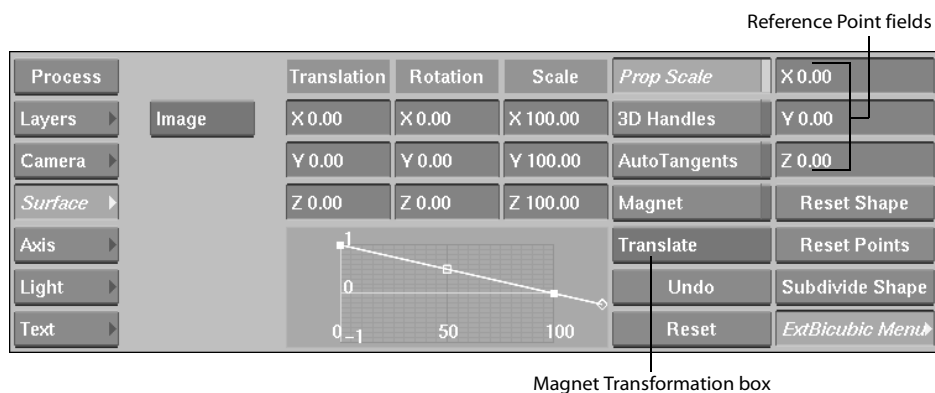
NOTE: Displacement mapping uses the layer's matte clip, so make sure you turn the matte on or off to get the right effect.

Extended Bicubics

In Action, use an extended bicubic surface to warp specific areas of the surface. An extended bicubic surface can be subdivided in many sections that are controlled by vertices, allowing you to bend and contort a surface. The vertices are joined using bicubic interpolation—curved lines.

You can animate the shape of a bicubic surface by changing the position of the vertices and moving the vertex tangent handles. See “Using the Shape Channel or Vertex Channels” on page 926.

To use extended bicubics, add an ExtBicubic node to the schematic and use the Extended Bicubics menu to add and transform vertices. To access the Extended Bicubics menu, double-click the ExtBicubic node in the schematic.



By default, the extended bicubic surface has four vertices and eight tangents. You can translate, scale, or rotate these points and tangents individually or as a group. You can also subdivide the

bicubic surface to increase the number of vertices and tangents and translate the vertices for a smaller region of the image for more precision.



Extended Bicubics have three types of points: vertices, tangents, and the reference point. Vertices and tangents are collectively called *surface points*.

Vertices lie on the surface and its tangents determine the curvature of the deformation at each vertex. The reference point indicates the axis of origin for applying rotation and scaling to surface points.

Use the surface points to control the deformation of the surface. Each vertex has tangent handles. By moving the tangents and their handles, you can warp specific areas of the surface. You can move tangents and handles individually, or as a group.

Tangent handles have three modes.

This mode: **Indicates:**



The handles are not broken. If you move this handle, the adjacent handle will move in the opposite direction and by the same magnitude (length).



The handles are broken. If you move this handle, no other handles are affected.

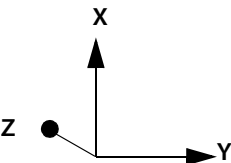


The handles are geometrically continuous. If you move this tangent, the adjacent handles will move in the opposite directions but maintain its magnitude (length).

NOTE: If you move a tangent explicitly, it is not affected by Auto Tangents. Click Reset Points or Reset Shape so that the tangents will be affected by the Auto Tangent mode.

The Extended Bicubic controls are described in the following table.

Select:	To:
Translation X, Y, Z fields	Move selected surface points along the X,Y or Z-axis. Hold down CTRL and select surface points to translate them.
Rotation X, Y, Z fields	Rotate selected surface points along the X,Y or Z-axis. Hold down CTRL and select surface points to rotate them.
Scale X, Y, Z fields and Prop Scale button	Scale selected surface points along the X,Y or Z-axis. Hold down CTRL and select surface points to scale them. Enable Prop Scale to scale surface points proportionally.
3D Handles button	Enable Z buffering of the vertices. By default, the vertices for the extended bicubics are always visible, regardless of their position in Z space in relation to other layers.
AutoTangents button	Scale adjacent tangents automatically. Auto Tangent creates a smooth curve between points in the deformation. If you want to work on a specific area of the image without affecting other tangents, disable Auto Tangent.
Magnet button	Transform selected vertices and tangents numerically. Use in conjunction with the Magnet Transformation box.
Magnet Transform- ation box	Select the type of transformation to use when Magnet is enabled. See "Using the Invisible Magnet" on page 1010.
Reference Point X, Y, Z fields	Set the location of the reference point. Use the reference point to constrain the rotation and scaling of an individual or group of vertices. By default, the reference point appears in the centre of the extended bicubic. When unselected, the reference point is green.



Reset Shape	Reset the shape of the extended bicubic surface.
Reset Points	Reset selected points on the extended bicubic surface.
Subdivide Shape button	Subdivide extended bicubic surface into more sections. You should warp surface with a small number of subdivisions to obtain the best results. After applying some deformations to a large portion of the surface, subdivide the surface further and perform deformations on a more localized region of the surface.
Undo button	Undo a change to the Magnet Curve Editor.
Reset button	Reset the Magnet Curve Editor.

To subdivide extended bicubics:

1. Go the Layers menu and select the layer to which you want to add an extended bicubic surface.
2. From the Object box, select ExtBicubic and click Add.
3. Go to the Surface menu and select Image from the Surface box.
4. In the Surface Image menu, click ExtBicubic Menu.
The Extended Bicubic menu appears.
5. Click Subdivide Shape.
6. Transform the tangents to achieve the effect you want.

To move the reference point:

1. Select the reference point.
The reference point turns red when selected.
2. Drag the reference point to a new location, or use the X, Y and Z fields to assign a new coordinate for the reference point.

To rotate multiple surface points:

1. Set the reference point values you want to use as the axis of origin.
2. Select multiple surface points by pressing **CTRL** and dragging to select the surface points.
3. From the Edit Mode box, select Rotate.

Edit Mode box



A 3D track ball appears on the reference point.

4. Use the 3D track ball to rotate the reference point and selected points.



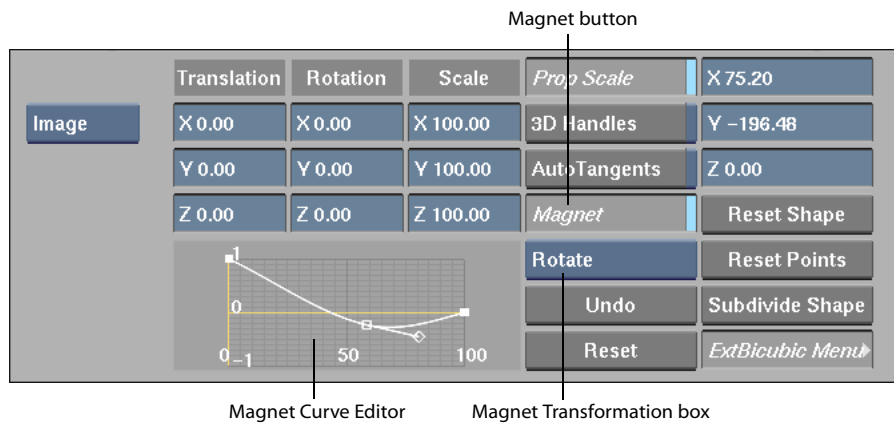
All selected points rotate around the reference point.

To scale multiple surface points:

1. Set the reference point values you want to use as the axis of origin.
2. Select multiple surface points by pressing **CTRL** and dragging to select the surface points.
3. From the Edit Mode box, select Scale.
4. Drag in a direction in the image window to scale accordingly.

Using the Magnet

Use the magnet to select and transform a range of extended bicubic surface points. Use the magnet when there are many surface points as a result of more than one subdivision. On the Extended Bicubic menu, the magnet controls include the Magnet button, the Magnet Transformation box, and the Magnet Curve Editor.



There are two ways to transform points with the magnet. You can use the magnet to select a range of points and transform them gesturally. Or, you can use the invisible magnet (enable the Magnet button) in conjunction with the Translation, Rotation, and Scale fields to transform selected points numerically.

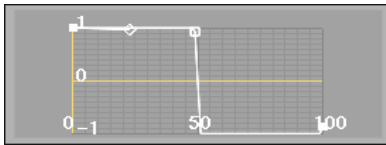
Before you can use the magnet, you must set the polarity of the magnet and the magnet area of focus. The magnet's area of focus is determined by where you click on the surface and the polarity of the magnet is set by the Magnet Curve Editor. The radius is determined by the distance from the centre to the farthest selected point.

Setting the Magnet Polarity

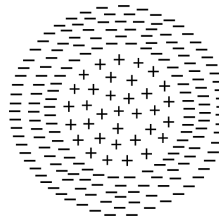
Use the Magnet Curve Editor to define the weighted polarity from the centre to the edge of the magnet. Points are either attracted or repelled depending on their location in the magnet and the shape of the magnet curve.

The area of the magnet is plotted on the X-axis where 0 is the centre of the magnet and 100 is the edge of the magnet. The polarity is plotted on the Y-axis where 1 is maximum positive strength and -1 is maximum negative strength.

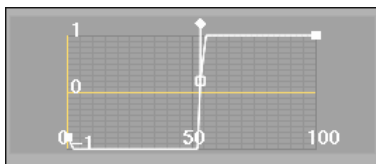
A positive polarity value attracts nearby points. A negative polarity value repels nearby points. The following example illustrates a magnet that has a strong positive polarity near the centre and strong negative polarity near the edges.



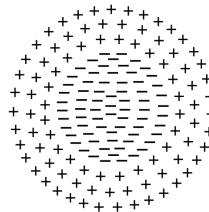
Magnet Curve Editor



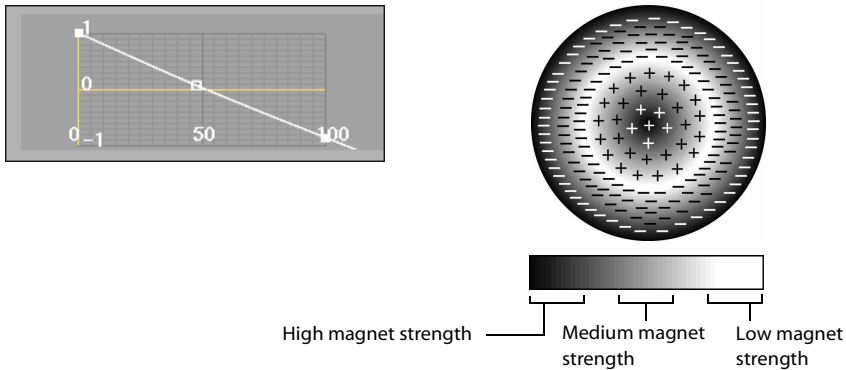
The following example illustrates a magnet that has a strong negative polarity near the centre and a strong positive polarity near the edges.



Magnet Curve Editor



The following example illustrates a magnet whose positive polarity decreases from the centre and whose negative polarity increases toward the edges.



When you transform points, the direction and size of the transformation is determined by the weighted polarity of the magnet at the location of the points.

Using the first example, if you use the magnet to translate the points on the X-axis by +200, the points in the centre of the magnet move in the positive direction on the X-axis—they are attracted to the magnet—while the points near the edge of the magnet move in the negative direction on the X axis—they are repelled by the magnet.

To change the polarity of the magnet:

1. In the Extended Bicubics menu, click Reset to return to the magnet curve to its default.
2. Click the left-most handle on the curve and drag to define the polarity for the centre of the magnet:
 - Drag the point up to increase the positive polarity.
 - Drag the point down to increase the negative polarity.
 - Drag the point to the middle to assign no polarity.
3. Click the right-most handle on the curve and drag to define the polarity for the edge of the magnet:
 - Drag the point up to increase the positive polarity.
 - Drag the point down to increase the negative polarity.
 - Drag the point to the middle to assign no polarity.
4. Click the middle handle on the curve and drag to define the transition of polarity from the centre to the edge of the magnet.

Using Magnet Mode

Use Magnet mode to transform points gesturally. The magnet’s area of focus is determined by where you click on the surface and the radius of the magnet is set by the Magnet Curve Editor. The radius is determined by the distance from the centre to the farthest selected point.

To use the magnet:

- 1. Use the Magnet Curve Editor to determine the polarity for the magnet. See “Setting the Magnet Polarity” on page 1007.
- 2. In the Edit Mode box, select Magnet.



The magnet appears as a red circular outline.

- 3. To resize the magnet, hold **CTRL-S** and drag left or right.
 - 4. In the Magnet Transformation box, select the type of transformation.
- | Select: | To: |
|-----------|--|
| Translate | Translate the selected points along the X-, Y-, or Z-axis. |
| Rotate | Rotate selected points about the centre of the magnet. |
| Scale | Scale selected points about the centre of the magnet. |
- 5. Click the image window to select the points you want to transform.
 - 6. Drag in the image window to apply the transformation to the selected points.

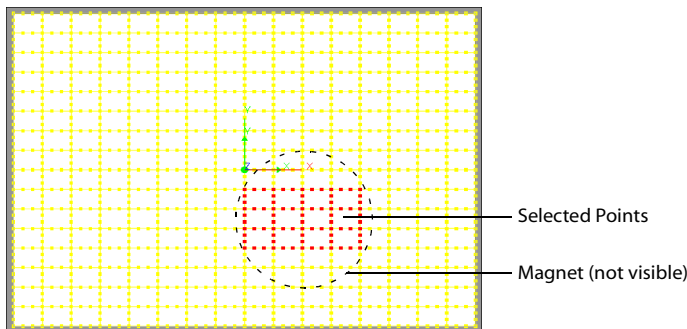
Using the Invisible Magnet

Use the invisible magnet to transform points you cannot select with the magnet. The centre of the magnet is determined by the centre of the selection and the radius is determined by the distance from the centre to farthest selected point. Any transformation to the selected points are affected by the magnet.

To use the Magnet mode:

1. Use the Magnet Curve Editor to determine the polarity for the magnet. See “Setting the Magnet Polarity” on page 1007.
2. From the Edit Mode box, select Move or Select.
3. Hold **CTRL** and drag to select a range of points.
4. In the Extended Bicubics menu, enable Magnet.

The magnet is not visible, but is illustrated here to show how the points will be affected when you transform them.



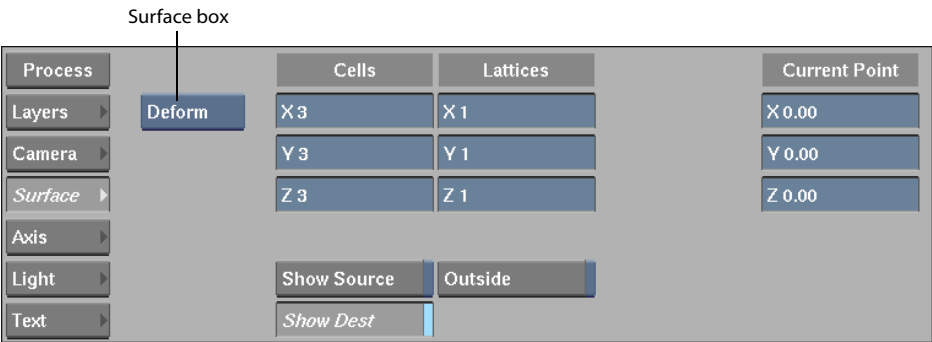
5. In the Magnet Transformation box, select the transformation type.

Select:	To:
Translate	Translate the selected points along the X-, Y-, or Z- axis.
Rotate	Rotate selected points about the centre of the magnet.
Scale	Scale selected points about the centre of the magnet.

6. Change the values in the Translation, Rotation, or Scale fields.

3D Deformations

Use the Deformation menu to create or modify a deformation mesh that you can use to deform a 3D model, 3D text, or surface. To use 3D deformations, attach a Deform node to the 3D text, object, or surface, and then use the Deformation menu to modify the deformation mesh. You can animate individual points on the deformation mesh, or move the entire mesh over the 3D text or object to apply the deformation as the mesh passes over the object.



Deformation Mesh

When you add a Deform node, the deformation mesh appears over the 3D text or object. You can view the source mesh, the destination mesh, or both to assist you as you deform the object. You can also turn both meshes off to view only the deformed object.

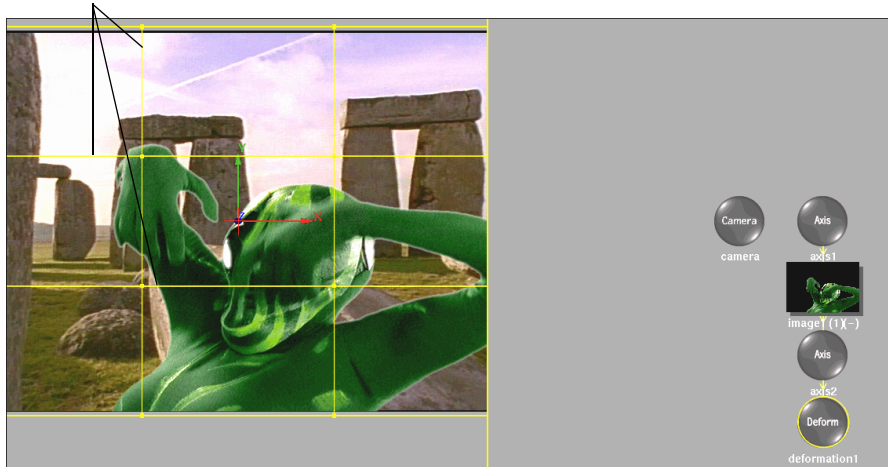
The deformation mesh consists of cells and lattices. You can divide the mesh into 1-100 lattices and each lattice can be divided by 1-3 cells. Increase the number of cells and lattices to deform specific areas of the object.

To add a deformation mesh:

1. In the schematic, select the axis for the image, 3D model, or 3D text.
2. In the Object box, select Deform.
3. Click Add.

The deformation mesh is added to the selected object.

Deformation mesh



If you do not see the deformation mesh, follow the next steps.

4. Double-click the Deform node.

The Deform menu appears.

5. In the Deform menu, enable Show Dest to view the deformation mesh in the image window.

Modifying a Deformation Mesh

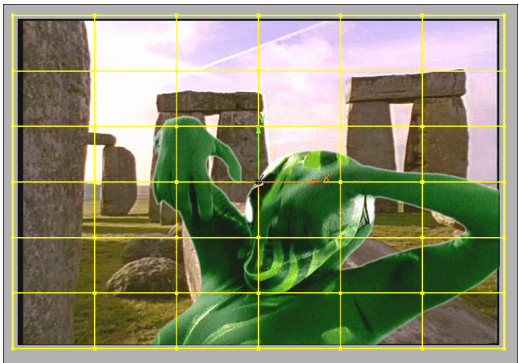
By default, the deformation mesh has three X,Y, and Z cells and one X, Y, and Z lattice. You can change the number of cells (1-3) or lattices (0-100).

NOTE: You can only change the number of cells or lattices before you modify the parameters of the mesh. If you modify a parameter, for example, if you translate a tangent, you cannot change the number of cells or lattices.

To change the number of cells or lattices:

1. In the image window, select the mesh.
2. In the Cells X, Y, and Z fields, change the number of cells. By default they are set to 3, 3, 3.
3. In the Lattice X, Y, and Z field, change the number of vertices.

The cells or lattices are added to the deformation mesh.



- 4. To view the original mesh, enable Show Source.

Use the Outside button to apply the deformation outside of the mesh or constrain the deformation within the mesh. When Outside is enabled, the deformation is also applied outside of the mesh. When Outside is disabled, the deformation is only applied to the object within the mesh. For example, if you animate a mesh so that it passes over the object and the Outside button is disabled, the object will only be deformed as the mesh passes over it. See “Animating a Deformation Mesh” on page 1014.

Selecting Points

You can select individual or multiple points on the mesh and move them to deform the object.

To select points:

- 1. In the Edit Mode box, select Move or Select.

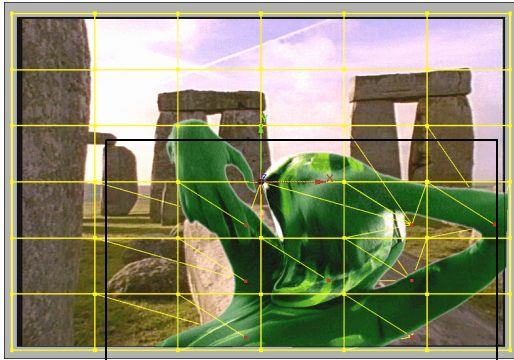
Edit Mode box



- 2. Select the point(s):
 - To select a single point, click the point.

- To select multiple points, hold the **SHIFT** key and click the points you want to select.
- To select a range of points, hold **CTRL** key and draw a box around points you want to select.
- To add a range of points to a selection, hold **SHIFT+CTRL** and draw a box around the points you want to add to the selection.

When you move a point, all select points also move.



To transform a point on the mesh, drag it on the image window, or change the value in one of the X,Y or Z Translation fields.

Animating a Deformation Mesh

You can animate individual or multiple points and any parameter in the Axis menu for the deformation mesh. See Chapter 9, “Animation.” This section describes animation features specific to the deformation mesh.

Passing a Deformation Mesh Over an Object

Animate the deformation mesh so that it passes over the object. If the Outside button is disabled, the deformation is only applied to the parts of the object within the mesh. To achieve this effect, you must move the mesh over the object. You cannot move the object through the mesh because the object axis parents the mesh axis.

To pass a Deformation Mesh over an object:

1. Add the deformation mesh to the object.
2. In the Deform menu, disable Outside.
3. Modify the points on the mesh to achieve the deformation you want.
4. In Schematic view, select the deformation axis.
5. Click Axis to view the Axis menu.
6. Move the deformation mesh so that it is on one side of the object.
7. Set a new keyframe with the deformation mesh on the other side of the object.

Action: The Particle System

Confetti party!

Use Action's 3D particle system to create a variety of effects such as snow, rain, fog, and tornadoes, as well as fireworks, sparks, and explosions.

Summary

In this chapter, you learn about:

- “Generating Particles” on page 1017
- “Customizing the Particle Stream” on page 1021
- “Working with Particle Manipulators” on page 1029
- “Working with Particle Bouncers” on page 1047
- “Exploding Objects” on page 1051
- “Particles on Particles” on page 1053
- “Sample Particle Setups” on page 1056

About Action's Particle System

Particles are 3D objects that originate from a surface, light, or 3D object. The particles can be spheres, cones, squares, points, lines, or any image, 3D text, or 3D object. Particles are generated in the direction of the normals of a surface, light, or 3D object—a normal is a line perpendicular to the surface of an object.

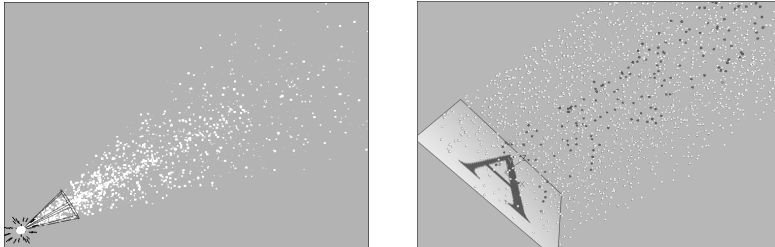
In Action, the main components for generating particles are generators, manipulators, and bouncers. You can also use the particle system to explode a 3D object or an image.

Generators

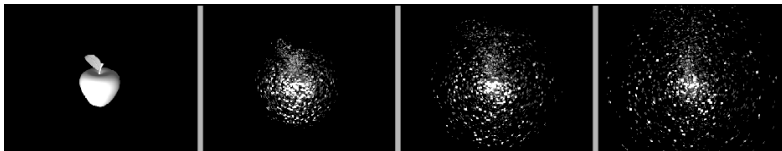
To create a particle stream, start by adding a particle generator that creates a stream of particles when attached to a light source, 3D object, or surface.

Particle generators have two settings:

- The Generate setting creates a stream of particles using the shape of a surface, a 3D object, or the spread of a light source. See “Generating Particles” on page 1017.



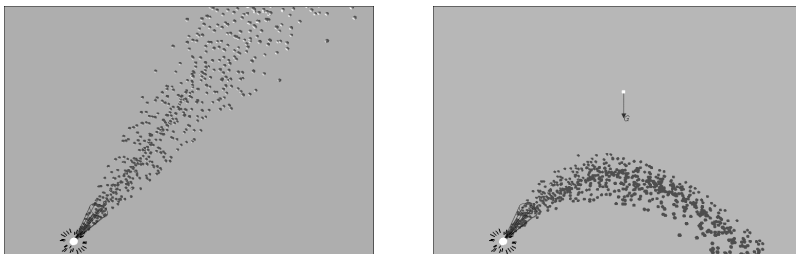
- The Explode setting breaks up a surface or 3D object. See “Exploding Objects” on page 1051.



Manipulators

Particle manipulators are used to influence either the position or the speed of particles.

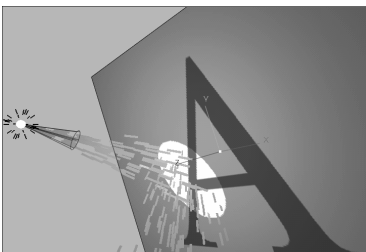
Manipulators can simulate the effects of a vortex, gravitational pull, or the pull toward a point, line, or plane. Manipulators are parented by the particle stream that they influence.



Action offers eight different manipulators. Instructions on how to use each manipulator are provided in “Working with Particle Manipulators” on page 1029.

Bouncers

Particle bouncers are used to influence the behaviour of particles when they come into contact with a surface. You can parent an axis or light source to a particle bouncer. See “Working with Particle Bouncers” on page 1047.



Generating Particles

You can create a particle stream that behaves in different ways depending on the type of generator you use. When creating a particle stream, you can use three different generators: light source, surface, and 3D object.

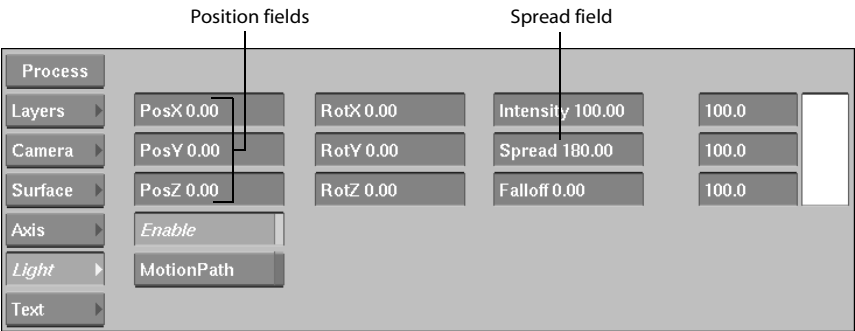
Using a Light Source

Use a light source with a particle generator to create a stream of particles starting from a single point. When a particle generator is attached to a light source, it uses the position and rotation of the light source to move and rotate the particle generator.

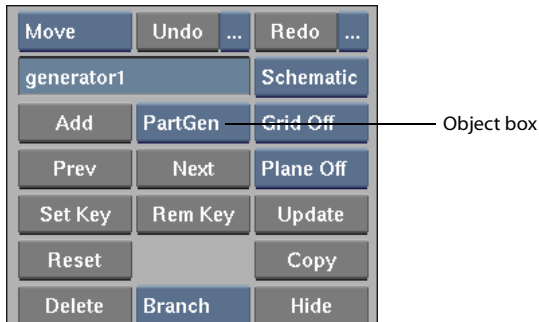
Use the Spread field to set the spread of the particle stream. The Intensity, Falloff, and Colour fields are not used by the particle generator.

To add a particle stream using a light source:

1. Add a light source.
2. Select the new light source in Schematic view.
3. Adjust the spread and position of the light source in the Light menu.



4. In the Object box, select PartGen, and click Add.



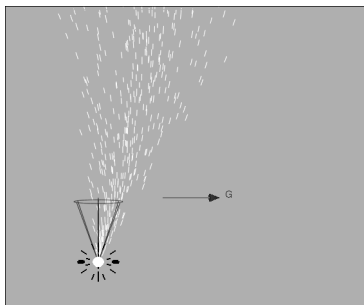
A particle generator object is added and automatically connected to the selected light source.

5. In the Surface menu, select PartGen from the Surface box and set the particle generator's properties.

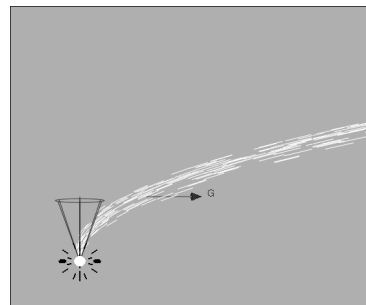


See “Customizing the Particle Stream” on page 1021.

With a light source, you can animate the position of the particle stream by moving the light, changing the spread, and rotating the light. Because the generator is a light source, you can animate the position of the particle generator using a motion path.



Position selected



Speed selected

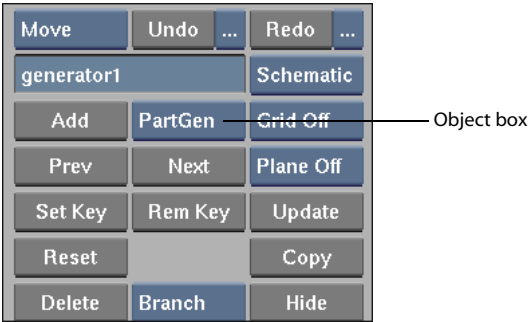
If you do not want the light source to illuminate the scene, you can deactivate the light by clicking the Enable button. When Enable is deactivated, the light source is used as a particle generator only.

Using a Surface

Use a surface as a particle generator to generate particles based on the area, shape, and colour of a surface.

To add a particle stream using a surface:

1. Select or add a surface that you want to use as a particle generator.
2. Adjust the surface's shape, position, and other attributes. See “Working with Surfaces” on page 921.
3. From the Object box, select PartGen and click Add.



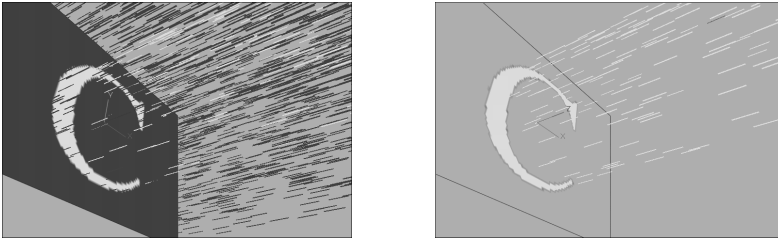
A particle generator object is added and automatically connected to the surface.

4. Set the particle generator's properties.
See “Customizing the Particle Stream” on page 1021.

You can animate the shape of the surface using its tangent handles. The starting area of the particle stream matches the surface's shape animation. See “Changing the Shape of a Surface” on page 925.

In addition, you can use a surface's matte to restrict where particles are generated. For example, the following figure shows two particle generators using surfaces: the surface on the left has its matte turned off and the surface on the right has its matte turned on. When the matte is off, the

particle stream is generated from the entire surface. When the matte is on, particles are only generated where the matte is white or grey.



If you do not want the surface to appear in the scene, you can hide the surface in Schematic view. A particle stream is generated from the hidden surface.

Enable the Indirect button to make the generated particles change as the surface's clip changes. See “Indirect button” on page 1022.

Using a 3D Object

You can use a 3D object as a particle generator. Particles are generated from the centre of each polygon.

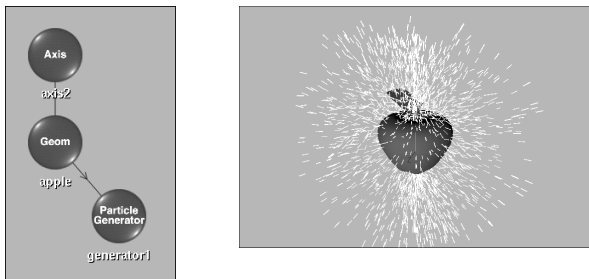
To add a particle stream using a 3D object:

1. Select or add the 3D object that you want to use as a particle generator.
2. From the Object box, select PartGen and click Add.

A particle generator object is added and automatically connected to the 3D object.

3. Set the particle generator's properties.

See “Customizing the Particle Stream” in the following section.



If you do not want the 3D object to appear in the scene, you can hide the model in Schematic view. A particle stream is generated from the hidden model.

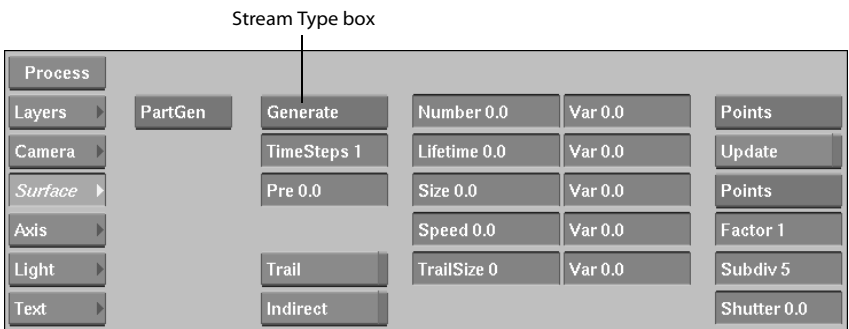
Customizing the Particle Stream

Whether you use a light source, 3D object, or surface, the particle stream uses the same properties, allowing you to create custom particles and apply textures to particles. These properties are found using two menus that correspond to the Surface box PartGen and Geom options.

Select:	To:
PartGen	Set the size, speed, number, and other physical properties and adjust rendering and previewing options.
Geom	Set colour, transparency, specular highlight, and other lighting properties.

Setting Stream Properties

The first column of properties in the PartGen menu affects the particle stream.



Stream Type box — A particle generator can have two stream types: Generate and Explode.

Timesteps field — Use the Timestep field to increase the accuracy of the particle stream. By default, Timestep is set to 1, which means that time simulation is once per frame.

When Timestep is set to 2, the position of each particle is calculated once between frames. Any change in speed or position caused by manipulators or bouncers is applied between frames. When Timestep is set to 3, the position of each particle is calculated twice between frames. The greater the timestep value, the longer it takes to process.

Typically, you use Timestep with a particle manipulator or bouncer on a particle stream. In the following example, the same particle settings and manipulator (AccPoint) are used. Only the timestep values are different. The particle stream on the left uses a timestep of 1 while the stream on the right uses a timestep of 6.

Pre field — Use the Pre field to set the start point of the particle stream. When you use a value of 0.0, the particle stream starts creating particles at frame 1. When the Pre field is set to 100, the clip begins as if the particle stream has been generating for 99 frames.

Trail button — Enable Trail to draw each particle with a trail. The length of the trail is shown in the TrailSize field.

Indirect button — Use the Indirect button only when the particle generator is a surface.

When Indirect is enabled, the part of the surface used by the particle changes with each frame. For example, if the layer applied to the surface changes from red to blue over 10 frames, each particle changes from red to blue as the surface changes.

When Indirect is disabled, the part of the surface used by the particle is locked to the frame when the particle was generated. For example, if the layer applied to the surface changes from red to blue over 10 frames and a particle is generated at the first frame, when the surface was red, the particle stays red.

Setting Particle Properties

In the middle of the PartGen menu, a table lists the different particle generation properties.

Particle generation properties

Process					
Layers ▶	PartGen	Generate	Number 0.0	Var 0.0	Points
Camera ▶		TimeSteps 1	Lifetime 0.0	Var 0.0	Update
Surface ▶		Pre 0.0	Size 0.0	Var 0.0	Points
Axis ▶			Speed 0.0	Var 0.0	Factor 1
Light ▶		Trail	TrailSize 0	Var 0.0	Subdiv 5
Text ▶		Indirect			Shutter 0.0

Number field — This value is the number of particles generated per frame. If you specify a number below 1 (from 0.1 to 0.9), particles are generated randomly.

For example, if you set the number to 0.1, a particle is generated every 10 frames. The exact frame within those 10 frames when the particle is generated is randomly selected. If you set the number of particles to 0.3, a particle is randomly generated approximately every three frames.

Lifetime field — This value is the number of frames each particle lasts. For example, if Lifetime is 20 frames, a particle generated at frame 15 disappears at frame 35.

Size field — This value is the size of each particle in pixels. This parameter only affects cones, spheres, quads, squares, and objects.

Speed field — This value is the speed of each particle in pixels per frame.

TrailSize field — This value is the length of each particle’s trail in particles. A particle trail only appears when the Trail button is enabled.

Varying Each Parameter

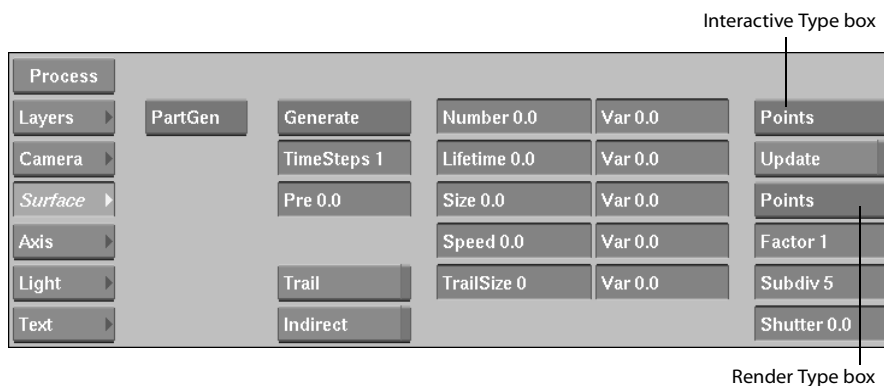
Each parameter has a variance value (Var field) used to specify how much the parameter varies each time particles are generated. Exactly what gets varied depends on the parameter.

For example, if you set the Number parameter to 10 and its Var parameter to 5, the number of particles generated at each frame is randomly selected between 5 and 15.

The unit of measurement used by each Var field matches the parameter it is varying. For example, the Number parameter is expressed in particles per frame, so is its Var field. The Speed parameter is in pixels per frame and so is its Var field.

Setting Particle Rendering

The last column in the PartGen menu lists the properties that affect the rendering and previewing of the particle stream.



Interactive Type box — Use the Interactive Type box to select the type of particle to display in the Action scene. For example, as you move between frames, the particle stream is redrawn. If you are using a custom 3D object for each particle, redrawing the scene may take a while. By changing the interactive type to a point or line, your interaction with Action is much quicker.

See “Render Type box” on page 1024 for a description of each particle type.

Update button — Enable Update to automatically update the scene when you change a particle generator parameter. Using Action with this button enabled may cause slowdowns since the particle stream is recalculated each time a parameter is changed.

Render Type box — Use this box to select the type of particle used when rendering the scene. The following is a complete list of particle types.

Select:	To:
Node	Use your own 3D object, text, or a surface for each particle. See “Using Custom Particles” on page 1026.
Cones	Use a cone for each particle. You can adjust the size of the cone’s tail. See “TailSize channel” on page 1025.
Spheres	Use a 3D object of a sphere for each particle.
Polygons	Explode 3D objects and surfaces. See “Exploding Objects” on page 1051.
Quads	Use a square with a tail for each particle. You can adjust the size of the square’s tail. See “TailSize channel” on page 1025.
Squares	Use a square for each particle.
Lines	Use a line for each particle.
Points	Use a single pixel for each particle. Points are not affected by size or trail.

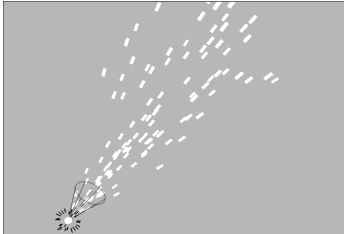
Factor field — Use the Factor field to speed up Action by changing the number of particles shown in the scene. This works only while using Action and does not affect the rendered result.

For example, if you generate 1000 particles with manipulators and bouncers, interacting with Action may become slow because calculations have to be made for each particle. If you specify a factor of 10, the number of particles is reduced to 10%, or 100 particles. Using Action is faster, yet when you click Preview or Process, 1000 particles are created.

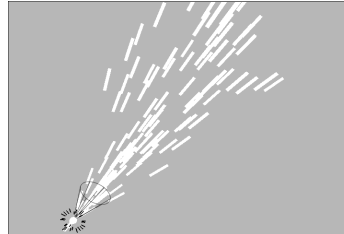
Subdiv field — Use the Subdiv field to set the number of polygons used to render cones and spheres. The larger the number of subdivisions, the smoother the cones or spheres are drawn. Increasing Subdiv also increases rendering time.

NOTE: If you are using an imported 3D object or text for a particle, use the Subdiv field and button in the Geom menu (Surface menu) to set the number of polygons. The Geom menu’s Subdiv field works differently than the Subdiv field in the PartGen menu. See “Using Subdivide” on page 991.

Shutter field — Use Shutter to define the length of the tail for lines, cones, and quads. Shutter is expressed as a percentage. At 100%, the tail's length is the same as the head (Size field). At 400%, the tail's length is four times the size of the head.



Shutter at 100%



Shutter at 400%

Setting Channel Editor Properties

The Channel Editor includes many detail particle properties for which you can set values and create particle animation effects. You set these properties in conjunction with or in addition to those located on the Surface menus for the Particle Generator, Particle Manipulator, and Particle Bouncer menus.

You must first add a particle generator, a particle manipulator, or a particle bouncer to the scene to view and set values for their corresponding properties in the channel hierarchy of the Channel Editor. See “The Channel Hierarchy” on page 130.

Mass channel — Set and animate the mass of each particle. Mass is used with damping when manipulating particles. See “Setting Manipulator Properties” on page 1030.

Mass_V channel — The Mass_V channel is a percentage used to vary the mass. This value works the same way as the Var fields for the properties found in the PartGen menu. See “Varying Each Parameter” on page 1023.

TailSize channel — Use the TailSize parameter to change the width of a particle's tail. TailSize is expressed as a percentage:

- At 100%, the tail is the same width as the head.
- At 200%, the tail is twice as wide as the head.

TailSize only works with quad and cone particle types.

TailSize_V channel — The TailSize_V channel is a percentage used to vary the TailSize channel. TailSize_V only works with quad and cone particle types.

Colour_V channel — Varies the colour of each particle based on the colour set in the Diffuse colour bar (Geom menu). The variance is expressed as a percentage.

Spread channel — Use the Spread channel to set the spread for the second particle generator when two particle generators are parented. When the first particle stream dies, the second particle stream begins. See “Particles on Particles” on page 1053.

This also sets the spread of a particle stream that bounces off surfaces. See “Working with Particle Bouncers” on page 1047.

Spread_V channel — The Spread_V channel varies the spread for a particle stream when parented by a second particle stream.

Using Custom Particles

You can create particle streams using your own 3D objects or text. You can also use the shape of any surface for each particle. You can create custom particles using a light source, 3D object, or surface as a particle generator. See “Generating Particles” on page 1017.

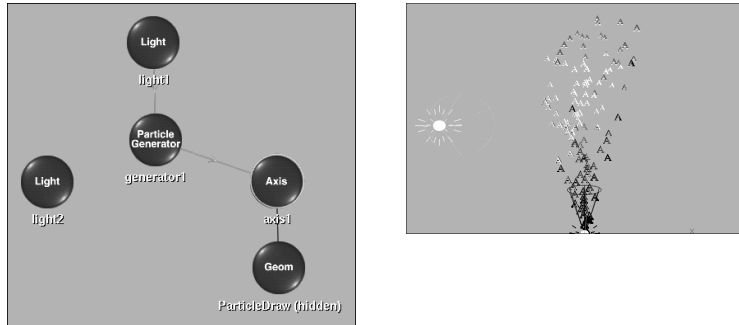
To use a 3D object or text as a custom particle:

1. Add a particle generator by selecting PartGen from the Object box and then clicking Add.
2. In the Surface menu, select PartGen from the Surface box.



3. In the Render Type box, select Node as the particle type and change the Interactive Type box to Node to view the 3D object or text as you make changes to your particle stream.
4. Import the 3D object or create the text. See “Importing 3D Models” on page 983 or “Creating 3D Text” on page 985.
5. In Schematic view, rename your 3D object or text “ParticleDraw”. This name is case sensitive so make sure it is typed exactly as shown.
6. Parent the particle generator to the 3D object or text.
If you want to add transformations, such as rotating all the particles, parent the particle generator to the axis of the 3D object or text. An example is shown as follows.
7. Hide the 3D object or text. See “Hiding Objects” on page 962.

The particle generator uses the 3D object or text named ParticleDraw. The following figure shows the schematic and the result of using the letter A as a custom particle and a light source as the particle generator.



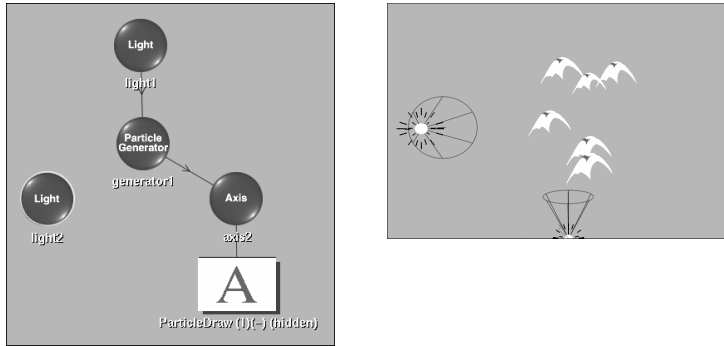
To use a surface as a custom particle:

1. Add a particle generator.
You can use a light source, 3D object, or surface as a particle generator. See “Generating Particles” on page 1017.
2. Select Node as the particle type in the Render Type box. To view the surface as you make changes to your particle stream, change the Interactive Type box to Node as well. See “Setting Particle Rendering” on page 1023.
3. Add a surface to the scene. It can be an image, a bilinear, a bicubic, or an extended bicubic surface.

NOTE: You do not have to apply a layer to the surface because the particle generator only uses the surface’s shape and not its layer. To have an image appear on each particle, you have to apply a texture to the particle generator. See “Applying Textures to Particles” on page 1028.

4. Rename your surface “ParticleDraw”. This name is case sensitive so make sure it is typed exactly as shown.
5. Parent the particle generator to the surface.
If you want to add transformations, such as rotating all the particles, parent the particle generator to the axis of the surface. An example is shown as follows.
6. Hide the surface.

The following figure shows the schematic and the result of using a bicubic surface as a custom particle and a light source as the particle generator. The shape of the bicubic surface is animated and its axis is used to rotate the particles.



NOTE: You can animate the shape of a bilinear, bicubic, or extended bicubic surface, and have the particles reflect the animation.

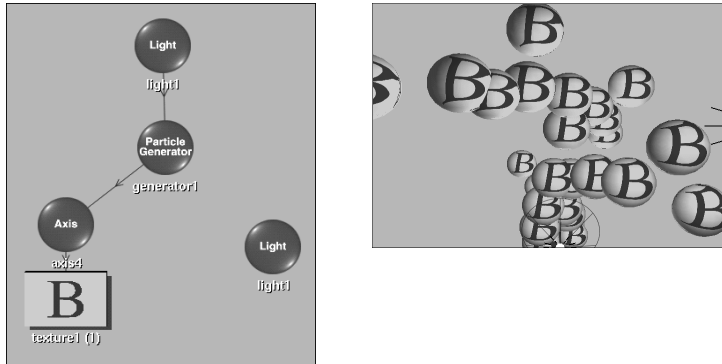
Applying Textures to Particles

You can apply textures to particles using the same method as applying textures to 3D objects and text. When applying a texture, you can also set its texture mapping using the Texture menu. See “Working with Textures” on page 992.

To apply a texture to a particle stream of spheres:

1. Create a particle generator.
2. In the Render Type box, select Spheres as the particle type. To view the 3D object or text as you make changes to your particle stream, change the Interactive Type box to Spheres as well. See “Setting Particle Rendering” on page 1023.
3. In the schematic, select the Particle Generator.
4. In the Object box, select Texture and click Add.
A texture is added using the currently selected layer and is parented by the particle generator.
5. Apply the appropriate layer to the Texture element and set the texture mapping.

For example, the following shows the schematic and the result of a particle stream using spheres and a texture. Reflection is selected as the texture mapping, making each particle resemble a glass ball.



Working with Particle Manipulators

You use manipulators to direct the particle stream and create a number of different effects, such as a swirling vortex, a swarm of objects orbiting a point, or a waterfall. A particle stream can have an unlimited number of manipulators. Manipulators only influence the particle stream that parents them.

When manipulating particles, you use the Surface Particle Manipulator menu to set a manipulator's type, magnitude, power, and damping properties. To set the position, rotation, scale, and shear of the selected particle manipulator, you use the Axis menu. “Working with Particle Manipulators” on page 1029.

Particle manipulators can be animated using motion paths. You can use the same operations on a particle manipulator that you use on an axis. See “Manipulating an Axis” on page 941.

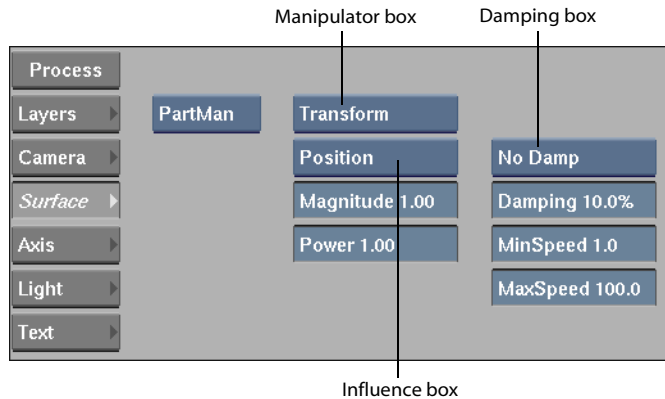
Not all manipulators use all the properties found in the Surface Particle Manipulator menu and Axis menu. See “Setting Manipulator Properties” on page 1030.

To create a particle manipulator:

1. In the schematic, select the particle generator that you want the manipulator to influence.
2. In the Object box, select PartMan and click Add.
A particle manipulator element is added to the scene and parented by the selected PartGen element.
3. In the Surface menu, select the PartMan load option from the Surface box.
4. Select the type of manipulator and set its properties.

Setting Manipulator Properties

You use the Surface Particle Manipulator menu to select the type of manipulator, its speed and position, and the power of its falloff. You can also select damping, damping percentage, and whether to damp on size, mass, or both.



Use the Manipulator Type box to select the type of manipulator as described in the following table.

Select:	To:
Gravity	Simulate the effect of gravity. It works with position or speed. Magnitude is the strength of the gravitational pull. Gravity does not use the Power field.
Transform	Apply transformations from the Axis menu to the position or speed of each particle. Magnitude is used as an extra scaling factor. Transform does not use the Power field.
Vortex	Mimic the effect of a vortex. It works with position or speed. Magnitude is used for vortex rotation. Power is the amount of falloff from the centre of the vortex.
AccPoint	Pull particles toward a point. Particles are manipulated on the X, Y, and Z axes. Works with position or speed. Magnitude is the strength of the pull. Power is the amount of falloff.
AccLine	Pull particles toward a line. Particles are manipulated on two axes. Magnitude is the strength of the pull. Power is the amount of falloff.
AccPlane	Pull particles toward a plane. Particles are manipulated on a single axis. Magnitude is the strength of the pull. Power is the amount of falloff.
Path	Make the particles follow a path.
Function	Enter a mathematical expression to use as a particle manipulator.

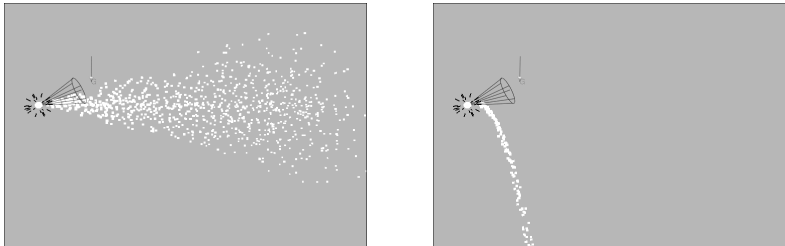
Position or Speed

A particle manipulator can have two types of influence: position or speed. Use the Influence box to select whether the manipulator influences each particle's position or speed.

Set the initial speed of each particle using the Speed field in the Surface Particle Generator menu.

By selecting Speed in the Influence box located on the Particle Manipulator menu, each particle's speed is changed with each pass by the selected manipulator.

When you influence by position, the position is applied only once. For example, the following figure uses a gravity manipulator to illustrate the difference between position and speed. The gravity manipulator on the left uses the default gravity manipulator with a magnitude of 4 and with position as the influence. The gravity manipulator on the right uses exactly the same settings except the influence is set to speed.



Magnitude

Magnitude is a unit of measurement specific to each manipulator. It is used by each manipulator for everything from extra scale when using Transform, to a pixel per frame acceleration factor when using AccPoint, AccLine, or AccPlane. By default, the Magnitude field is set to 1.0.

Power

Power is the amount of falloff from the centre of a manipulator. By default, power is set to 1. This means that each particle's speed or position is influenced if the particle is within a 100-pixel radius of the manipulator. When the power is set to 2, the falloff is approximately 200 pixels. The higher the power, the higher the falloff from the manipulator.

If you specify 0 as the power, there is no falloff; the manipulator's influence is universal. This means that a particle is affected no matter where it is located in the scene. A negative value makes particles that are farther away affected by the manipulator.

Damping

Damping affects the speed of particles by applying friction based on each particle's mass, size, or both. Select the type of friction from the Damping box.

Select:	To:
No Damp	Turn off damping. Particles have no friction.
Damp Mass	Activate damping based on mass. The greater a particle's mass, the slower its speed in relation to the manipulator. You set the mass using the Mass channel in the Channel Editor. See "Mass channel" on page 1025.
Damp Size	Activate damping based on particle size. The larger the particle, the slower it moves in relation to the manipulator. This corresponds with the Size field in the Surface Particle Generator menu. See "Size field" on page 1022.
Damp Both	Activate damping based on mass and particle size.

Damping Percentage

The damping percentage is the percent value of how much Damp Mass, Damp Size, or Damp Both affects the position or speed of the particles.

Min Speed and Max Speed

The Min Speed and Max Speed fields specify the minimum and maximum speed range for particles affected by damping. Particles are not decelerated below the minimum speed or accelerated above the maximum speed. Both speeds are specified in pixels per frame.

You can use the Min and Max Speed fields to force a minimum and maximum speed on particles without damping. To do this, select Damp Mass, Damp Size, or Damp Both. Change the Damping Percentage to 0% and change the Min and Max Speed fields accordingly. Use this feature to stop particles from being over accelerated.

For example, when you use a vortex manipulator, the closer particles pass to the centre of the vortex, the more they are accelerated. Occasionally, a particle may be over accelerated and shot too far, too fast. By forcing a maximum speed, particles are not accelerated past a certain value.

Applying Transformations

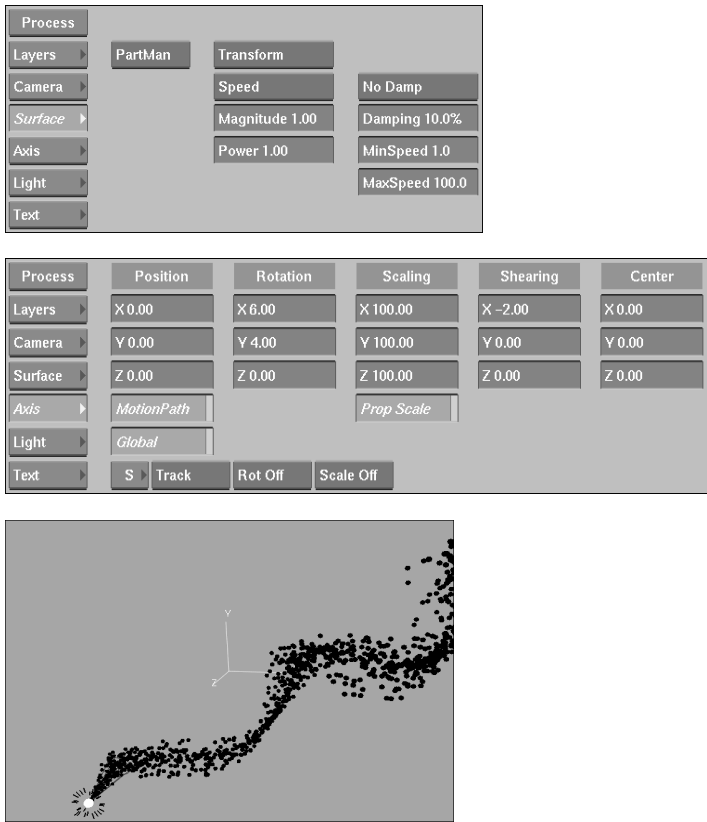
Use a transform manipulator to apply the accumulated transformations from the Axis menu to the position or speed of each particle. Unlike other manipulators, the Axis menu is not used to position the manipulator in the scene. Transform manipulators are always placed at the centre of the particle stream they influence. The following translations are accessed through the Axis menu.

Translation	Description
Position	Use the PosX, PosY, and PosZ fields to displace each particle along the X, Y, and Z axes.

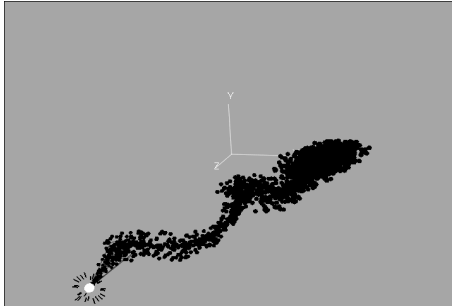
Translation	Description
Rotation	Use the RotX, RotY, and RotZ fields to rotate each particle. Each particle is not the centre of its rotation. Particles rotate around the manipulator's axis.
Scale	Use the ScaleX, ScaleY, and ScaleZ fields to increase or decrease the speed or position of each particle along the X,Y, and Z axes. For example, if speed is the selected influence, a ScaleX of 100% has no effect while a ScaleX of 105% increases the speed along the X axis by 5% with each pass.
Shear	Use the ShearX, ShearY, and ShearZ fields to shear the shape of each particle along the X, Y, and Z axes.

NOTE: For the transform manipulator to work properly, the Global button must be enabled. Otherwise, the position, scale, rotation, and shear are inherited and may result in particles being over-manipulated.

The following figure provides an example of creating a particle stream influenced by a transform manipulator. The translations in the Axis menu, the settings in the Surface Particle Manipulator menu, and the resulting effect are shown.



In the Surface Particle Manipulator menu, use Magnitude as an extra scaling factor whose sensitivity depends on the transformations in the Axis menu. For example, the following is a copy of the above particle stream with Magnitude set to 0.99 instead of 1.0.



The particle generator and transform settings are the same as the example above except for Magnitude, which is set to 0.99 instead of 1.00.

Simulating Gravity

Use a gravity manipulator to simulate the effects of gravity. This manipulator works on speed or position. The Power field has no effect.

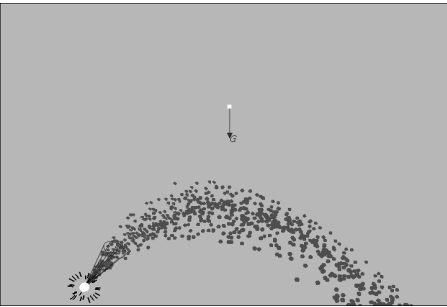
Magnitude specifies the gravitational pull in pixels per frame. A higher magnitude produces a higher gravity, which exerts a greater influence on each particle.

The icon for a gravity manipulator uses an arrow to indicate the direction of the influence. You can change the position and rotation of the arrow using the Axis menu. You can also change the scale of the gravity manipulator as another way of increasing its influence.

The following figure provides an example of creating a simple particle stream influenced by the gravity manipulator. The settings in the Axis menu, the Surface Particle Manipulator menu, and the resulting effect are shown.

Process			
Layers	PartMan	Gravity	
Camera		Speed	No Damp
Surface		Magnitude 0.20	Damping 10.0%
Axis		Power 1.00	MinSpeed 1.0
Light			MaxSpeed 100.0
Text			

Process	Position	Rotation	Scaling	Shearing	Center
Layers	X -106.81	X 90.00	X 100.00	X -2.00	X 0.00
Camera	Y 69.85	Y 4.00	Y 100.00	Y 0.00	Y 0.00
Surface	Z -361.33	Z 0.00	Z 100.00	Z 0.00	Z 0.00
Axis	MotionPath		Prop Scale		
Light	Global				
Text	S	Track	Rot Off	Scale Off	



Creating a Vortex Effect

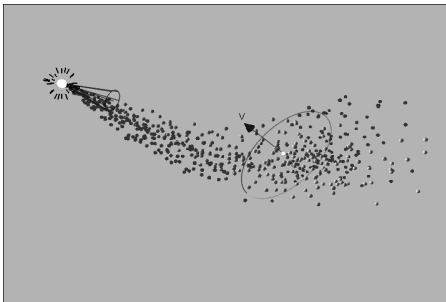
Use a vortex manipulator to influence the motion and rotation of particles. This manipulator works on speed or position. Power is the falloff from the centre of the vortex and magnitude is used for the scale of the vortex's rotation.

You can also use the Axis menu to apply additional transformations to the vortex. Scale changes the size of the vortex, rotation affects the spin of the vortex, and shear is used to make the vortex oval instead of circular.

The following figure provides an example of creating a simple particle stream influenced by the vortex manipulator. The settings in the Axis menu, the Surface Particle Manipulator menu, and the resulting effect are shown.

Process	Position	Rotation	Scaling	Shearing	Center
Layers ▾	X -9.35	X -39.00	X 100.00	X -2.00	X 0.00
Camera ▾	Y 11.72	Y -24.00	Y 100.00	Y 0.00	Y 0.00
Surface ▾	Z -385.33	Z 0.00	Z 100.00	Z 0.00	Z 0.00
Axis ▾	MotionPath		Prop Scale		
Light ▾	Global				
Text ▾	S ▾ Track Rot Off Scale Off				

Process		
Layers ▾	PartMan	Vortex
Camera ▾		Speed
Surface ▾		Magnitude 2.00
Axis ▾		Power 1.00
Light ▾		No Damp
Text ▾		Damping 10.0%
		MinSpeed 1.0
		MaxSpeed 100.0



NOTE: The icon used to show a vortex does not indicate the limit of the vortex. The icon is only used to represent the vortex in the scene and is not meant as an accurate depiction of the size, strength, or limit of the vortex's influence.

Using an AccPoint Manipulator

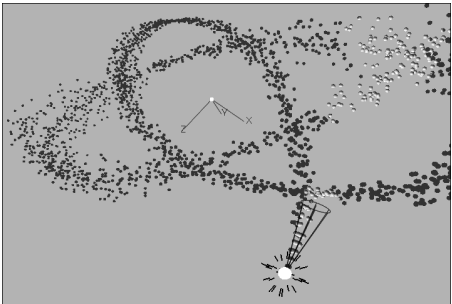
The AccPoint manipulator mimics the gravitational pull between bodies. Unlike the gravity manipulator, which pushes particles in a specific direction, AccPoint continually pulls particles toward itself on all three axes (X, Y, Z).

In the Surface Particle Manipulator menu, magnitude is the gravitational pull in pixels per frame. A higher magnitude produces a higher gravity, which results in a greater influence. Power is the distance from the centre, or the radius, of the pull.

The following figure provides an example of creating a simple particle stream influenced by the AccPoint manipulator. The settings in the Axis menu, the Surface Particle Manipulator menu, and the resulting effect are shown.

Process	Position	Rotation	Scaling	Shearing	Center
Layers	X 108.69	X 124.38	X 100.00	X -2.00	X 0.00
Camera	Y 80.87	Y -22.96	Y 100.00	Y 0.00	Y 0.00
Surface	Z -143.05	Z 33.40	Z 100.00	Z 0.00	Z 0.00
Axis	MotionPath		Prop Scale		
Light	Global				
Text	S	Track	Rot Off	Scale Off	

Process		
Layers	PartMan	Acc Point
Camera	Position	No Damp
Surface	Magnitude 1.00	Damping 10.0%
Axis	Power 1.00	MinSpeed 1.0
Light		MaxSpeed 100.0
Text		



NOTE: The Axis menu is used only to position the AccPoint in the scene. Scale, rotation, and shear have no effect.

Using an AccLine Manipulator

Like AccPoint, the AccLine manipulator continually pulls particles toward itself. The difference between AccPoint and AccLine is that AccLine manipulates particles on two axes instead of all three.

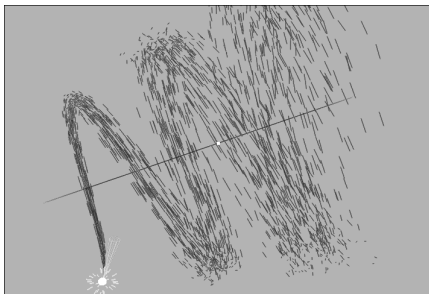
You can set the two axes by rotating the manipulator using the Rot fields in the Axis menu. The Axis menu is also used to position the AccLine manipulator in the scene. Scale and shear may change the appearance of the AccLine icon, but they have no effect on particles.

In the Surface Particle Manipulator menu, Magnitude is the gravitational pull in pixels per frame. A higher magnitude produces a higher gravity which results in a greater influence. Power is the falloff from the manipulator, or the radius of the pull.

The following figure provides an example of creating a simple particle stream influenced by the AccLine manipulator. The settings in the Axis menu, the Surface Particle Manipulator menu, and the resulting effect are shown.

Process	Position	Rotation	Scaling	Shearing	Center
Layers ▶	X 195.66	X -13.27	X 100.00	X 0.00	X 0.00
Camera ▶	Y 28.90	Y -56.32	Y 100.00	Y 0.00	Y 0.00
Surface ▶	Z -358.91	Z 2.00	Z 100.00	Z 0.00	Z 0.00
Axis ▶	MotionPath		Prop Scale		
Light ▶	Global				
Text ▶	S ▶ Track		Rot Off Scale Off		

Process		
Layers ▶	PartMan	Acc Line
Camera ▶	Position	No Damp
Surface ▶	Magnitude 3.00	Damping 10.0%
Axis ▶	Power 0.00	MinSpeed 1.0
Light ▶		MaxSpeed 100.0
Text ▶		



Using an AccPlane Manipulator

Like AccPoint and AccLine, the AccPlane manipulator continually pulls particles toward itself. With AccPlane, particles are manipulated on a single axis set by rotating the plane using the Rot fields in the Axis menu. AccPlane manipulates particles on only one axis instead of two or three.

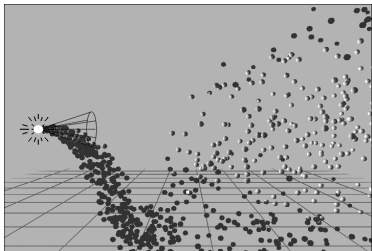
Use the Axis menu to position the AccPlane manipulator in the scene. The Scale and Shear fields are not used.

In the Surface Particle Manipulator menu, magnitude is the gravitational pull in pixels per frame. A higher magnitude produces a higher gravity, which results in a greater influence. Power is the falloff from the manipulator, or the radius, of the pull.

The following figure provides an example of creating a simple particle stream influenced by the AccPlane manipulator. The settings in the Axis menu, the Surface Particle Manipulator menu, and the resulting effect are shown.

Process	Position	Rotation	Sealing	Shearing	Center
Layers ▶	X 0.00	X 90.00	X 100.00	X 0.00	X 0.00
Camera ▶	Y -125.00	Y -56.32	Y 100.00	Y 0.00	Y 0.00
Surface ▶	Z 0.00	Z 2.00	Z 100.00	Z 0.00	Z 0.00
Axis ▶	MotionPath		Prop Scale		
Light ▶	Global				
Text ▶	S ▶	Track	Rot Off	Scale Off	

Process		
Layers ▶	PartMan	Acc Plane
Camera ▶	Position	No Damp
Surface ▶	Magnitude 1.00	Damping 10.0%
Axis ▶	Power 0.00	MinSpeed 1.0
Light ▶		MaxSpeed 100.0
Text ▶		



NOTE: Because AccPlane influences particles on only one axis, the effect may not be noticeable until you change views or orbit the camera.

Forming a Particle Path

Use the path manipulator to make particles follow the motion of a path.

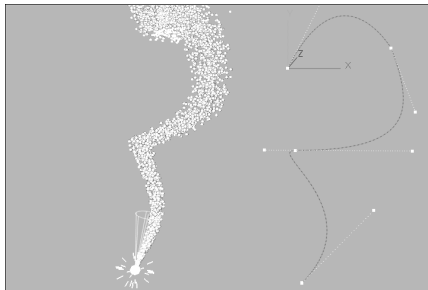
The motion of a path is applied to the speed or position of each particle. Note that particles do not directly follow the path. You can make the particles seem to follow the path by lowering the particle speed and magnitude, which is used as a scaling factor. Power is not used.

Use the Axis menu to animate the path. You can also use the Rotation, Scale, and Shear fields to add extra transformations.

The following figure provides an example of creating a simple particle stream influenced by the path manipulator. The settings in the Axis menu, the Surface Particle Manipulator menu, and the resulting effect are shown.

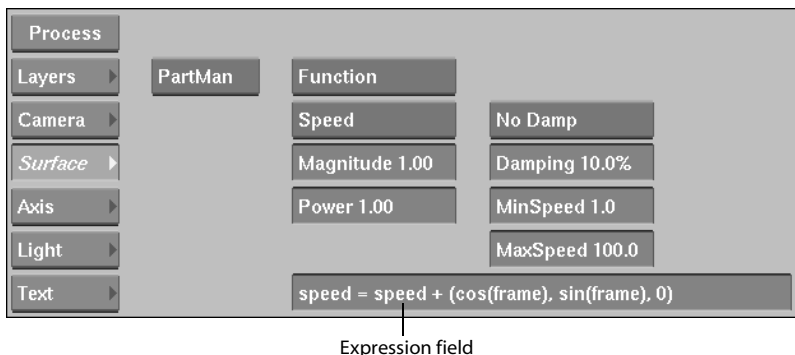
Process	Position	Rotation	Scaling	Shearing	Center
Layers ▾	X 103.41	X 90.00	X 100.00	X 0.00	X 0.00
Camera ▾	Y 136.25	Y -56.32	Y 100.00	Y 0.00	Y 0.00
Surface ▾	Z 0.00	Z 2.00	Z 100.00	Z 0.00	Z 0.00
Axis ▾	MotionPath		Prop Scale		
Light ▾	Global				
Text ▾	S ▾ Track		Rot Off		Scale Off

Process		
Layers ▾	PartMan	Path
Camera ▾	Position	No Damp
Surface ▾	Magnitude 0.60	Damping 10.0%
Axis ▾	Power 0.00	MinSpeed 1.0
Light ▾		MaxSpeed 100.0
Text ▾		



Using a Function as a Manipulator

Select Function to specify your own mathematical expressions to be used as a particle manipulator. You can use the channels from the Surface Particle Manipulator and Surface Particle Generator menus in your expressions, as well as arithmetic operations, mathematical conventions, functions and constants.



The arithmetic operators, conventions, constants, and functions that you can use in your expressions are listed in the following sections. Before writing your own expressions, you should understand the following:

- A vector is a 3D coordinate written using the convention (x, y, z) where x, y, and z are separate values.

For example, pos represents the position of each particle. If you want to increase the Y position of each particle with each pass, use the expression `pos = pos + (0,1,0)`.

- Make sure assigned values and vector values are within an acceptable range. For example, transparency (opacity of each particle) is a value between 0 and 1. The expression `transparency = size` does not work unless size (particle size from the PartGen menu) falls between 0.0 and 1.0.

To make transparency dependent on size, size must be divided by an appropriate value. For example, if size is between 1 and 10, use the expression `transparency = size / 10`.

- Some functions return scalars and other functions return vectors. Make sure that when you use a function it returns the right value and that this value is within an acceptable range.

For example, rgb is a vector of values between 0.0 and 1.0. The expression `rgb = (0, 0, noise3(pos))` gives an error because noise3 returns a vector. The expression `rgb = (0, 0, frame)` does not give an error, but frame is the frame number and it is never less than 1. This means the blue channel is always set to full blue (1).

- You can specify more than one function by separating each with a semicolon.

For example, to place the two expressions in the Expression field, you would type:

```
speed = speed + pos; pos = pos + (0,size,0)
```

Expression field operators are listed in the following table.

Operators	Description
=	Equals
+	Addition
-	Subtraction
*	Multiplication
/	Division
%	Percentage
(x,y,z)	Vector where x,y,z may also be the results of functions
==	Equivalence
!=	Not equal to
<	Less than
<=	Less than or equal to
>	Greater than
>=	Greater than or equal to

Particle and manipulator variables that you can use in the Expression field are listed in the following table.

Particle Symbols	Description
pos	Position, a vector (x,y,z).
speed	Speed, a vector (x,y,z).
rgba	Red, green, blue, and alpha colour channels for each particle, expressed as a 4D vector (r,g,b,a). Each component is a value between 0 and 1.
rgb	Red, green, and blue colour channel, expressed as a vector (r,g,b). Each component is a value between 0 and 1.
red	Red channel, a value between 0 and 1.
green	Green channel, a value between 0 and 1.
blue	Blue channel, a value between 0 and 1.
transparency	Transparency of each particle, a value between 0 and 1.
lifetime	Lifetime of each particle, in frames.
lifetime1	A value between 1.0 and 0.0 where 1.0 is when a particle is first generated and 0.0 is when it ends.
mass	Mass of each particle.

Particle Symbols	Description
size	Size of each particle, in pixels.
tailSize	Width of the particle's tail, a value between 0 and 1.

The following variables are read only. You cannot change them in the Expression field, but you can use them in your calculations.

Manipulator Symbols	Description
frame	Current frame
magnitude	Value from the Magnitude field
power	Value from the Power field
damping	Value from the Damping field
minSpeed	Value from the MinSpeed field
maxSpeed	Value from the MaxSpeed field

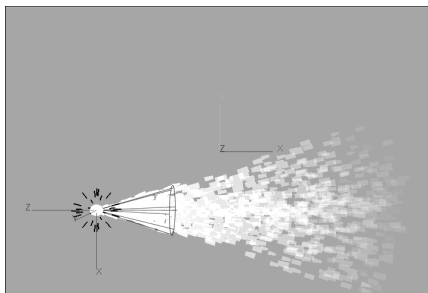
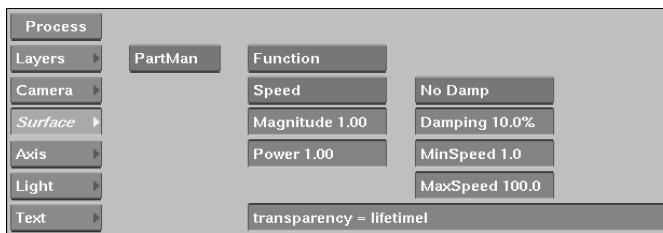
Single argument arithmetic functions are listed in the following table.

One Argument	Description
$\sin(a)$	Sine of a
$\cos(a)$	Cosine of a
$\tan(a)$	Tangent of a
$\text{asin}(a)$	Arcsine of a
$\text{acos}(a)$	Arccosine of a
$\text{atan}(a)$	Arctangent of a
$\exp(x)$	Exponential function of x
$\expm1(x)$	Equivalent to $\exp(x)-1$
$\log(x)$	Natural logarithm of x
$\log_{10}(x)$	Base 10 logarithm of x
$\log_{1p}(x)$	Equivalent to $\log(1+x)$
$\text{sqrt}(x)$	Square root of x
$\text{abs}(x)$	Absolute value of x
$\text{trunc}(x)$	Integer value of x
$\text{floor}(x)$	Smallest integer greater than or equal to x
$\text{ceil}(x)$	Largest integer greater than or equal to x
$\text{round}(x)$	x rounded to the nearest integer

One Argument	Description
<code>radians(<i>a</i>)</code>	<i>a</i> converted to radians
<code>degrees(<i>r</i>)</code>	<i>r</i> converted to degrees
<code>sign(<i>x</i>)</code>	Returns +1 or -1 depending on the sign of <i>x</i>
<code>length(<i>p</i>)</code>	Euclidean length of point <i>p</i>
<code>noise(<i>v</i>)</code>	Noise of vector <i>v</i> , returns a float
<code>fnoise(<i>v</i>)</code>	Fractal noise vector <i>v</i> , returns a float
<code>noise3(<i>v</i>)</code>	Noise of vector <i>v</i> , returns a vector

Making Particles Transparent

The following example illustrates how to make particles become transparent as they reach the end of their lifetime using the expression `transparency = lifetime`. The settings in the Surface Particle Manipulator menu, PartGen menu, and the resulting effect are shown.

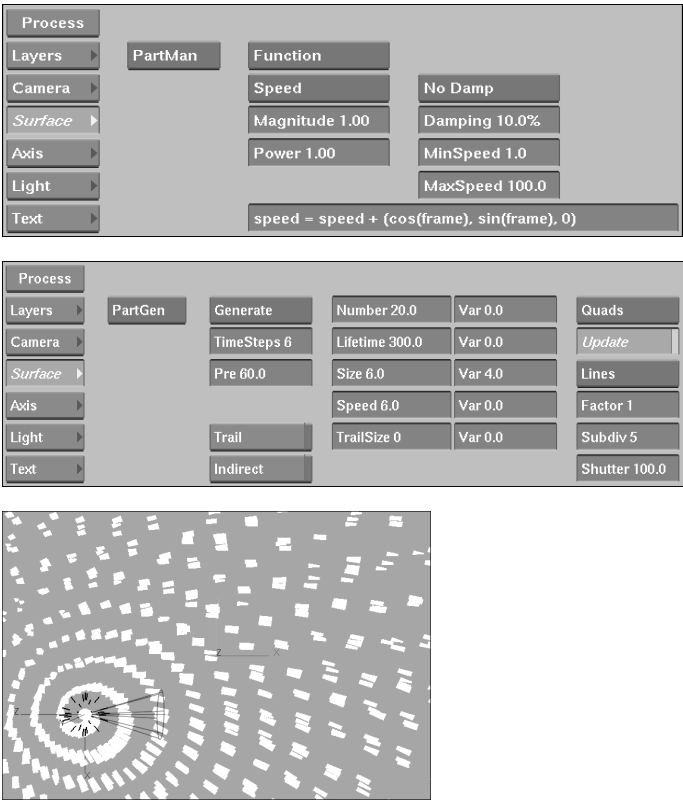


Making Particles Spin

The following example illustrates how to spin particles using the expression:

```
speed = speed + (cos(frame),sin(frame),0)
```

The settings in the Surface Particle Manipulator menu, PartGen menu, and the resulting effect are shown. Notice that the particle Timestep value is increased to improve the accuracy of the particle stream.



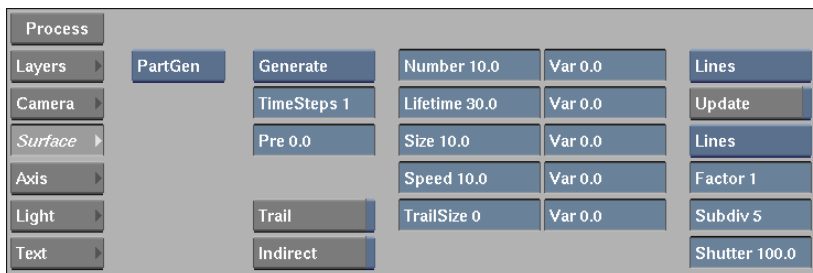
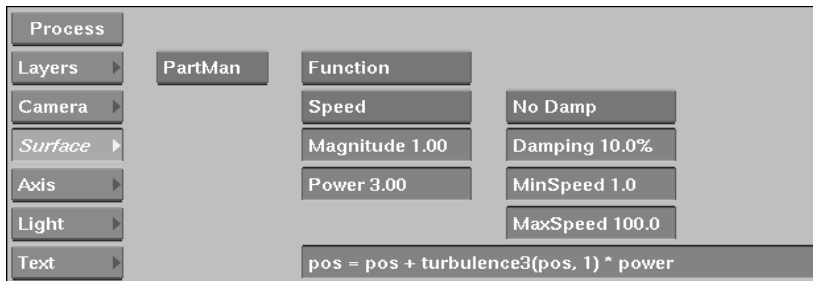
Arithmetic functions that have two or more arguments are listed in the following table.

Symbol	Description
atan(x,y)	Arctangent of y over x
pow(x,y)	x to the power of y
mod(x,y)	Returns the remainder of dividing x by y
min(x,y)	Minimum value of x and y
max(x,y)	Maximum value of x and y

Symbol	Description
<code>step(x,y)</code>	Returns 0 if $x < y$, 1 if $x \geq y$
<code>dot(v1,v2)</code>	Dot product of two vectors; returns a scalar.
<code>cross(v1,v2)</code>	Cross product of two vectors; returns a vector.
<code>turbulence(v,o)</code>	Turbulence of vector v and octave o ; returns a float.
<code>turbulence3(v,o)</code>	Turbulence of vector v and octave o ; returns a vector.
<code>smoothstep</code> <code>(min,max, x)</code>	Returns 0 if $x < min$, 1 if $x \geq max$. If neither are true, returns a hermite interpolation between 0 and 1.
<code>clamp(x, min, max)</code>	x clamped to the range $[min, max]$

The following example illustrates how to create animated turbulence using the expression:

```
pos = pos + turbulence3(pos, 1) * power
```

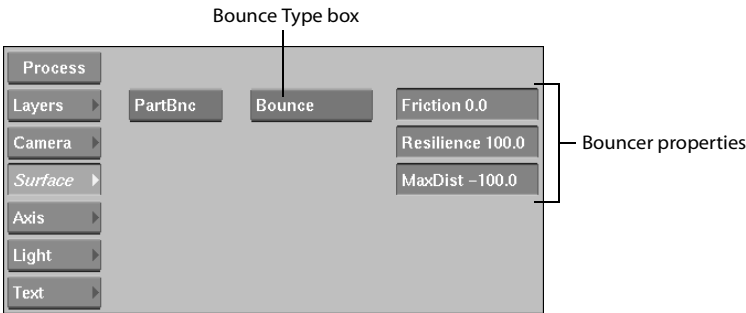


Working with Particle Bouncers

You can bounce a particle stream off either an image or a bouncer. When particles bounce on an image, they bounce off the surface wherever its matte is white or grey. Particles only pass through the areas of the image where its matte is pure black. If a surface's matte is turned off, particles bounce off the entire surface.

When you use a bouncer, it must be the parent of an axis or a light source. A bouncer shows up as a sphere in the Action scene when you use it with a light source. Otherwise, all other types of surfaces create flat bouncers.

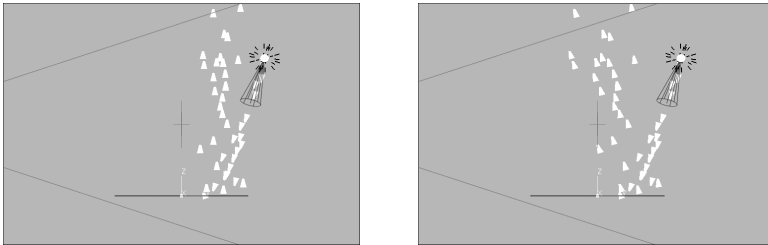
Whether you use a bouncer or an image, bouncing particles are controlled using the PartBnc menu in the Surface menu.



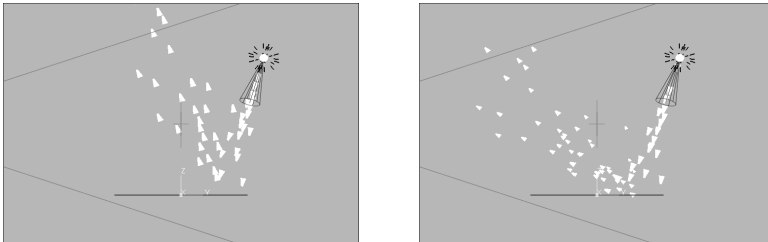
Select the type of bounce from the Bounce Type box. There are four types of bouncing that you can perform. Each corresponds to a different type of bouncing behaviour.

Select:	To:
Bounce	Bounce particles off images or bouncers. All bounce properties work with this type of bounce behaviour.
Extinct	Make particles disappear when they reach a surface. Only the MaxDist parameter works with this type of bounce.
Generator	Create an additional particle stream when particles bounce. When using this bouncing behaviour, you must add another particle generator. See "Particles on Particles" on page 1053.
Gen+Extinct	Make particles disappear and create an additional particle stream when particles reach a parented surface. Only the MaxDist parameter works with this type of bounce unless the particles end in another particle stream. See "Particles on Particles" on page 1053.

Friction field — Friction is a percentage that influences the angle of bouncing particles. The greater the percentage, the more particles bounce straight. For example, in the left figure, Friction is set to 100%, which bounces particles almost perfectly straight. In the figure on the right, Friction is set to 20%.



Resilience field — Resilience determines the amount of energy lost with each bounce. For example, at a Resilience of 100%, the figure on the left shows there is no energy lost between bounces. Each bounce is at the same height as the last bounce. When Resilience is set to 80%, 20% of energy is lost between each bounce. At 50%, the figure on the right shows the height of each bounce is reduced by 50%. Note that Friction is set to 0 for both examples.



MaxDist field — Use the MaxDist field to specify the maximum distance a particle travels past an image for it to still bounce or become extinct.

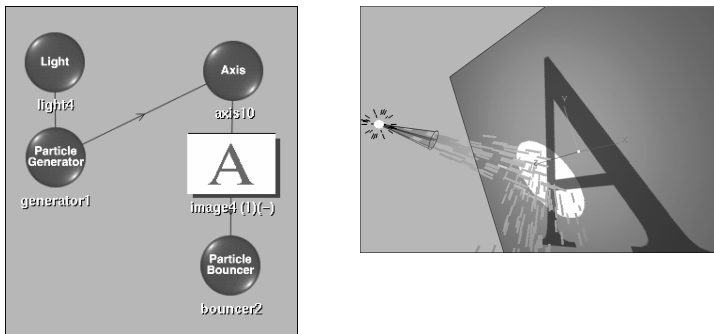
Because an image is only a pixel in width, it is unlikely that fast moving particles will hit the image directly. MaxDist acts like a buffer zone after the surface. If particles fall through the surface, set the MaxDist to -100. Particles that miss the surface by 100 pixels will still bounce.

If particles still fall through the surface, increase the timestep to increase the accuracy of the particle stream. See “Timesteps field” on page 1021.

Bouncing Particles Off an Image

You can bounce particles off an image surface only. Particles will bounce off bilinear and bicubic surfaces, but only on the original flat surface and not on any changes in shape.

To create the bouncing particle effect, you start with a particle generator. When you determine the image off which you want to bounce particles, you must parent the particle generator to the image; otherwise, the particles will ignore the image. The following figure shows the schematic and the result of bouncing particles off an image using a light source as a generator.



To bounce particles off an image:

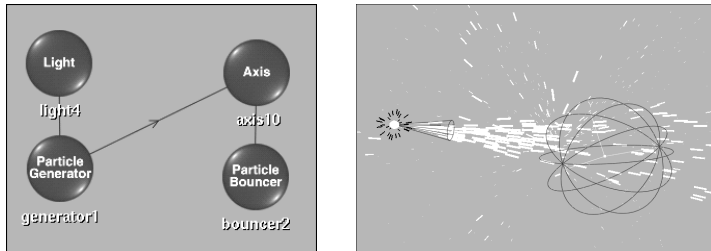
1. Create a particle generator using a light source, 3D object, or surface as a particle generator. See “Generating Particles” on page 1017.
2. Add an image off which you want to bounce particles.
3. Parent the particle generator to the image’s axis.
4. Select the image’s axis and in the Axis menu, enable Global.
Enabling Global in this way severs the parent-to-child relationship between the particle generator and the image, allowing you to place the image in the scene more easily. Any transformations and animations from the particle generator are not passed to the image.
5. Move the image in the scene until the particle stream passes through it.
To do this, you may have to switch views or orbit the camera.
6. In the schematic, select the image.
7. In the Object box, select PartBnc and click Add.
A particle bouncer is added and parented by the image.
8. In the Surface Particle Bouncer menu, select Bounce from the Bounce Type box and set the bounce properties.

Bouncing Particles Off a Bouncer

A particle bouncer element is added and automatically parented to a light source. The particle bouncer may not be parented by the right light source or axis. The bouncer must be parented by the light or axis added in the last step. If the particle bouncer is not parented correctly, unparent the particle bouncer and make the axis or light source added in the last step its parent.

NOTE: The shape of the bouncer depends on the particle emitter type.

The following figure shows the schematic and the result of bouncing particles off a bouncer.



To bounce particles off a bouncer:

1. Create a particle generator using a light source, 3D object, or surface as a particle generator. See “Generating Particles” on page 1017.
2. Add an axis or a light source to the scene.
3. In the Object box, select PartBnc and click Add.
4. If you are using an axis to set the position of the particle bouncer, select the axis and go to the Axis menu.
5. In the Axis menu, enable Global to sever the parent-to-child relationship between the particle generator and the axis.
Any transformations and animations from the particle generator are not passed to the axis. This makes it easier to place the particle bouncer in the scene.
6. Parent the particle generator to the particle bouncer’s axis or light source.
You must parent the particle generator to the axis or light source; otherwise, the particles will ignore the particle bouncer.
7. In the Surface Particle Bouncer menu, select Bounce from the Bounce Type box and set the appropriate bounce properties.

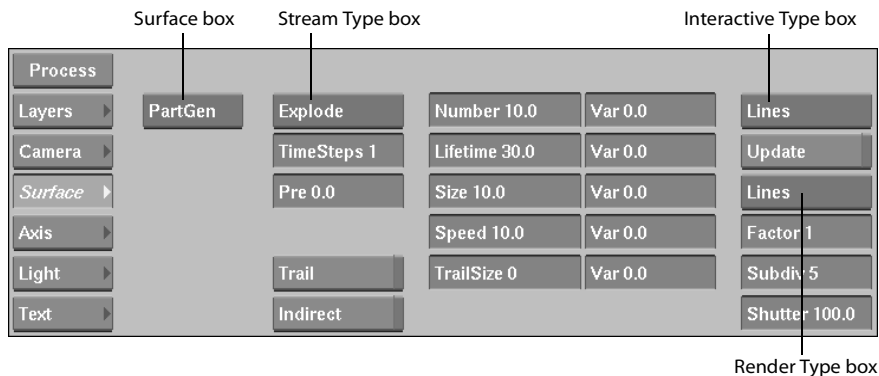
Exploding Objects

Whether you use a 3D object or a surface, exploding works much the same way. The shape of the 3D object or the surface is recreated by the particle generator, then the shape is exploded into polygons. For this to work correctly, the original 3D object or surface must be hidden.

The properties used to explode an object cannot be animated. When something explodes, it happens at a single point in time. This means that all of the settings in the Particle Generator menu are used at once, at the beginning of the explosion.

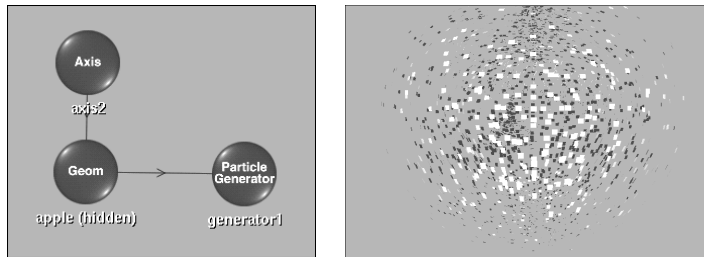
To explode a 3D object:

1. Import a 3D object or create 3D text, and in the Schematic view, select 3D object.
2. Add a particle generator and from the Surface menu Surface box, select PartGen.
3. In the Stream Type box, select Explode.



4. In the Render Type box, select Polygon as the particle type.
To view the explosion as you make changes to your particle stream, change the Interactive Type box to Polygon as well.
See “Setting Particle Rendering” on page 1023.
5. In the Number field, set the number of polygons.
Use the Number field to set the number of polygons to be removed from the 3D object with each pass. For example, if you specify 10 as the number of polygons per frame, then at each frame, 10 polygons are removed from the 3D object.
6. Set the other properties in the PartGen menu. See “Customizing the Particle Stream” on page 1021.
7. Hide the 3D object or 3D text.

The following figure shows the schematic and the result of exploding a 3D object of an apple. The number of polygons is set to 200. The result shows frame 20.

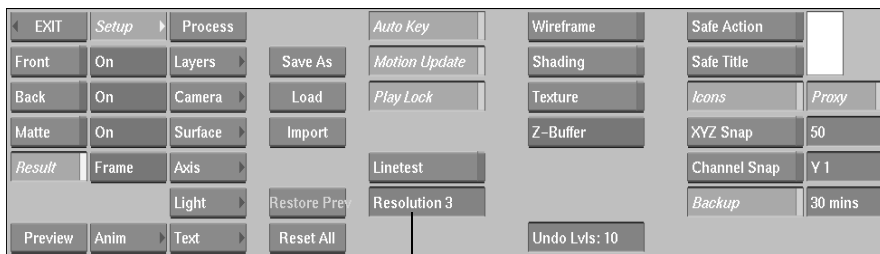


Exploding Surfaces

Exploding a surface is more complicated than exploding a 3D object. To explode a surface successfully, you must hide the original surface. In addition, to make the layer applied to the surface appear during the explosion, you must also apply a texture to the particle generator.

To explode a surface:

1. Select or add a surface and adjust its shape, position, rotation, and other attributes. See “Using Layers on Surfaces” on page 917.
2. Select the surface in Schematic view and add a particle generator.
A particle generator is automatically connected to the surface.
3. Go to the PartGen menu in the Surface menu and select Explode as the Stream Type.
4. Select Polygon as the particle type in the Render Type box. To view the explosion as you make changes to your particle stream, change the Interactive Type box to Polygon as well. See “Setting Particle Rendering” on page 1023.
5. Set the other properties in the PartGen menu. See “Customizing the Particle Stream” on page 1021.
6. In the Setup menu, adjust the resolution.



Resolution field

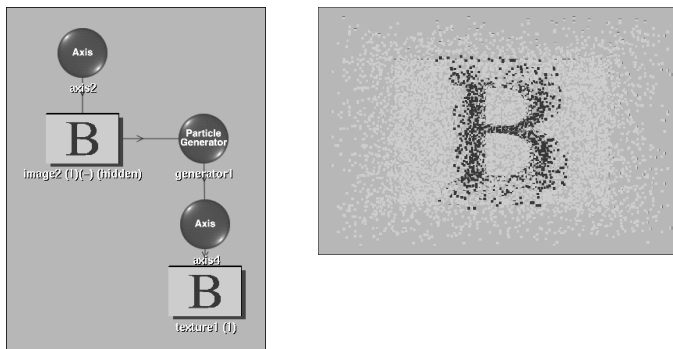
Resolution is used to set the size of the polygons exploded from the surface. The higher the resolution, the larger the polygon.

7. Hide the surface.
8. Select the particle generator in Schematic view.
9. Add a Texture object.

A Texture object appears in the schematic and is parented by the particle generator.

10. Double-click the texture and in the Surface texture menu, select Wrap from the Texture Mapping box.
11. Apply the appropriate layer to the Texture element and set the texture mapping.

The following figure shows the schematic and the result of exploding a surface. The number of polygons is set to 200. Resolution is set to 6, which is the default setting. Texture mapping is set to Wrap.



Particles on Particles

You can combine two particle streams or more by parenting them together. This has two possible results depending on whether the first particle stream ends or bounces off an object.

Particles Generating Another Particle Stream

When each particle in the particle stream ends, the second particle stream begins. This can be used, for example, to create fireworks.

To make particles generate other particles:

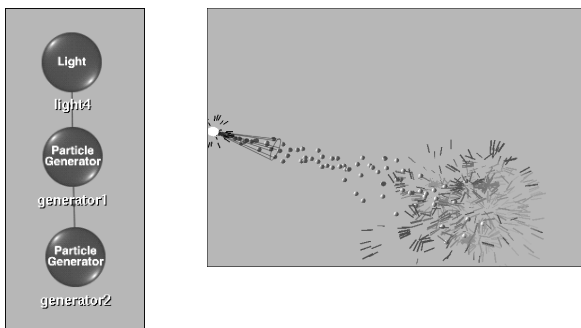
1. Create a particle generator.
You can use a light source, 3D object, or surface as a particle generator. See “Generating Particles” on page 1017.
2. In the schematic, select the Particle Generator.

3. Add another particle generator by selecting PartGen from the Object box and clicking Add.
4. In the Surface Particle Generator menu, set the second particle generator's other properties as appropriate.
5. In the Channel Editor, open the second particle generator folder to reveal the properties folder.



6. Set the Spread and Spread_V channels to control the spread of the second particle generator. Use the Spread channel to set the scope of the second particle stream. This is the same as using the Spread field in the Light menu for a generator that uses a light source. The Spread_V channel adds variance to the value in the Spread channel.

The following figure shows the schematic and the result of a particle stream that generates a second particle stream. The first particle stream generates spheres and the second particle stream generates quads.



Particles Bouncing into Another Particle Stream

When bouncing particle off a bouncer, you can generate a second stream of particles. For example, you this technique to give the effect of water or sparks bouncing off an image.

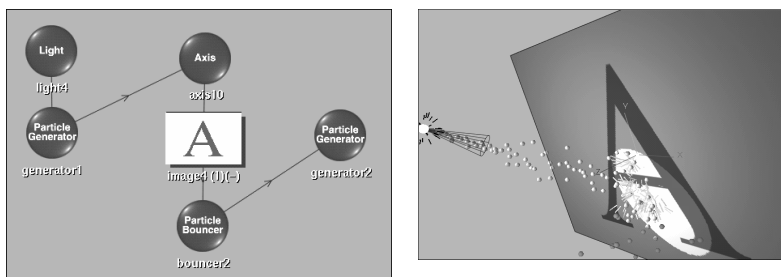
There are two different effects that you can create depending on your bouncer's settings. The following example illustrates the Generator bounce type.

To make bouncing particles generate another particle stream:

1. Create a schematic that bounces particles off an image or a bouncer. For instructions, see “Working with Particle Bouncers” on page 1047.
2. In the schematic, select the Bouncer object.
3. In the Surface Particle Bouncer menu, select Generator from the Bounce type box.
4. Add another particle generator by selecting PartGen from the Object box and clicking Add.
5. In the Surface Particle Generator menu, set the particle generator's other properties as appropriate.
6. In the Channel Editor, open the second particle generator folder to reveal the properties folder.
7. Set the Spread and Spread_V channels to control the spread of the second generator.

If you set the Spread to less than 180, you can also control the rotation of the second particle stream using the Friction field in the Surface Particle Bouncer menu.

The following figure shows the schematic and the result of bouncing particles into another particle stream where the first particle stream continues after bouncing off an image surface. The first particle stream generates spheres and the second particle stream generates quads.



Sample Particle Setups

Ten particle setups are provided in the directory `/usr/discreet/[product name]/example_setups`. To load one of the setup files, open Action using a black frame as the front, back, and matte clips.

A brief description of each example particle setup is provided in the following table.

Setup File Name	Description
<i>BounceExample</i>	Uses a gravity manipulator and a bouncer attached to an axis. The bouncer appears as a sphere in Perspective view. The particles fall down and bounce off the bouncer.
<i>BounceExample2</i>	Uses a gravity manipulator and two image surface bouncers.
<i>BounceExample3</i>	Uses an Accpoint manipulator and a bouncer linked to an axis. The bouncer follows the Accpoint manipulator.
<i>ExplodeExample</i>	An example of exploding 3D text. The Geom node is hidden in Schematic view. The setup uses an Explode generator and Polygon particles. To add 3D text (Geom node), go to the Text menu, click the text field and type the text string, then click Create.
<i>ExplodeExample2</i>	An example of exploding an image surface. The image surface is hidden. Layer1 is applied to the generator as a texture and uses Wrap mapping mode. Replace Layer1 with an image of your choice.
<i>FireWorksExample</i>	Uses two linked generators. The first generator generates few particles at each frame. When these reach the end of their lifetime, the second generator is activated.
<i>FunctionExample</i>	Uses a function manipulator that increases the particles' transparency as they reach the end of their lifetime.
<i>FunctionExample2</i>	Uses a function manipulator with sine and cosine functions to modify the speed of the particles.
<i>NodeExample</i>	An example of using 3D text for the particles. The Geom node (the 3D text) must be named "ParticleDraw". The generator is parented to the axis of the Geom node, and the selected particle type is Node.
<i>TransformExample</i>	Uses a transform manipulator.



Section 7: Retouching Clips

*Paint on the
image, add
filters, and
create special
effects.*

48

Paint: Overview

Put on your smock

Use Paint to create original images and to retouch clips. Use the anti-aliased brushes to apply colours, filters, and Special Effects media to the image. Create your own custom brushes for special applications.

Summary

The following chapters teach you how to use Paint to create images and apply effects to your images:

- Chapter 49, “Paint: Painting on the Canvas,” describes how to paint with brushes and geometry tools.
- Chapter 50, “Paint: Setting Brush Attributes and Modes,” describes how to set the attributes and modes that determine how paint is applied to the image.
- Chapter 51, “Paint: Using Filters and Special Effects Media,” describes how to apply filters and special effects to the image.
- Chapter 52, “Paint: Using Wipe, Fill, and AutoPaint,” describes how to use the Wipe, Fill, and AutoPaint commands.
- Chapter 53, “Paint: Using Mattes,” describes how to create and use mattes in Paint.
- Chapter 54, “Paint: Using Graphics,” describes the graphic tools used to add and edit objects on the image.
- Chapter 55, “Paint: Cut and Paste,” describes the cut and paste commands, and how to add drop shadows or embossing effects to the cutouts.
- Chapter 56, “Paint: Creating, Loading, and Saving Setups,” describes how to load and save setups in Paint, create custom brushes and brush sets, and apply colour correction setups to the clips.

About Paint

Use the Paint tools to create graphics, paint on images, and retouch clips. Use a matte to protect areas of the front clip during painting. Record, animate, and apply a series of brush strokes to a clip. Use the Graphic and Cut/Paste tools to create rotoscoped sequences and shape animations.

Accessing the Paint Menu

To access the Paint menu, click the Paint button in the Effects menu and select the clips that you want to use from the desktop reels. Any of the following combinations of clips can be used in Paint:

- A front clip only
- A front clip and a back clip
- A front clip and a matte clip
- A front clip, a back clip, and a matte clip
- None (a blank canvas)

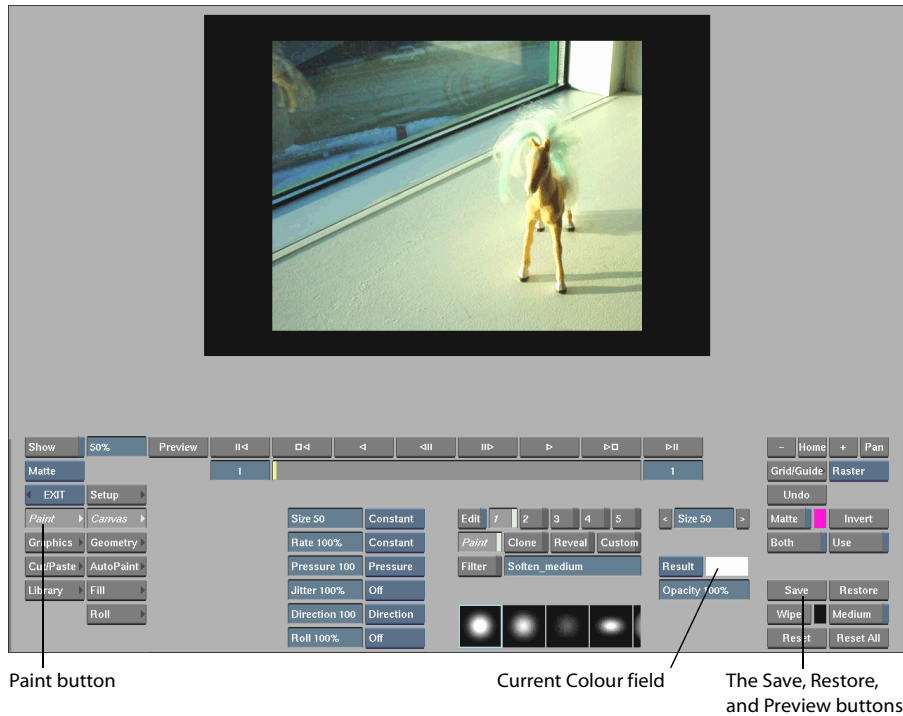
The front clip appears on the Paint canvas, which you can use to apply colours and effects to the clip. The matte clip delimits the area of the front clip that is affected by painting. The back clip can be revealed or brushed through onto the front clip. The colour of the blank canvas is defined by the wipe colour.

NOTE: The image window in Paint is referred to as the canvas in the following sections.

To access Paint:

1. Click the Paint button in the Effects menu.
The Input Mode box appears.
2. Select an option from the Input Mode box.
3. Select the source clips.
4. Select the destination reel.

The Paint menu appears.



Click:	To:
Paint	Use the brushes or geometric shapes to paint on the canvas, record and apply strokes to the canvas, fill areas of the image with a reference colour or image, and move the image on the canvas.
Graphics	Create, edit, and animate shapes and apply them to the canvas.
Cut/Paste	Create and add effects to cutouts and apply them to the canvas.
Library	Save and load your own colour palettes, mattes, pictures, cutouts, geometry, paint strokes, custom brushes, and stamps.
Setup	Set preferences, rendering options, grid guides, and colour correction options.

Paint Basics

The following section covers topics relevant to all Paint modules:

- Using the mouse (see “Using the Mouse” on page 1062)
- Using reference images (see “Displaying a Reference Image” on page 1062)
- Using the colour palette and selecting colours (see “Using the Colour Palette” on page 1063)
- Using grids and guides (see “Using Grids and Guides” on page 1065)
- Saving and restoring (see “Saving and Restoring the Image” on page 1067)
- Exiting and processing (see “Exiting Paint” on page 1068)

Using the Mouse

If you want to use the mouse instead of the tablet and stylus, press **SHIFT M** and **INSERT**. Press and hold on the right mouse button to apply paint at 80% pressure, the middle button to paint at 40% pressure, and the left button to paint at 20% pressure.

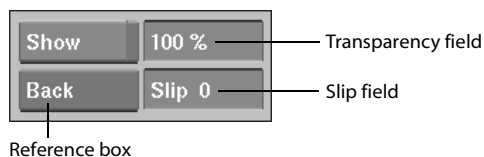
The tablet and stylus are capable of a far greater variety of pressure settings. To switch back to the tablet and stylus, press **SHIFT+T** and **INSERT**.

Displaying a Reference Image

You can display a reference image in the background of the result image to use as a guide for rotoscoping.

To display the reference image:

1. Enable the Show button.



The reference image appears by default at 50% transparency.

2. Select the reference image you want to display from the Reference box.
3. Set the transparency for the reference image in the Transparency field.
4. Set the Slip field value to show different images from the reference clip.

This option only works if the reference image you selected is from a clip with more than one frame.

NOTE: The Slip field does not appear if you select Matte or Saved from the Reference box.

Selecting Colours

In Paint, you select colours using the colour picker, colour palette, and colour gradient. You store your own colours in the colour palette, mix or try out colours on the scratch pad, and create colour or greyscale gradients using the colour gradient.

Click the Current Colour field to use the colour picker. For more information, see Chapter 4, “Fundamentals.”

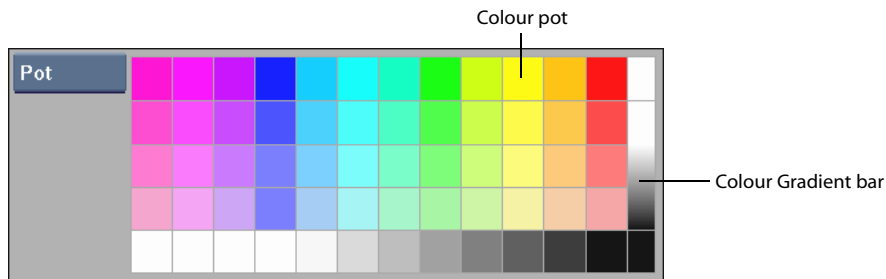
Using the Current Colour

The current colour is the colour used when you paint. The current colour is also used to set the colour for the wipe function and the matte colour. Select the current colour from the colour palette, scratch pad, or colour gradient.

Using the Colour Palette

The colour palette appears in the Paint, Graphics and Cut/Paste menus. It contains the scratch pad and colour gradient. To display the colour palette, swipe the cursor across the grey swipe bar at the bottom of the screen. To hide it, swipe the cursor across the swipe bar a second time.

A series of colours are stored in the colour palette in colour pots. To select the current colour in the colour palette, click a colour pot. To set the current colour, press and hold a colour pot.



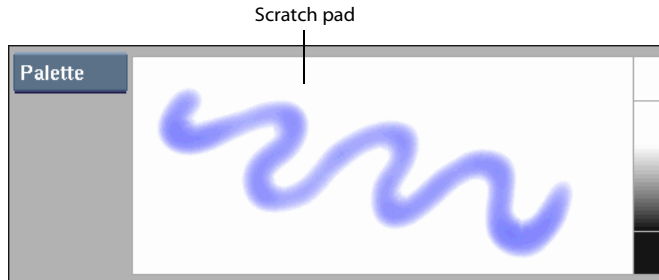
Build your own colour palette by storing the current colour in the colour pots. You can also save and load colour palettes. For more information, see Chapter 56, “Paint: Creating, Loading, and Saving Setups.”

To store a colour in the colour palette:

1. Click the Pal (Palette) button if the scratch pad is displayed.
The colour palette appears.
2. Set the current colour.
3. Click on a colour pot in the colour palette and hold the mouse button down momentarily.
The colour is saved in the colour pots.

Using the Scratch Pad

Use the scratch pad to mix colours selected from the image or colour pots and to test the selected brush type. You can also paste cutouts into the scratch pad to use when mixing colours.



To mix colours on the scratch pad:

1. Click the Pot button.
The scratch pad appears.
2. Paint on the scratch pad.
3. Change the current colour and paint over the previous brush strokes.
The colours are mixed.

You can also use the Wash, Shade, Smear, Drag, Warp, Recursive Clone, and Stamp Special Effects media in the scratch pad.

To use one of the media types in the scratch pad:

1. Select the Special Effects medium you want to use.
For more information, see Chapter 51, “Paint: Using Filters and Special Effects Media.”
2. Click and hold the M button.
3. Use the brush cursor to apply the medium in the scratch pad.

To set the current colour using the scratch pad:

1. Click the current colour field.
2. Use the colour picker to select the mixed colour within the scratch pad.
3. Click the current colour field to use the selected colour.

Using the Colour Gradient Bar

The colour gradient bar is used to set the gradients for graphics you create in the Graphics menu. For more information, see Chapter 54, “Paint: Using Graphics.” You can also set the current colour by selecting a colour on the gradient using the colour picker.

To set the gradient:

1. Set the current colour.
2. Click either the upper or lower colour pot on the gradient bar.

NOTE: You must hold the cursor down for a moment to set the Gradient colour pot.

To set the current colour using the colour gradient bar:

1. Click and drag the cursor across the gradient bar.
2. The selected colour appears in the Current Colour field.
3. Select a colour.

The selected colour becomes the current colour.

Using Grids and Guides

Use the Grid/Guides button to set up reference points when painting and accurately place strokes on image.

Use the field guides to provide reference points when painting, and the grid to help you accurately place strokes on the image. Neither the field guides nor the grids appear on the processed clip.

For more information on using grids and guides, see Chapter 4, “Fundamentals.”

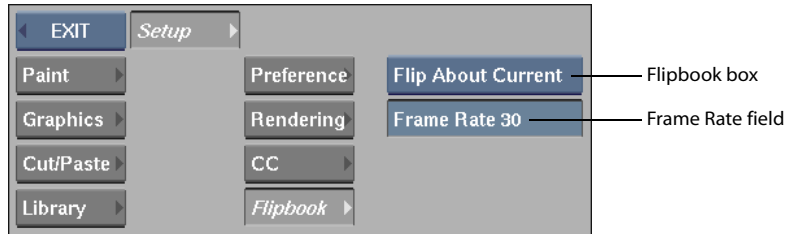
The Flipbook

Use the Flipbook command to preview an animation by playing a sequence of either five or nine frames.

To set up the Flipbook command:

1. Click the Setup button.
The Setup menu appears.
2. Click the Flipbook button.
The Flipbook menu appears.

3. Select an animation mode from the Flipbook box.



Select:	To:
Flip About Current	Play two (or four) frames before the current frame and two (or four) after the current frame. This is the default setting.
Flip From Current	Play five (or nine) frames beginning at the current frame.
Flip To Current	Play five (or nine) frames ending at the current frame.

4. Enter the frame rate in the Frame Rate field.

NOTE: If the system is running at high resolution, it may not be able to achieve a flip rate of 30 frames per second.

Press **F** to play five frames, or press **SHIFT+F** to play nine frames.

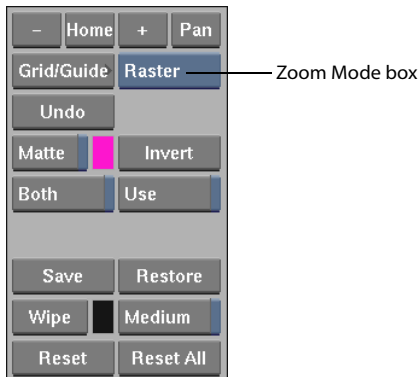
If you select Flip About Current, pressing **F** plays two frames before and after the current frame. Pressing **SHIFT+F** plays four frames before and after the current frame.

Painting on Full-Resolution Film Images

Use the Zoom Mode box to paint on a film resolution image at full-resolution from a zoomed out perspective. With this feature, you can perform complex operations like full-frame rotoscoping in Paint without having to constantly zoom in and out from the image on which you are painting.

In the Paint menu, the Zoom Mode box shows the Zoom mode in which you are working. This mode is set automatically by **flame**; however, you can also manually select either of these modes if you want to override the default mode. If you zoom out from the image, the Zoom mode will

automatically switch from Raster to Tiled. This allows you to paint on the image from a zoomed out perspective.



NOTE: If you manually switch the Zoom mode to Raster while zoomed out from the image, you will not be able to paint on it.

There are several advantages to manually switching this mode: In Raster mode, if you apply paint to the edge of the image while zoomed in, the paint is applied in an abruptly sharp straight edge along the border of the canvas; the brush stroke is not completed on the image. In Tiled mode, if you attempt the same operation, the paint is applied from the full diameter of the brush you are using to the area of the image that is not visible on the canvas.

Saving and Restoring the Image

You can save the current frame on the canvas. If you want to reverse a paint application, you can restore the image without having to recreate it.

The Save Button

Click Save to save the image that currently appears on the canvas. A single frame is stored in the Save buffer. The next time you click Save, the current frame replaces the frame stored in the Save buffer.

The Restore Command

Click Restore to replace the current image on the canvas with the contents of the Save buffer.

The Preview Command

Select the Saved option in the Source box and click and hold Preview to view the contents of the Save buffer.

Exiting Paint

When you exit Paint, you may keep or discard the changes have made to the front clip. Click Exit and select an option.

Select:	To:
Exit	Keep the changes you made to the front clip. The modified front clip appears on the destination reel.
Keep One	Keep only the current frame of the front clip. The current frame appears on the destination reel.
Cancel	Quit Paint without saving changes to the front clip. When this option is selected, a Confirm button appears to the right of the canvas. Click Confirm to quit without saving your changes, or click elsewhere to cancel.

Paint: Painting on the Canvas

Mark it up

The canvas is the portion of your work area where you can apply effects to your images. Add colour, filters, and Special Effects media to the image on the canvas.

Summary

In this chapter, you learn about:

- “Selecting a Brush” on page 1070
- “Rolling the Image” on page 1071
- “Painting Straight Lines” on page 1072
- “Painting with Geometry” on page 1072

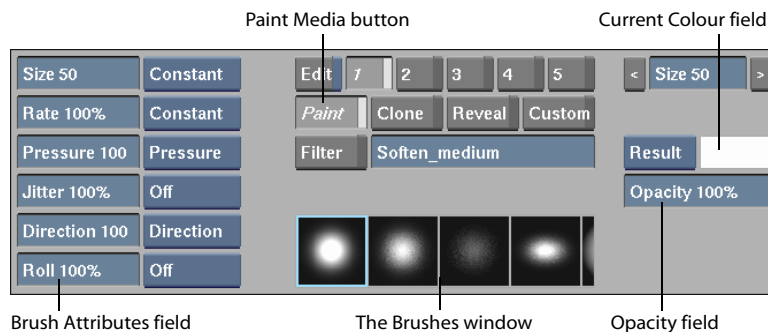
About Canvas Mode

When you open Paint, you are in Canvas mode. You can use brushes to modify your images. While in Canvas mode, the brush cursor appears as a green cross surrounded by a circle when placed over the canvas. Use this brush to apply colour, filters, and Special Effects media to the image on the canvas.

To paint on the canvas:

1. Click Paint.

The Paint menu appears.



2. Click Canvas.

If the scratch pad is covering this button, swipe down to hide the scratch pad.

3. Click Paint.

4. Set a colour in the Current Colour field.

For more information, see Chapter 48, “Paint: Overview.”

5. Select a brush from the Brushes window.

For more information, see “Selecting a Brush” on page 1070.

6. Set the brush attributes and modes in the Brush Attributes field.

For more information, see Chapter 50, “Paint: Setting Brush Attributes and Modes.”

7. Set the brush opacity in the Opacity field. Set the opacity to 100% to apply a fully opaque colour.

8. Stroke the brush over the canvas.

9. Click Undo to erase the stroke(s) applied to the canvas since the last time you zoomed, panned, or changed a brush attribute. Alternatively, click **CTRL+Z**.

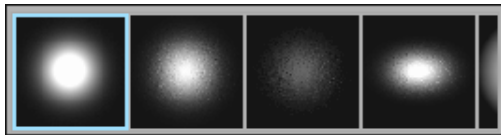
Selecting a Brush

A number of predefined brush types are available in Paint including round, square, and elliptical brushes of various sizes and edge softness. The cursor has the same appearance regardless of which brush type you are using. Each brush has its own icon in the Brushes window. The white portions of the brush icon indicate where the paint will be applied onto the image.

For information on creating your own brush, see Chapter 56, “Paint: Creating, Loading, and Saving Setups.”

To select a brush:

1. Scroll through the Brushes window.



To scroll the selections, click on the Brushes window and drag left or right. Alternatively, you can click on either the left or right border and hold down the cursor.

Use the left mouse button to scroll slowly, the middle button to scroll faster, and the right button to scroll very quickly.

2. Click on the brush icon you want to use.

The selected brush is highlighted by a blue outline.

NOTE: Only one brush can be active at a time.

Using the Large Canvas

In Large Canvas mode, almost the entire image window is available for painting, but not all Paint options are available. You can still change the brush characteristics, medium, and show options. You can also save, restore, and wipe the canvas.

To display the large canvas:

1. Click Paint.
2. Click Canvas.
3. Swipe your cursor anywhere on the right edge of the screen.

You can also toggle between the large canvas and Paint menu by pressing **ESC**.

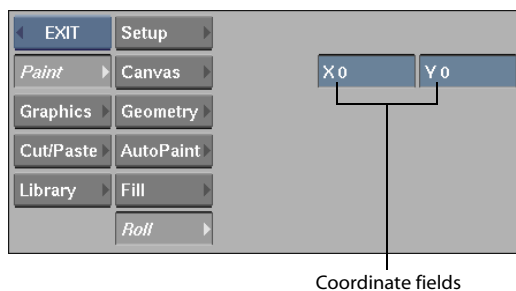
Rolling the Image

Use the Roll command to reposition the current frame on the canvas to paint its edges.

To roll the image:

1. Click Paint.
2. Click Roll.

The Roll menu appears.



3. Zoom in on the image if necessary.
4. Drag the image using the pan cursor.
You can also enter the roll values in the Coordinate fields.
5. Click Reset to recentre the frame.

NOTE: The frame is automatically recentred when you go to another frame or exit Paint.

Painting Straight Lines

You can paint straight horizontal and vertical lines in Canvas mode by pressing the **SHIFT** key and dragging the brush up and down or left and right.

Painting with Geometry

Use the Geometry feature to draw lines, rectangles, circles, and triangles on the canvas.

Painting with geometry is different from drawing objects in Graphics mode. In Graphics mode, you create objects that can be edited. When you paint with geometry, you define paths for the brush to follow. Paint uses the current brush setup to apply the stroke to the canvas.

Painting Lines

Use the Line option to paint straight lines. Paint single lines one at a time, or draw connected multiple lines. You can constrain the lines to vertical or horizontal paths or specific angles.

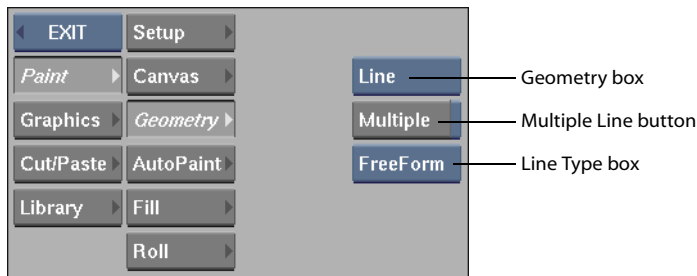
To paint a line or a multiple line:

1. Click Geometry in the Paint menu.

The Geometry box appears.

2. Select Line from the Geometry box.

The Multiple Line button and the Line Type box appear.



3. To draw connected multiple lines, enable Multiple Line.
4. Select the type of line you want to draw from the Line Type box.

Select:	To:
Angle	Paint a line at a specific angle. This option displays a box in which you enter the angle in degrees.
Horizontal	Paint horizontal lines.
Vertical	Paint vertical lines.
FreeForm	Paint lines at any angle with no constraint.

5. If you selected Angle, enter a degree in the box.
6. To draw single lines, move to the canvas and click, drag and release.
A brush stroke is painted along the line.
7. To draw multiple lines, move to the canvas and click to place the start point of the first line. Click again to draw the end point. Continue clicking to place additional points and draw more lines.
8. To end multiple lines, click below the timeline or on the menu panel to turn the option off.
The brush strokes are painted along the lines.

Painting Rectangles

Use the Rectangle option to draw rectangles or squares.

To paint a rectangle or square:

1. Click Geometry.
The Geometry box appears.
2. Select Rectangle from the Geometry box.
The Equal Sides button appears.



3. To draw a square, enable Equal Sides or press and hold **P**.
4. Press the cursor on the canvas to anchor the first corner of the rectangle. Drag the cursor to size the rectangle.
5. When the rectangle is the size you want, release the cursor.
A brush stroke is painted along the sides of the rectangle.

Painting Circles

Use the Circle option to draw circles of any size.

To paint a circle:

1. Click Geometry.
The Geometry box appears.
2. Select Circle from the Geometry box.
3. Press the cursor on the canvas to anchor the centre point of the circle. Drag the cursor to size the circle.
4. When the circle is the desired size, release the cursor.
A circular brush stroke is painted.

Painting Triangles

Use the Triangle option to draw equilateral or asymmetric triangles.

To paint a triangle:

1. Click the Geometry button.
The Geometry box appears.
2. Select Triangle from the Geometry box.
The Equal Sides button appears.



3. To draw an equilateral triangle, enable Equal Sides or press and hold **P**.
4. Move to the canvas and click to anchor the first vertex of the triangle.
5. Click to place the second vertex and then again to place the third vertex. If you are drawing an equilateral triangle, drag the cursor until the triangle is the required size.
A brush stroke is painted along the sides of the triangle.

Paint: Setting Brush Attributes and Modes

Brush up

Use brush attributes to customize brush pressure, movement, size, direction, and shape. Control how paint is applied when creating or touching up images.

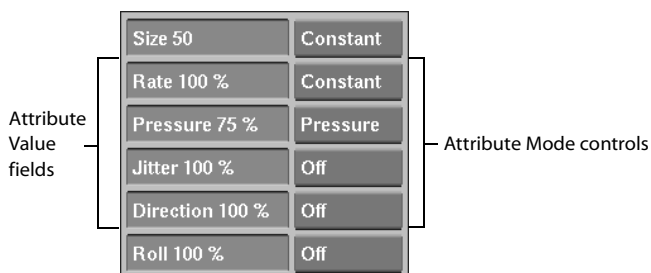
Summary

In this chapter, you learn about:

- “Brush Attributes” on page 1076
- “Brush Modes” on page 1080
- “Preferences Affecting Brushes” on page 1082

About Brush Attributes and Modes

You can set various brush attributes and attribute modes that determine how paint is applied to the image. You use the Preference menu to affect the way paint is applied to the image.



In the Paint menu, the Brush Attributes and Attribute Mode controls share the same space as the colour palette. To display the brush attributes and modes in the Paint menu, swipe the cursor at the bottom of the screen.

In the Graphics Edit submenu, the Brush Attributes and Mode controls appear only when the object attribute is set to Outline or Fuzzy. To display the brush attributes and modes in the Graphics menu, swipe the cursor twice at the bottom of the screen.

NOTE: The brush attribute and attribute mode values set in the Paint menu are independent from values set in the Graphics menu.

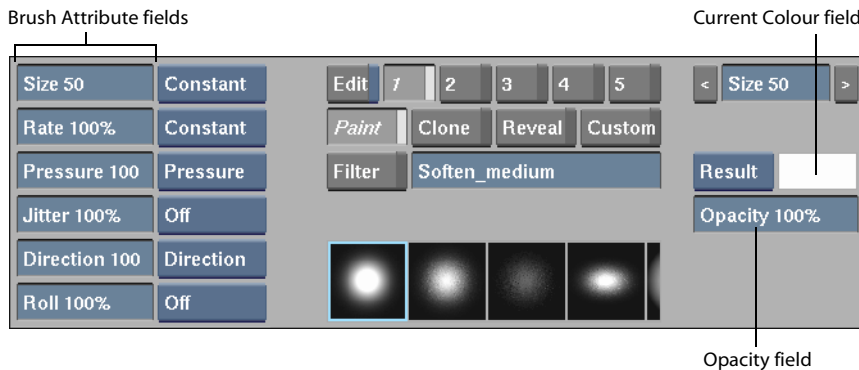
Brush Attributes

The Brush Attribute fields set the size of the brush, the distribution of the paint, and the rate and direction of the paint application. There are six brush attributes:

- Opacity
- Size
- Rate
- Pressure
- Jitter
- Direction
- Roll

Brush Opacity

The brush opacity affects the transparency of the brush. A value of 100% applies a fully opaque colour. Use a lower value to apply a more transparent colour. Enter a value in the Opacity field.

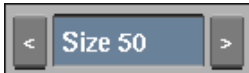


NOTE: You can only use the Front, Back, Result and Saved attribute modes with the Opacity attribute.

Brush Size

The brush size field affects the size of the brush applied to image. The diameter of the green dashed circle surrounding the cursor brush reflects the brush size. To increase the brush size, hold **S**, press and drag the brush to the right. To decrease it, hold **S**, press and drag to the left.

You can also use the Size buttons to increase or decrease the brush size. Click the > button to increase the brush size. Click the < button to decrease the brush size. You can also click the Size field and enter a value.

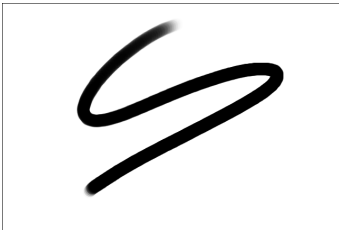


NOTE: You can use any of the attribute modes with the Size attribute.

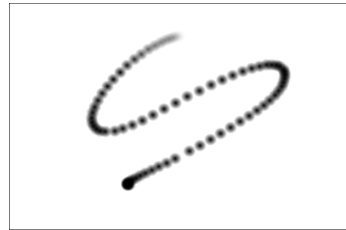
Brush Rate

The brush rate is the rate at which brush strokes are applied to the canvas. Use a high value to produce a smooth continuous stroke, or a low value to produce a less continuous stroke with larger gaps between brush images.

NOTE: You can use any of the attribute modes with the Rate attribute.



Rate attribute value = 100



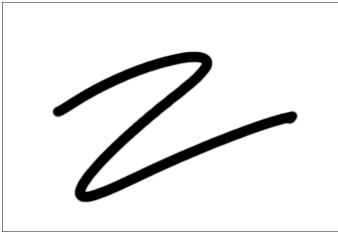
Rate attribute value = 25

Brush Pressure

The brush pressure affects the transparency of the paint applied to the image. To apply opaque paint, use a high percentage value. For more transparent paint, use a lower value.

The Pressure attribute is different than the Opacity attribute in that you can set the Pressure Attribute mode so that the paint transparency varies according to the pressure applied to the stylus, or the direction of the brush.

NOTE: You can use any of the attribute modes with the Pressure attribute.



Pressure attribute value at 100%

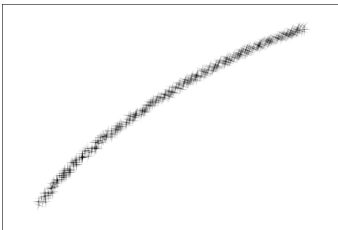


Pressure attribute value at 30%

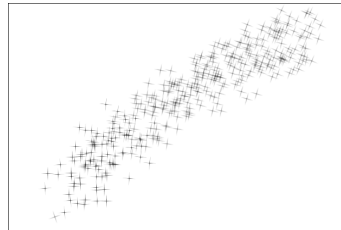
Brush Jitter

The Jitter attribute randomizes the brush strokes applied to the image. A high value produces a greater dispersion of paint, while a low value produces a greater concentration.

NOTE: You can use any attribute mode with the Jitter attribute.



Jitter attribute value at 10



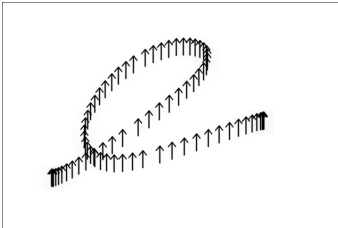
Jitter attribute value at 100

Brush Direction

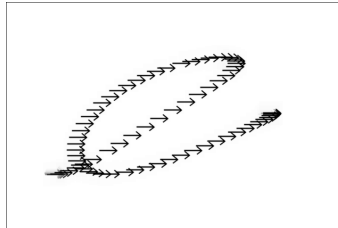
The Direction attribute causes the brush to rotate around the Z-axis, and can be used to produce a calligraphy effect. The effect of the Direction attribute is most noticeable when used with one of the elliptical or star brushes.

The value of the Direction attribute causes the brush strokes to rotate by 90 degrees for each increment of 25 percent.

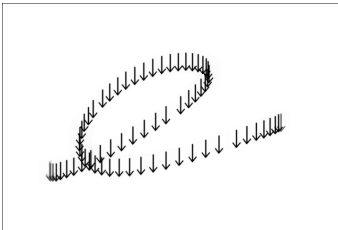
NOTE: You can use any attribute mode with the Direction attribute.



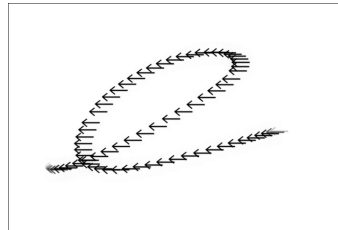
Direction attribute at 100%



Direction attribute at 75%



Direction attribute at 50%



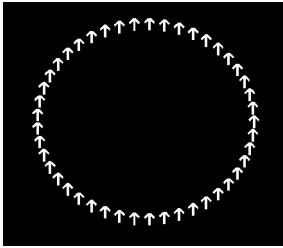
Direction attribute at 25%

Brush Roll

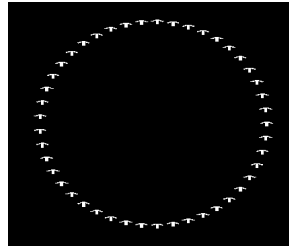
The Roll attribute rolls the brush around the horizontal or X-axis. The effect of the roll is most noticeable when used with one of the non-symmetrical brushes.

Each increment of 25 percent as the Roll attribute value creates a rolled brush stroke of 90 degrees.

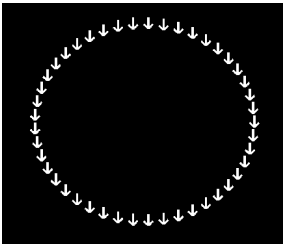
NOTE: You can use any attribute mode with the Roll attribute.



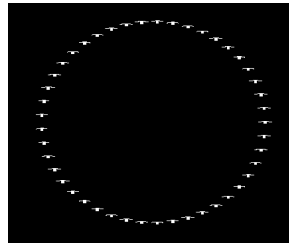
Roll attribute value at 100%



Roll attribute value at 75%



Roll attribute value at 50%



Roll attribute value at 25%

Brush Modes

The value of a brush attribute depends on the brush attribute mode. In Constant mode, the brush attribute values remain constant. In Off mode, the attribute is disabled.

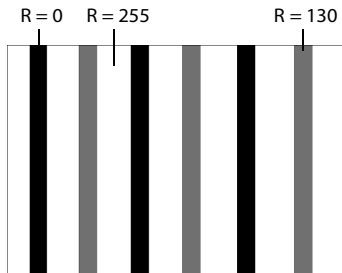
NOTE: Do not set the size, rate, or pressure attributes to Off mode.

The Front, Back, Result, Saved, Pressure, and Direction modes affect how paint is applied by causing the Brush Attribute value to vary in relation to one of the following reference values:

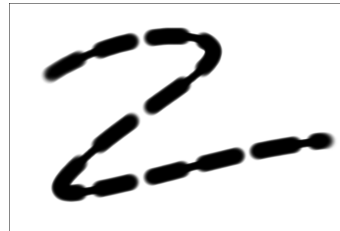
- The colour values in a reference image
- The direction of the brush
- The pressure exerted on the stylus

Using Reference Images

The Front, Back, Result, and Saved modes use the red channel in the reference images to set the brush attribute values. The Front mode uses the front clip as the reference, the Back mode uses the back clip, the Result mode uses result clip, and the Saved mode uses the image in the Save buffer.



Back image



Using Size attribute with mode set to Back

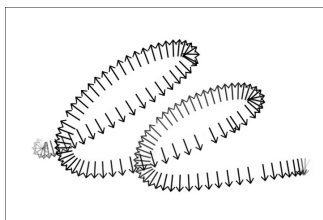
Direction of the Brush

The Direction attribute mode uses the direction of the brush stroke as the reference.

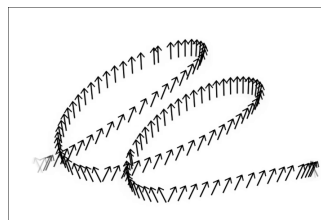
Using Direction Mode

When used with the Direction mode, the Direction attribute causes the brush strokes to follow the trajectory of the brush. Increase the percentage value to enhance the effect on brush direction.

Drag:	To:
Right	Use 100% of the attribute value.
Left	Use 0% of the attribute value.
Up	Use 25% of the attribute value.
Down	Use 75% of the attribute value.



Direction attribute value at 100%, Direction mode



Direction attribute value at 15%, Direction mode

Pressure Exerted on the Stylus

The Pressure attribute mode uses the pressure exerted on the stylus as the reference value. The harder you press on the stylus, the greater the brush attribute value. The softer you press on the stylus, the lower the brush attribute value.

NOTE: The Pressure attribute mode cannot be used in the Graphics menu.

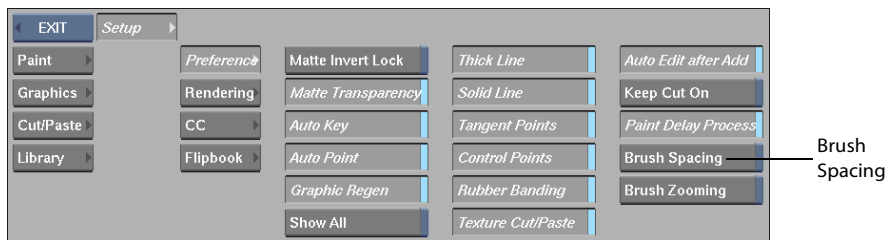
Preferences Affecting Brushes

The Brush Spacing setup preference affects how paint and brush strokes are applied to the canvas.

To enable Brush Spacing:

1. Click Setup in the Paint menu.
2. Click Preferences.

The Preferences menu appears.



3. Enable Brush Spacing.

This option sets a uniform distance between paint strokes. No matter how fast you move the brush, the brush spreads the paint evenly.

HINT: Use a high brush rate when using stamps with the brush spacing option enabled.

Paint: Using Filters and Special Effects Media

Wash up

Use filters to add textures and effects such as emboss and soften to your entire image or to a specific area of your image. Use Special Effects media to wipe or brush on effects such as warp, blur and clone.

Summary

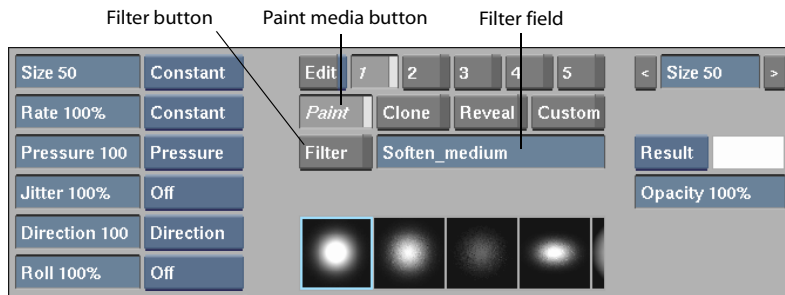
In this chapter, you learn about:

- “Using Special Effects Media” on page 1085
- “Blurring the Image” on page 1086
- “Cloning the Image” on page 1087
- “Dragging the Image” on page 1088
- “Using the Impressionist Medium” on page 1088
- “Revealing a Reference Image” on page 1089
- “Using the Recursive Clone Medium” on page 1090
- “Smearing the Image” on page 1091
- “Using the Stamp Medium” on page 1091
- “Warping the Image” on page 1092
- “Washing and Shading the Image” on page 1093

About Filters

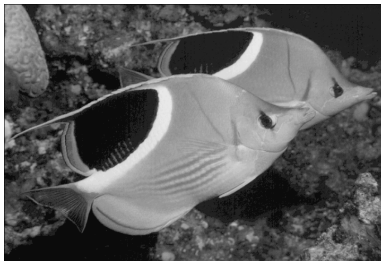
When applying filters and Special Effects media, it is important to try different brush types, and to change the brush attributes to create different effects.

Apply filters to the canvas using the brush, Wipe command, Wash or Shade media. Paint uses the same filter library as the Filter command in the Processing module.

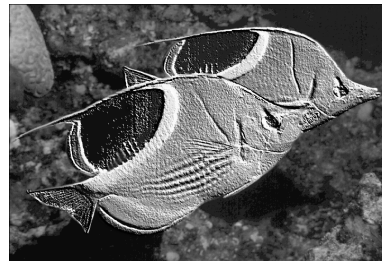


To brush a filter onto the image:

1. Click the Filter field.
The filter library appears.
2. Select the filter you want to use from the filter library.
You are returned to the Paint menu and the filter name appears in the Filter field.
3. Click Filter to enable the selected filter.
4. Set the brush opacity. The opacity determines the level of filtering. Reduce the opacity value to reduce the level of filtering.
5. Paint on the image.
6. Click Paint media or select one of the other Special Effects media to disable the filter.



Original image

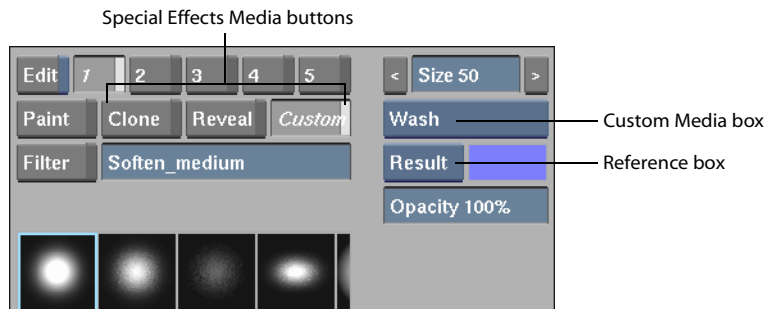


After using the emboss filter

Using Special Effects Media

Special Effects media can be brushed onto the image in Canvas mode. You can apply the Wash, Shade, and Reveal media to the entire image using the Wipe command. For more information, see Chapter 52, “Paint: Using Wipe, Fill, and AutoPaint.” Only one media type can be used at a time.

Use the media buttons to enable the Special Effects media. The Paint, Clone, and Reveal media are enabled using buttons. Other media types are enabled using the Custom Media box.



The following Special Effects media are available:

- Blur is used to apply a blur filter to portions of the image.
- Clone is used to copy a portion of the image to a new location.
- Drag is used to create an image trail from a selected region of the screen.
- Impressionist is used to paint on colours from a reference clip.
- Recursive Clone makes a number of copies of a selected area of the image.
- Reveal brushes a reference image onto the current image.
- Shade darkens or lightens the image depending on the luminance value of the current colour.
- Smear is used to smudge areas of the image.
- Stamp applies a captured image to the image.
- Warp is used to stretch and distort a region of the image.
- Wash applies a transparent wash of the current colour to the image.

Blurring the Image

Use the Blur medium to blur portions of the image.

To blur the image:

1. Click Custom and select Blur from the Custom Media box.

The Blur option boxes appear.



2. Click Canvas.
3. Set the brush size according to the size of the area that you want to blur.
4. Select either a Box or Gaussian filter to perform the blur from the Filter box.
For more information on filters, see Chapter 28, "Filters."
5. Set the density of the Blur brush from the Blur Density box.
You can choose light, medium, or heavy.
6. Drag the brush on the image.



Original image



After using the Blur medium

Cloning the Image

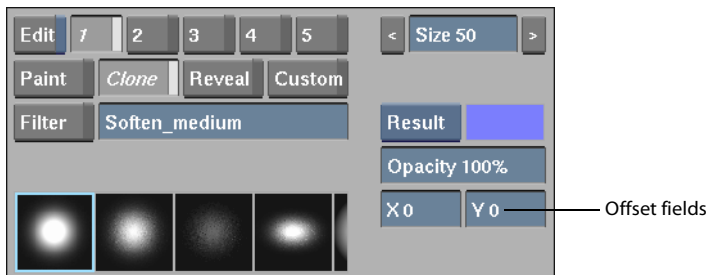
Use the Clone medium to copy a region of the image and paint it on a destination point. The result image is used as the source for the Clone medium.

The offset between the origin point and the destination point is set in the Offset fields.

To paint using the Clone medium:

1. Click the Clone media button.

The Clone option boxes appear.



HINT: Press **CTRL** and drag the cursor to set the destination point.

2. Click Canvas.
3. Set the brush opacity. The brush opacity determines the transparency of the clone. When the opacity value is set to 100%, the clone is completely opaque.

4. Set the brush size.
5. Set the offset between the origin point and the destination point in the Offset fields.

A tracking circle appears at the offset co-ordinates you specified; this is the destination point for the cloned image.

6. Paint on the image.

The image contained within the green circle brush is copied to the region defined by the red circle.



Original image



After using the Clone medium

Dragging the Image

Use the Drag medium to drag a selected area of the image across the canvas. The selected area is painted on the canvas as you drag the brush, creating an image trail.

NOTE: The Direction brush attribute cannot be used with the Drag medium.

To use the Drag medium:

1. Click Custom and select Drag from the Custom Media box.
2. Click Canvas.
3. Set the brush size.
4. Position the brush over the region of the image that you want to use. Hold down the cursor to select that region.
5. Drag the brush.

The selected region is painted onto the image as you drag. The density of the image trail is dependent on the pressure of the brush. For more information on the pressure brush attribute, see Chapter 50, “Paint: Setting Brush Attributes and Modes.”



Original image



After using the Drag medium

Using the Impressionist Medium

Use the Impressionist medium to brush on colours from a reference clip. The brush opacity determines how much colour is taken from the reference clip. When the opacity value is set to 100%, the colour is taken from the reference image. At 50%, the colour applied is a 50/50 blend of the reference colour and the result image.

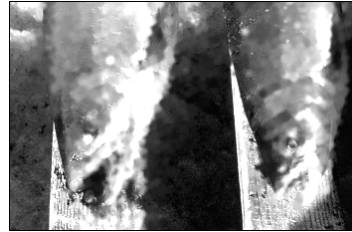
To use the Impressionist medium:

1. Click Custom and select Impressionist from the Custom Media box.
2. Click Canvas.
3. Set the brush opacity.
4. Select a reference image from the Reference box.
Select Front, Back, Saved, or Result.

5. Paint on the image.



Original image



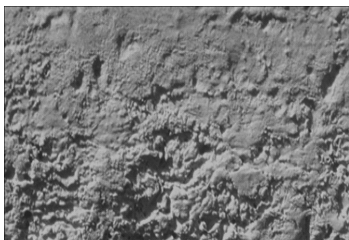
After using the Impressionist medium

Revealing a Reference Image

Use the Reveal medium to reveal portions of an image or an entire reference image. You can reveal specific areas using the brush, or the entire image using the Wipe command. The opacity of the brush determines how much of the reference image is revealed on the image. When the opacity is set to 100%, the reference image applied is completely opaque. At 50%, the reference image applied is a 50/50 blend of the reference image and the result clip.

To reveal a reference image on the canvas:

1. Click Reveal.
2. Click Canvas.
3. Set the brush opacity.
4. Select a reference image from the Reference box.
5. Paint on the image.



Back Image



Result Image



The Back image revealed on the Result image

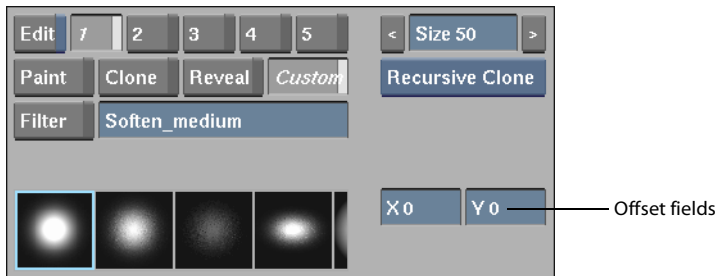
Using the Recursive Clone Medium

Use the Recursive Clone medium to make multiple copies of an area of an image. The result image and the paint applied to the result image is used as the source. Each copy is a degraded version of its predecessor. The offset between the origin point and the destination point is set in the Offset fields.

To paint using the Clone medium:

1. Click Custom and select Recursive Clone from the Custom Media box.

The Recursive Clone options appear.



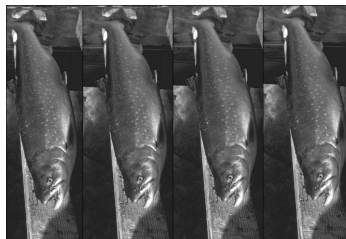
2. Click Canvas.
3. Set the offset between the origin point and the destination point in the Offset fields.
A red tracking circle appears around the destination point.

4. Paint on the image.

The image contained within the green circle brush is copied to the region defined by the red circle.



Original image



After using the Recursive Clone medium

Smearing the Image

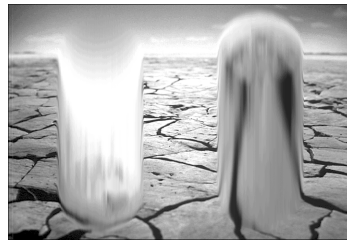
Use the Smear medium to smudge portions of the image.

To smear the image:

1. Click Custom and select Smear from the Custom Media box.
2. Click Canvas.
3. Drag the brush over the area that you want to smear.



Original image



After using the Smear medium

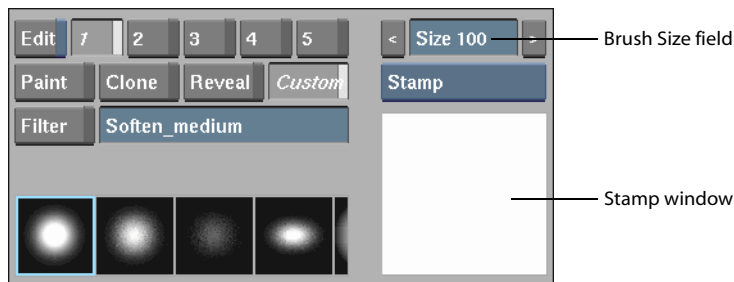
Using the Stamp Medium

Use the Stamp medium to capture a portion of the image and apply it to the canvas.

To capture and apply a stamp:

1. Click Custom and select Stamp from the Custom Media box.

The Stamp window appears beneath the Brush Size field.



2. Click Canvas.
3. Set the brush size. Use a small brush to isolate a specific detail of the image. Use a large brush to capture a bigger sample.
4. Click and hold the cursor on the Stamp window.
The colour picker appears.
5. Without releasing the cursor, move it over the image.

The Stamp window is updated as you move the cursor across the image.

6. Release the cursor when the Stamp window contains the part of the image you want to capture.

The brush cursor appears.

7. Paint on the image.

Saving and Loading Stamps

You can save a stamp and load it in another session to use with a different clip. For more information and saving and loading setups, see Chapter 56, “Paint: Creating, Loading, and Saving Setups.”

Warping the Image

Use the Warp medium to stretch and distort regions of the image.

To warp the image:

1. Click Custom and select Warp from the Custom Media box.
2. Click Canvas.
3. Set the brush size.

The area that can be warped is determined by the brush size.

4. Select a portion of the image and hold down and drag the cursor to warp the selection.



Original image



After using the Warp medium

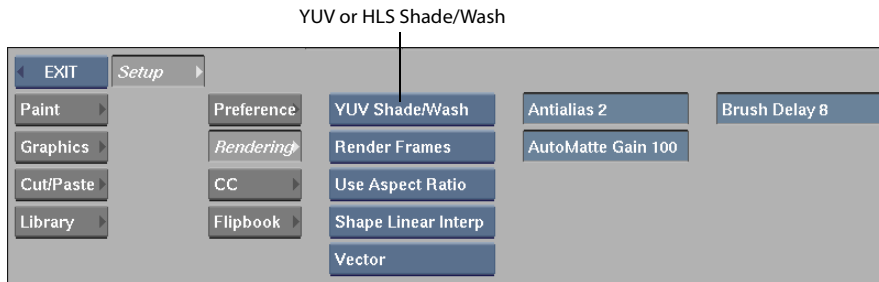
Washing and Shading the Image

Use the Wash medium to apply a transparent wash of the current colour to the image. Use the Shade medium to darken or lighten the image. You can wash or shade specific areas using the brush, or the entire image using the Wipe command.

With the Shade medium, you darken the image by using a colour with a low luminance value, and lighten the image by using a colour with a high luminance value. The opacity of the brush affects the transparency of the paint applied to the image. When the opacity value is set to 100%, the paint applied is completely opaque. As you decrease the opacity, the paint becomes more transparent.

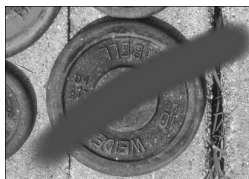
Selecting a Colour Model

You can apply the colour using either the YUV or HLS colour. Select either the YUV Shade/Wash or the HLS Shade/Wash option in the Rendering setup menu. Click Setup and Rendering to display the menu.



To use Wash or Shade:

1. Click Custom, and select Wash or Shade from the Custom Media box.
2. Click Canvas.
3. Set the current colour.
4. Set the brush opacity.
5. Paint on the image.



A paint stroke using the Paint medium, current colour red



A paint stroke using the Wash medium, current colour red



A paint stroke using the Shade medium, current colour red

[illegible]

Paint: Using Wipe, Fill, and AutoPaint

Fill it up

Wipe colours, filters or Special Effects media over your entire image. Fill areas of your image with colour or a reference image. Use AutoPaint to apply paint strokes to each frame of your clip.

Summary

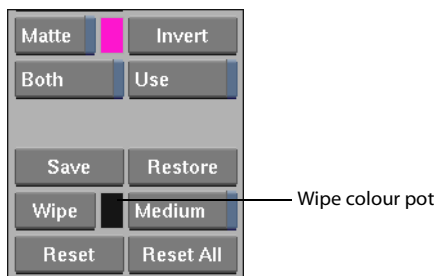
In this chapter, you learn about:

“The Wipe Command” on page 1095

- “The Fill Command” on page 1097
- “AutoPaint” on page 1101

The Wipe Command

The Wipe command applies colours, filters, and Special Effects media to the entire result image.



NOTE: You can also wipe an image with the AutoPaint Wipe mode. For more information, see “Wiping the Canvas in AutoPaint” on page 1101.

Wiping Using a Colour

Use the Paint medium to wipe the image with a selected colour. The colour used is set in the Wipe colour pot.

To wipe the image with a colour:

1. Set the current colour to the colour you want to use for the wipe.
For more information on setting the current colour, see Chapter 48, “Paint: Overview.”
2. Click the Wipe colour pot.
The current colour is transferred to the wipe colour.
3. Click Medium.
4. Set the brush opacity.
A value of 100% wipes the image with a completely opaque colour.
5. Click Wipe.

Wiping Using Special Effects Media and Filters

You can use a filter, or the Reveal, Clone, Wash, and Shade Special Effects media. For more information, see Chapter 51, “Paint: Using Filters and Special Effects Media.”

To use the Wipe command with Special Effects media and filters:

1. Set the wipe colour.
2. Select the Special Effects medium to be applied.
3. Set the brush opacity.
A value of 100% wipes the image with the full effect of the Special Effects Media or filter.
4. Click Medium.

NOTE: If you are using a filter for the wipe, you must enable the Filter button.
5. Click Wipe.

Using the Wipe Command in Graphics

You can also use the Wipe command in the Graphics menu.

The Wipe command wipes over any objects that have been tacked down on the image. Any objects not tacked down are not a part of the image and are not wiped over.

The Fill Command

The Fill command is used to fill areas of an image that have similar colour values or areas delimited by a colour. These areas can be filled with either a solid colour or a reference image. You can choose how far the filled area will extend by specifying how similar the pixels must be in order to be filled. This enables you to fill just the dark areas of an image, or include slightly lighter areas.

Click Fill in the Paint menu to view the Fill controls. If the Fill button is hidden by the colour palette, swipe the bottom of the screen.

Defining the Range for the Fill

When you use the Fill command, you specify a range for the colour comparison. The Fill command compares the colour values of adjacent pixels to determine if the values are within the specified colour range. Adjacent pixels that are within the range are filled.

The range is determined using two values: the colour value of the pixel that you select to begin the fill, which is called the selection point, and the value that you set in the Range field.

Selecting Colour Channels

Use either the RGB or YUV model. Within each colour space, you can work with any combination of colour channels. For example, if you select the R, G, and B channels in the RGB colour model, the Fill command considers the red, green, and blue values. Adjacent pixels with red, green, and blue values within the specified range are filled.

Adjusting the Softness

The softness value for the fill determines the amount of colour diffusion that is applied at the edges of the fill. This value can be adjusted to modify the transition between the filled and unfilled areas. A softness value of 100% produces the greatest amount of diffusion. A softness value of 0% produces a filled area with well-defined edges.

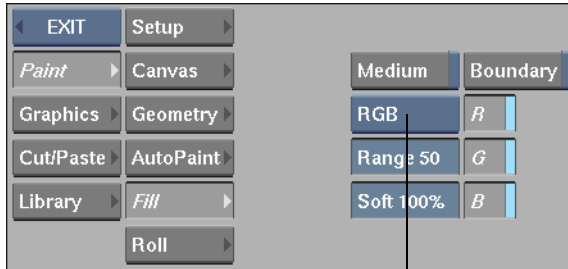
Filling a Region with a Colour

Use the Fill controls to set the colour model, range, and softness of the fill.

To fill a region of the image with a solid colour:

1. Click Fill.

The Fill controls appear.



Colour Model box

NOTE: To fill a matte, click From Matte. When this button is enabled, **flame** uses the pixels in the matte for the colour comparison. If this button does not appear at first, click Matte on the right side of the menu panel.

2. Select a colour model from the Colour Model box.

Select: To:

RGB Display the R, G, and B channel buttons. Enable the button for each channel that you want to use.

YUV Display the Y, U, and V channel buttons. Enable the button for each channel that you want to use.

3. Set the colour range in the Range field.

You can also set a tolerance by enabling the Boundary button, and selecting a distinct boundary colour from the image.

4. Set the softness in the Soft field.
5. Set the current colour.
6. Select a point in the area that you want to fill. This is the selection point.

The pixels that fall within the specified range are filled with the current colour.



Original image



Fill with range 75 and softness 100



Fill with range 75 and softness 50



Fill with range 50 and softness 100

Filling a Region with an Image

Enable Use Medium to fill a region of an image with a reference image instead of a colour. The front, back, or result image can be used as the reference image. The Wash, Shade, Clone, or Reveal media can also be used to perform the fill.

To fill a region with a reference image:

1. Click Fill.
2. Select the colour model and channels you want to work with.
3. Set the range and softness.
4. Select the Special Effects medium that you want to use (Clone, Reveal, Wash, or Shade).
5. Set the Reference box to Front, Back, Result, or Saved, and set the brush opacity.
A value of 100% will fill the area completely with the reference image.
6. Enable Use Medium.
7. Select a point in the area that you want to fill.

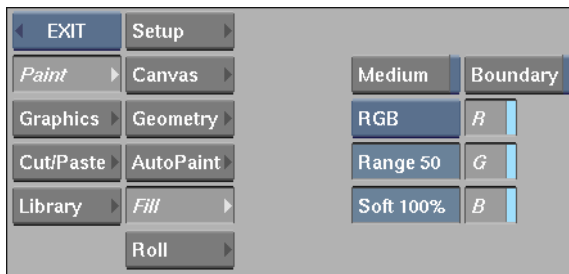
Filling a Boundary

Use the Boundary option to fill an area of the canvas that is delimited by another colour.

To use boundary fill:

1. Click Fill.
2. Select the colour model and channels you want to work with.
3. Set the range for the fill in the Range field.
4. Enable Boundary.
5. Click Pick.

The cursor changes to a colour picker when dragged over the image.



6. Drag the colour picker over the canvas without clicking.
The various colour channel values in areas of the image are displayed as you move the colour picker around the canvas.
7. Click on a point to select the colour for the boundary. You can also select a colour by entering the RGB values directly in the colour channel fields.
The cursor changes to a paint bucket.
8. Click the area inside the boundary to fill that region.

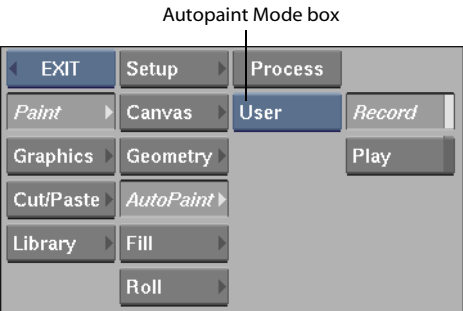
NOTE: To fill a boundary with a reference image, set the Reference box to Front, Back, or Saved, and enable Use Medium.

AutoPaint

Use the Autopaint feature to apply paint strokes to each frame in the result clip or to the current frame.

Select:	To:
Wipe	Wipe the entire canvas with the current colour or medium.
Random	Apply a number of random strokes to the image.
User	Record and play back a series of manual paint strokes.

Click Autopaint in the Paint menu to display the AutoPaint controls. If the second column is hidden by the colour palette, swipe the bottom of the screen.



Wiping the Canvas in AutoPaint

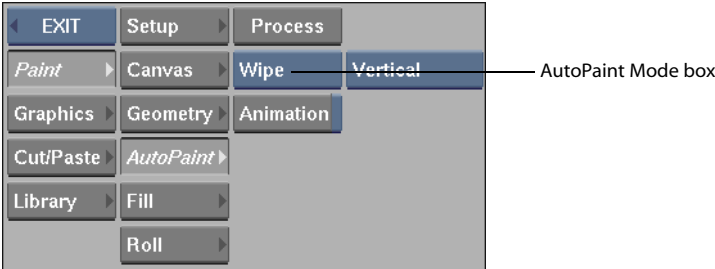
In AutoPaint, you select Wipe mode to automatically apply paint strokes from left-to-right, top-to-bottom, or diagonally across the entire canvas.

You can use a filter, or any of the Special Effects media except Warp, Drag, and Smear. The number of strokes applied is determined by the size of the brush. Use a small brush to apply several strokes, or a larger brush to apply fewer strokes.

To wipe the canvas in AutoPaint:

1. Click AutoPaint, and select Wipe from the AutoPaint Mode box.

The Wipe menu appears.



2. Set the current colour.
3. Select your Special Effects medium or filter, and set the brush attributes.
4. Select the Wipe mode in the Wipe box. If this box is not visible, make sure that the attribute mode for the Direction attribute is set to Direction.

Select Diagonal to apply diagonal strokes, Vertical to apply strokes from top-to-bottom, and Horizontal to apply strokes from left-to-right.

5. Enable Animation to display the Channel Editor.

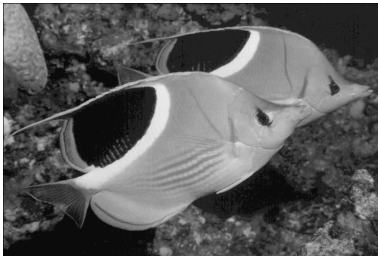
If the Channel Editor is not visible, swipe down through the menu.

The following parameters can be animated:

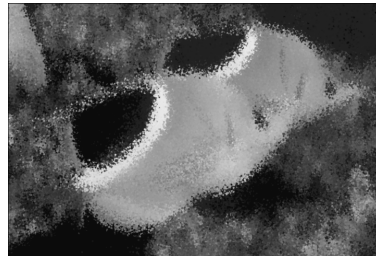
- All of the brush attributes
- Current colour

For more information on using the Channel Editor, see Chapter 9, “Animation.”

6. Enable Wipe if you are using a filter or colour.
7. Click the image to apply the paint strokes only to the current frame. Click Process to apply the paint strokes to each frame in the front clip.



Original image



After using the Wipe command with Jitter attribute at 25% and Colour attribute mode set to Front

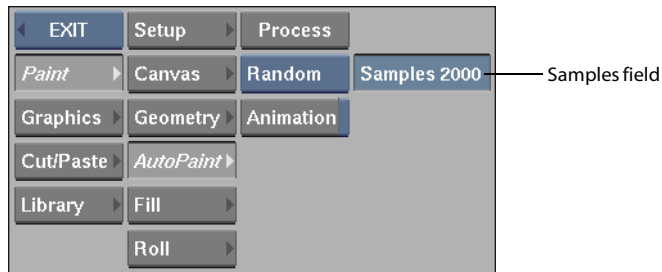
Applying Random Strokes

Use the Random mode to automatically generate a number of paint strokes with random orientation on the image. Use the Samples field to specify the number of random strokes that you want to generate. You can use any Special Effects media in Random mode except Warp and Drag.

To apply random paint strokes to the image:

1. Click AutoPaint and select Random from the AutoPaint Mode box.

The Samples field appears.



2. Enter the number of strokes that you want to apply in the Samples field.
3. Set the current colour.
4. Select your Special Effects medium, and set the brush attributes.
5. Enable the Animation button to display the Channel Editor.

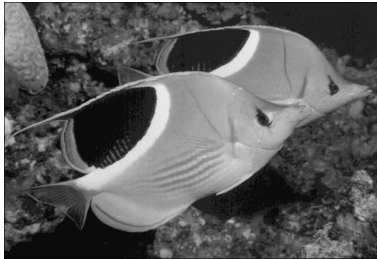
If the Channel Editor is not visible, swipe downward through the menu.

The following parameters can be animated:

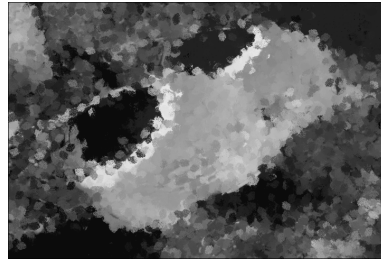
- Number of strokes
- All of the brush attributes
- Colour

For more information on using the Channel Editor, see Chapter 9, “Animation.”

- Click on the image to apply the paint strokes to the current frame only. Click Process to apply the paint strokes to each frame in the front clip.



Original image



After using Random command with Jitter attribute at 30% and Colour attribute mode set to Front

Recording Brush Strokes

In AutoPaint, use the User mode to record and play back a series of paint strokes. Only the positions of the brush strokes are recorded. This means that you cannot change the brush type, current colour, or brush attributes while recording the strokes. You can, however, change or animate these parameters after you finish recording and before you play back the strokes.

To record a series of brush strokes:

HINT: You can also play paint strokes created by converting objects (write-ons).

- Click AutoPaint and select User from the AutoPaint Mode box.
- Set the brush characteristics so that you will be able to see the recorded brush strokes on your image.

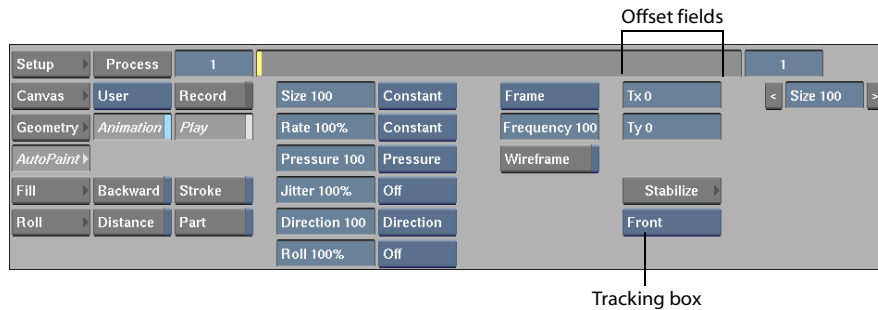
NOTE: The strokes you paint are recorded as a series of points or stamps of the brush. The number of points is divided by the number of frames to determine the number of strokes. The quality of the rendered strokes depends on the number of recorded points. To record a larger number of points, select a small brush size.

- Enable Record.
- Begin painting on the image.
The position of each brush stroke is recorded, and the stroke count appears in the message bar.
- To stop recording, click below the timeline.
All paint strokes that you applied to the image while recording are removed from the image and the number of strokes is recorded.

NOTE: AutoPaint stores only one set of recorded paint strokes at a time. If you record another set of paint strokes, you lose your previously recorded strokes.

Tracking with AutoPaint

You can apply tracking data to the painted strokes.



To apply tracking data to your recorded strokes:

1. Enable Animation.
2. Select Front, Back, or Result from the Tracking box and click S to track a point and apply the offset information to the recorded strokes.

NOTE: You can only track after you record your paint strokes.

3. Use the Tx and Ty field to offset the painted strokes.

For more information on tracking, see Chapter 38, “Tracking and Stabilizing.”

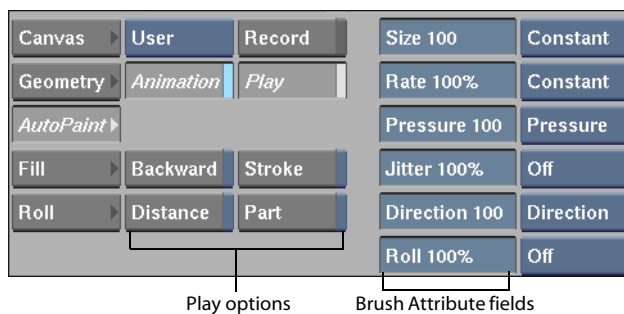
Playing Recorded Brush Strokes

After recording brush strokes, you can apply them to the front clip using the Play option, which is automatically activated when you finish recording your paint strokes.

To play back recorded strokes:

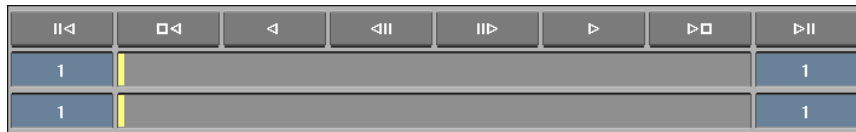
1. Enable Play.

The Play options appear.



2. Define the brush characteristics using the Brush Attribute fields.

- Define the duration of the stroke sequence using the second timeline.



The number of strokes is divided by the length of the stroke as defined by the second timeline.

The bar in the second timeline is identical to a track in the Channel Editor. For more information on using tracks, see Chapter 9, “Animation.”

- Select the Play options.

Enable: **To:**

Part Play only a part of the recorded strokes. AutoPaint applies the strokes to a frame, erases those strokes, and moves to the next frame. This has the effect of creating streaks on your rendered clip.

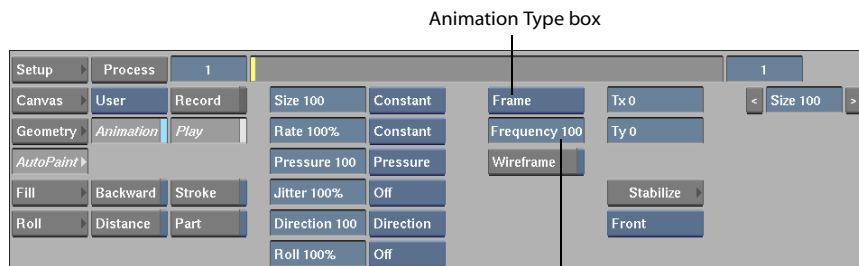
Back Play the paint strokes backward.

Dist Play paint strokes based on distance. AutoPaint divides the length of the paint strokes by the number of frames to determine what to render in each frame.

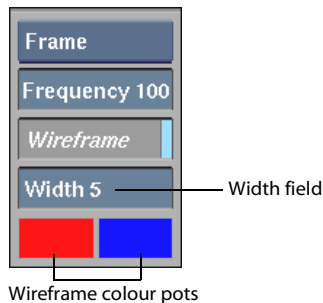
Stroke Play the paint strokes simultaneously.

- Click Animation.

The Animation menu appears.



6. Enable Wireframe to preview the path(s) of the painted strokes.



The wireframe preview shows the complete path(s) of the painted strokes. It also shows what will be painted in each frame as set using the second timeline.

You can set the width of the wireframe using the Width field. You can also change the colour of the path and progression wireframes using the colour pots beneath the Width field.

7. Select Frame or Path animation for your AutoPaint strokes from the Animation Type box. Frame reads the Channel Editor values once every frame and applies those values to all the AutoPaint strokes in that frame. For Path animation, you define how often AutoPaint reads the Channel Editor values when rendering the points or stamps that make up a given stroke. At a frequency of 100%, AutoPaint reads the Channel Editor values most frequently.

HINT: Use the Frequency field to lower the processing time for your AutoPaint sequence. Use a low frequency to render a quick preview of the result.

For example, if you set your AutoPaint sequence of 100 stamps to run over 10 frames using Path animation, AutoPaint reads the Channel Editor values 10 times every frame if you set the frequency field to 100%. If you set the frequency field to 50%, AutoPaint will read the Channel Editor values 5 times every frame.

8. Swipe down through the menu to display the Channel Editor.

You can animate the following parameters:

- Sampling amount
- Tx and Ty values
- Brush size, rate, pressure, jitter, direction, roll, opacity, and colour
- Tracker Translation

For more information on using the Channel Editor, see Chapter 9, “Animation.”

NOTE: The Channel Editor only appears if Animation is enabled. Also, animation data in the Channel Editor is not erased when you record new strokes.

9. To apply the paint strokes, click Process.

NOTE: If you are presently zoomed in on the image, and are in the Raster Zoom mode, only the visible portion of the image will be processed. This will process much faster than in Tiled mode, but will not apply your modifications to the entire frame. For more information on Zoom modes, see “Painting on Full-Resolution Film Images” on page 1066.

53

Paint: Using Mattes

Cover up

Use mattes to protect an area of your image. View, create and modify mattes.

Load a matte from the Matte library or create your own.

Summary

In this chapter, you learn about:

- “Viewing the Matte” on page 1110
- “Creating or Modifying a Matte” on page 1111
- “Using the Auto Matte Command” on page 1113

For more information on loading mattes into Paint, see Chapter 48, “Paint: Overview,” and Chapter 56, “Paint: Creating, Loading, and Saving Setups.”

About the Matte

Mattes protect specific areas of the image when you apply paint, filters, or Special Effects media to the canvas. You can also use mattes to limit the area of a cutout.

To use the matte:

1. Click Use to enable the matte.



2. Paint on the image.
3. Click Use again to disable the matte.

Using Mattes with Cutouts

If you enable your matte when creating a cutout, the cutout is limited by the matte. Only objects outside the matte will show up in the cutout.

For more information, see Chapter 55, “Paint: Cut and Paste.”

Viewing the Matte

You can view the matte while you are using it.

To view the matte:

1. Click Show.



Reference box

2. Select the Matte option in the Reference box.

The matte appears. The default matte colour is pink.

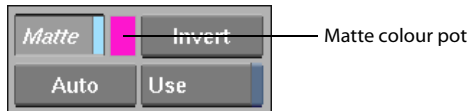
Changing the Matte Colour

If the colour used to display the matte blends with the image, you can change its colour.

NOTE: The matte is always created using shades of grey.

To change the matte display colour:

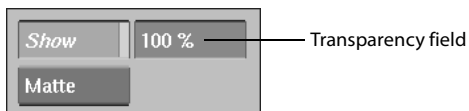
1. Set the current colour.
2. Click the Matte colour pot.



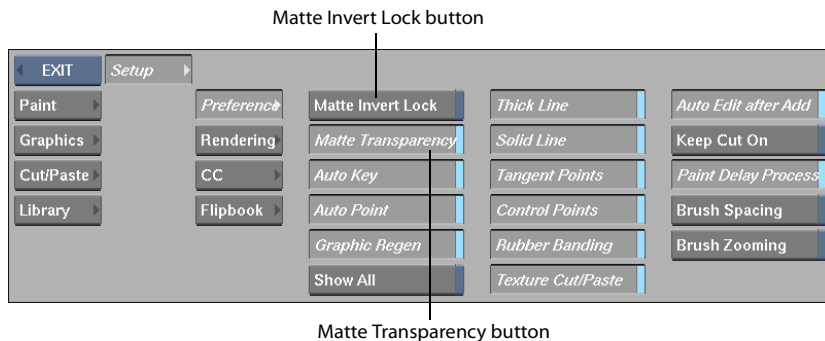
The matte colour display changes to the current colour.

Changing the Matte Display Transparency

When viewing the matte, use the Transparency field to vary the matte display transparency from 0% (completely transparent) to 100% (fully opaque). Press **T** or click Matte Transparency in the Preferences menu to toggle between the last set value and 100%.



NOTE: The transparency value only affects the display of the matte; it does not affect how the matte is used.



Inverting the Matte Clip

Click Inv next to the Matte button to invert the matte in the current frame. Enable Matte Invert Lock in the Preferences menu to invert the matte in every frame of the clip.

Creating or Modifying a Matte

You can create or modify a matte by painting directly on the matte using any brush, Special Effects media, or graphic tool. For information on using graphic and cutout tools to create mattes, see Chapter 55, “Paint: Cut and Paste.”

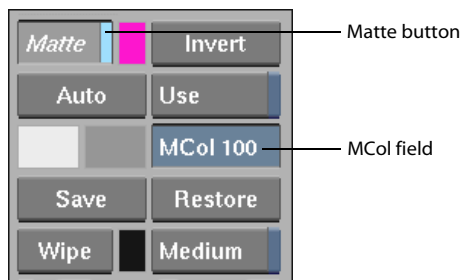
To create or modify a matte:

1. Load the matte into paint.

NOTE: You must load a matte clip in order to process a matte in Paint. If you want to create a matte from scratch, load a black source as the matte clip.

2. Click Matte.

The matte appears.



3. Set the paint transparency in the MCol field.

The value in the MCol field determines the transparency of the paint applied to the matte. Use a value of 100 to apply fully opaque paint or objects to the matte. Reduce the MCol value to increase the transparency of the paint.

4. Paint or place graphics or cutouts on the matte.

The paint and graphics are applied to the matte. If you use graphic tools, use Tack to tack the graphic to the matte.

5. You can save the matte in the matte library. See Chapter 56, “Paint: Creating, Loading, and Saving Setups.”
6. To process the matte clip, click Exit.

The new matte clip is saved to the desktop with the front clip.

Painting on the Matte and Image

Enable Both to paint the image and the matte simultaneously. This feature can be used when applying paint strokes or graphics to the image.

Resetting the Matte

If you do not load a matte, Paint displays the last matte that was loaded.

To erase the matte:

1. Click Matte.
2. Set the MCol to 0%.
3. Click Wipe.

The matte is erased.

Creating a Gradient Matte

You can use graphics that contain gradients to create a matte that blends from one level of the matte colour to another level. A gradient matte is useful for blending paint strokes or creating soft edges.

To create a gradient matte, draw a graphic with a gradient on the matte, and then use the Channel Editor to change the “a” channel of the start or end colour of the gradient.

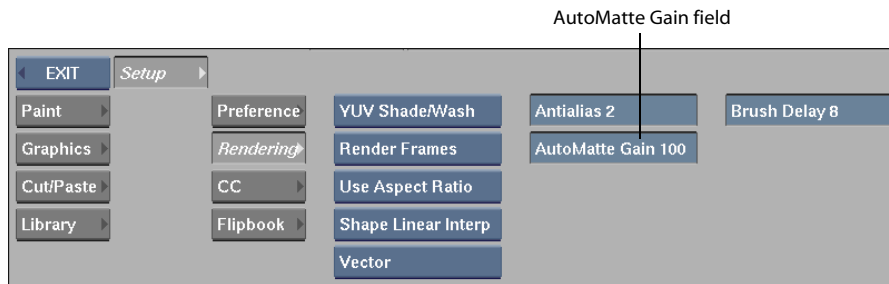
For more information, see Chapter 54, “Paint: Using Graphics.”

Using the Auto Matte Command

Use the Auto Matte command to create a high-contrast matte. This command works like the Auto Matte command in the Processing menu.

The minimum and maximum luminance values for the matte are set using the two colour pots in the Auto Matte menu. Any pixel with a luminance value below the minimum is set to black, and any pixel with a luminance value above the maximum is set to white.

Unwanted grey areas can be removed from the matte by increasing the percentage in the Auto Matte Gain field in the Rendering menu.

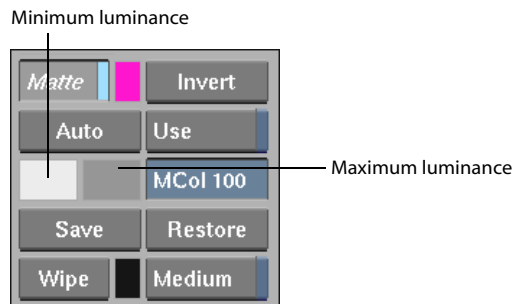


Pixel values between the minimum and maximum values are multiplied by the Gain value. The resulting values are clipped at the specified maximum luminance value. The Gain is expressed as a percentage value. The default value of 100% has no effect on the image since the luminance values are multiplied by 1.

To use the Auto Matte command:

1. Click Matte.

The Auto Matte menu appears.



2. Set the minimum luminance value for the matte in the colour pot on the left. To set the value, click on the field and use the colour picker to select the lightest colour from the image.

3. Set the maximum luminance value for the matte in the colour pot on the right.
4. Set the Auto Matte Gain for the matte. The Auto Matte Gain field is in the Rendering Setup menu.
5. Click Auto to generate the matte.
6. Click Use to enable the generated matte.

Paint: Using Graphics

Shape up

Use the Graphic tools to draw geometric objects on your image. Add objects such as lines, polygons, fills and text. Resize and reshape objects, and add colour and gradients to your graphics.

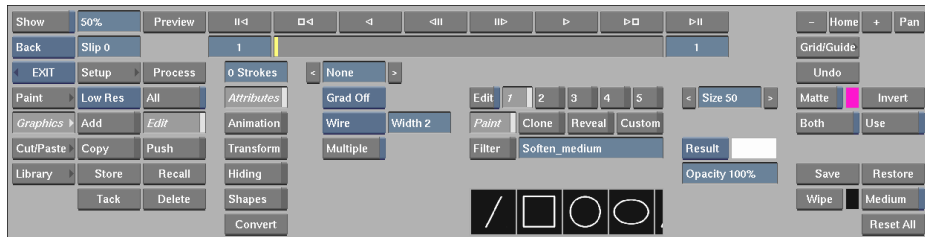
Summary

In this chapter, you learn about:

- “Accessing the Graphics Menu” on page 1116
- “Creating Graphics in Paint” on page 1116
- “Using the Object Tools” on page 1117
- “Selecting Objects” on page 1126
- “Editing Objects” on page 1127
- “Setting the Object Resolution and Display” on page 1128
- “Setting the Object Attribute” on page 1131
- “Changing the Object’s Appearance” on page 1132
- “Changing the Size or Position of an Object” on page 1134
- “Changing the Shape of an Object” on page 1136
- “Converting Graphics to Autopaint Strokes” on page 1138
- “Animating Graphics” on page 1139
- “Saving and Restoring Objects” on page 1143
- “Tacking Down Objects” on page 1144

Accessing the Graphics Menu

To access the Graphics menu, click the Graphics button in the Paint menu. Many options may be hidden if the colour palette is open. Swipe the cursor across the bars at the bottom of the screen to hide the colour palette.



The Graphics Brush

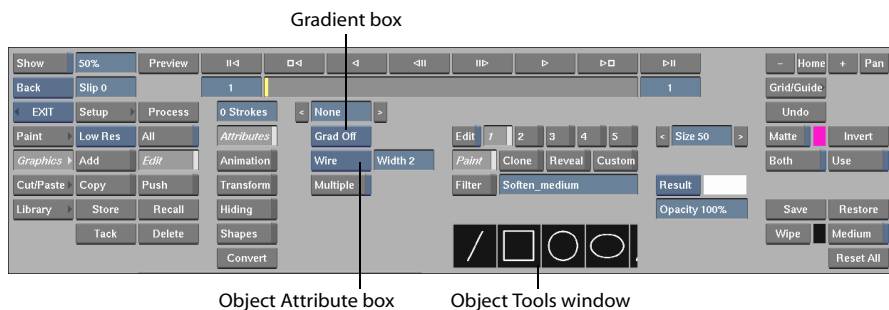
When you move the cursor onto an image, it appears as a small green cross with crosshairs that extend the width and height of the canvas. The crosshairs help align objects on the image.

Creating Graphics in Paint

The following steps describe how to create graphics in Paint. Graphics or objects can be used to create cutouts or mattes, or to apply geometrical shapes, text, and fills to the result clip.

To draw an object:

1. Click the Graphics button.
2. Click the Add button.



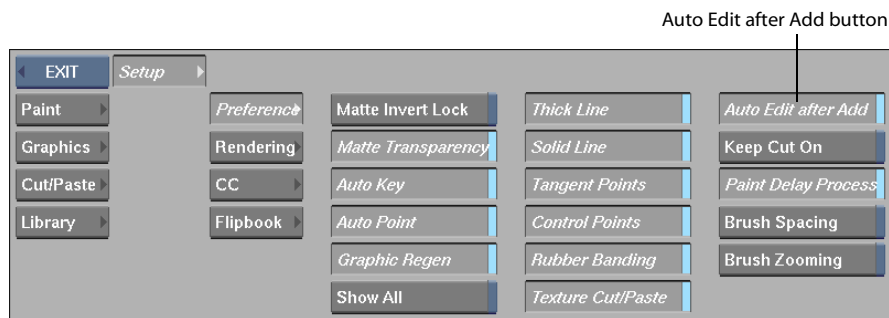
3. Select the type of object to add by selecting an icon in the Object Tools window.
4. Draw the object on the image. See “Using the Object Tools” on page 1117.
5. Set the resolution to be used when adding or editing objects. See “Setting the Object Resolution and Display” on page 1128. Objects appear at full resolution when they are drawn.

6. Set the object's attributes and gradient. See “Setting the Object Attribute” on page 1131, and “Changing the Object Gradient” on page 1132.
7. Set the object's colour in the Current Colour field. See Chapter 48, “Paint: Overview.”
8. Use the Transformation box or controls to move, rotate, or scale the object. See “Changing the Shape of an Object” on page 1136.
9. Use the Animation controls to animate the object. See “Animating Graphics” on page 1139.
10. Click the Tack button to permanently place the object on the result clip.

Once the object has been tacked down, it cannot be moved, rotated, deleted, resized, or copied. See “Tacking Down Objects” on page 1144.

Auto Edit after Add

By default, Paint automatically switches from Add mode to Edit mode after you add an object to the image. You can turn off this default setting by disabling the Auto Edit After Add button in the Preferences Setup menu. You can now add objects one after another without interruption.



Using the Object Tools

A number of predefined graphic tools are available in Paint. The cursor looks the same for all graphic types. Each tool has its own icon in the Object Tools window.

To select an object tool:

1. Scroll through the Object Tools window.



Click the Object Tools window and drag left or right, or click on either the left or right border and hold down the cursor to scroll. The borders are pressure sensitive.

Use the left mouse button to scroll slowly, the middle button to scroll faster, and the right button to scroll the fastest.

2. Click on the object tool icon you want to use.

The selected object tool is highlighted by a yellow outline.

NOTE: Only one object tool can be active at a time.

Drawing a Line

Use the Line object tool to draw a single straight line or a multi-line object (a series of lines joined end-to-end).

To draw a line or multi-line:

1. Click the Add button in the Graphics menu.
2. Select the Line tool in the Object Tools window.
The Multiple button appears.
3. Click the Multiple button to draw multiple lines.
4. To draw single lines, move to the canvas and click, drag, and release.
Paint draws the line as you drag.
5. To draw multiple lines, move to the canvas and click to place the start point of the first line.
Click again to draw the end point. Continue clicking to draw more lines.
6. To end a multiple line object, click outside the canvas or click the Multiple button.

Drawing a Rectangle

Use the Rectangle object tool to draw a rectangle or square in one of two ways:

- Corner-to-corner, with the two control points located at diagonally opposite corners of the rectangle
- Centre-to-corner, with the first control point at the centre of the rectangle and the second at one corner

To draw a rectangle from corner-to-corner:

1. Click the Add button in the Graphics menu.
2. Select the Rectangle tool in the Object Tools window.
3. Press the cursor to anchor one corner of the rectangle. Do not release the cursor.
4. Drag the cursor diagonally. You can adjust the width and height of the rectangle as long as you hold down the cursor.
5. Release the cursor to anchor the second control point.
The rectangle is drawn on the image.

To draw a rectangle from centre-to-corner:

1. Click the Add button in the Graphics menu.
2. Select the Rectangle tool in the Object Tools window.
3. Press **ALT**.
4. Position the cursor where you want to place the centre of the rectangle and press down.
5. Drag diagonally, and release the cursor when the rectangle is the correct size.

Drawing a Square

Repeat the procedure for drawing a rectangle from corner-to-corner, and press **P** while you draw.

Drawing a Triangle

Use the Triangle object tool to draw a triangle or equilateral triangle. The control points are located at the three vertices of the triangle.

To draw a triangle:

1. Click the Add button in the Graphics menu.
2. Select the Triangle tool in the Object Tools window.
3. Position the cursor and press to anchor the first control point on the image. Do not release the cursor.
4. Drag the cursor horizontally to draw the base of the triangle. You can continue to adjust the length of the base as long as you press down on the cursor.
5. Release the cursor to anchor the second control point.
6. Move the cursor to where you want to place the third vertex. You can continue to adjust the position of the third vertex until you press down on the cursor. Press to anchor the third control point.

The triangle is drawn on the image.

Drawing an Equilateral Triangle

An equilateral triangle has three sides of equal length. To draw an equilateral triangle, repeat the procedure for drawing a triangle, and press **P** while drawing the base of the triangle. This also establishes the height of the triangle. Release the cursor to anchor the second and third control points on the image.

Drawing an Ellipse

Use the Ellipse object tool to draw an ellipse defined by three control points. The first control point determines the centre of the ellipse. The location of second control point determines horizontal radius of ellipse. The location of the third control point determines the vertical radius.

To draw an ellipse:

1. Click the Add button in the Graphics menu.
2. Select the Ellipse tool in the Object Tools window.
3. Position the cursor at the centre of the ellipse and press to anchor the point on the image. Do not release the cursor.
4. Drag the cursor horizontally to establish the width of the ellipse. You can continue to adjust the width as long as you press down on the cursor.
5. Release the cursor to anchor the second control point.
6. Move the cursor along the vertical axis to position the third control point. You can continue to adjust the height of the ellipse until you press down on the cursor. Press to anchor the third control point.

The ellipse is drawn on the image.

To draw a circle with the Ellipse tool:

1. Click the Add button and select the Ellipse tool.
2. Press **P**.
3. Position the cursor at the centre of the circle and press to anchor the point on the image. Do not release the cursor.
4. Drag the cursor horizontally to establish the radius of the circle. Notice that there are two control points at the edge of the circle, as there are for an ellipse.
5. Release the cursor and the **P** key to anchor the second and third control points.

The circle is drawn on the image.

Drawing a Circle

Use the Circle object tool to draw a circle in one of two ways:

- Centre-to-edge, with one control point at the centre of the circle and a second on the circumference
- Edge-to-edge, with two control points at opposite points on the circumference

To draw a circle from edge-to-edge:

1. Click the Add button in the Graphics menu.
2. Select the Circle tool in the Object Tools window.

3. Position the cursor where you want the centre of the circle and press to anchor that point on the image. Do not release the cursor.
4. Drag the cursor to establish the radius of the circle. You can continue to adjust the size of the circle as long as you press down on the cursor.
5. Release the cursor to anchor the second control point. The circle is drawn on the image.

To draw a circle from centre-to-edge:

1. Click the Add button and select the Circle tool.
2. Press and hold **ALT**.
3. Drag the cursor to establish the radius of the circle.
When the circle is the required size, release the cursor.

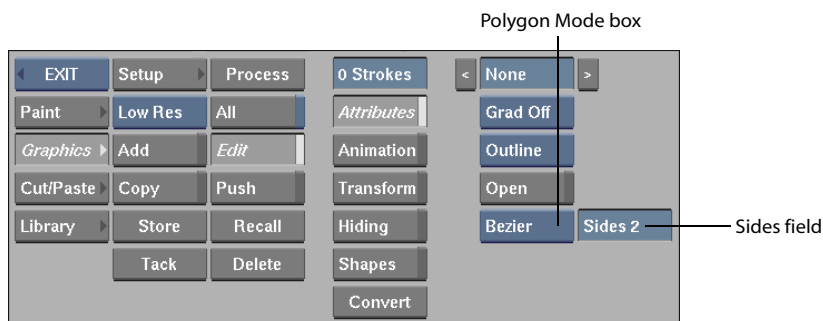
Drawing a Polygon

Use the Polygon object tool to create a series of connected lines that form a closed or open object. You can specify how the vertices of the polygon should be connected by selecting one of the options from the Polygon Mode box.

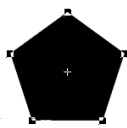
To draw a polygon:

1. Click the Add button in the Graphics menu.
2. Select the Polygon tool in the Object Tools window.

The Polygon menu appears.



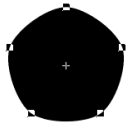
3. Select an option from the Polygon Mode box to set the type of curve used to join the vertices of the polygon.

**Select:**

Linear

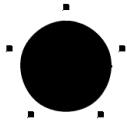
To:

Use straight lines to join the vertices of the polygon.



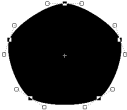
Cardinal

Use a smooth curve that passes through the vertices of the polygon.



B-spline

Use a very smooth curve that passes on the inner side of the vertices of the polygon.



Bezier

Use Bezier curves. Each vertex of the polygon has a tangent with two tangent handles. In Edit mode, you can move the tangent handles to adjust the slope of the polygon. For more information, see "Editing a Bezier Curve" on page 1137.

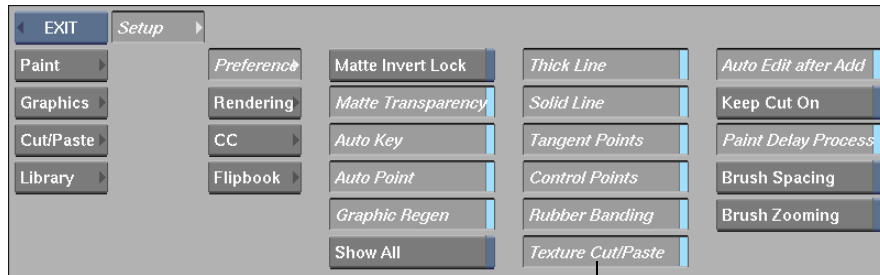
The minimum number of vertices for a polygon depends on the type of interpolation used. A polygon should have at least three vertices when using Linear interpolation, or four when using Cardinal or B-spline interpolation. The maximum number of vertices is 255. A polygon can be concave.

4. Click the Open button if you want to create an open-ended polygon.
5. Position the cursor at one vertex of the polygon, and click to anchor that point on the image.
6. Release the cursor and move it to where you want the next vertex of the polygon. You can continue to adjust the positioning of the next vertex until you press down on the cursor. Press to anchor the point on the image.
7. Repeat steps 4 and 5 for each vertex.
8. To close the polygon, press anywhere outside of the canvas.
The polygon is drawn on the image.

Rubber Banding

Rubber banding is the default setting for drawing polygons and lines. This means that a vertex is added to the polygon only when you press and release the cursor.

To draw a polygon with multiple vertices, disable the Rubber Banding button in the Preferences Setup menu. When you drag the cursor while drawing a polygon or line, a number of control points are drawn on the image.



Rubber Banding button

Drawing Regular Polygons

Draw regular polygons using the Sides field and the **P** key. The polygon can have three or more sides. This enables you to draw a triangle or a square based on the centre point of the object.

To draw a regular polygon:

1. Click the Add button in the Graphics menu.
2. Select the Polygon tool in the Object Tools window.
3. Enter the number of sides for the polygon in the Sides field.
4. Press **P**.
5. Position the cursor over the image and press down.
This is the centre of the polygon.
6. Drag the cursor away from the centre point. The polygon is drawn on the image. You can continue to adjust the size of the polygon as long as you press down on the cursor.
7. When the polygon is the correct size, release the cursor.
The polygon is drawn on the image.

Drawing a Fill Object

Use the Fill object tool to fill areas of an image with colour. While this is similar to using the Fill command in the Paint menu, you can only use this tool with colour, Shade and Wash, and not to fill areas with a reference image. However, you can edit fill objects after you draw them.

NOTE: A fill object cannot have a gradient.

To draw a fill object:

1. Click the Add button in the Graphics menu.
2. Select the Fill object in the Object Tools window.

The Fill object controls appear.



3. Select a colour model from the Colour Model box (RGB or YUV).
4. Select the colour channels you want to use (R, G, and B, or Y, U, and V).

The choice of colour channels determines how Paint evaluates the area you want to fill. For example, if you select R, G, and B, Paint fills areas whose adjacent pixels have red, green, and blue values in the specified range. If you select only R, Paint fills only areas whose adjacent pixels have red values in the specified range.

5. Enter the Range and Softness values for the fill.

The range determines how far the fill spreads from the point you click on the image; if you increase the range, you increase the fill area. The softness determines how much diffusion occurs at the edges of the fill; if you increase the softness, you increase the diffusion at the edges of the fill.

6. Move the cursor to the canvas and click inside the area to fill.

Paint fills the area with the current colour starting at the point you click.

NOTE: The colour of the image and the channel, range, and softness settings determine the extent of the fill. As you move the fill object, it changes size and shape in response to the different colours in the image.

Drawing a Text Object

Use the Text object tool to add text to the image. You can change the size, kerning, and inclination of the text string.

To add a text object:

1. Click the Add button in the Graphics menu.
2. Select the Text tool in the Object Tools window.

The Text object controls appear.



3. By default, Paint uses the font declared on the TextDefaultFont line in the configuration file. If you want to change the font, click the Font field and select a new font from the font library.
4. Click the Text field, type the text string and click Enter.
5. Click on the image to place the text object.

In Edit mode, you can adjust the size, kerning, and italics of the text.

Use:	To:
Size	Adjust the size (in pixels) of the text.
Italic	Incline the text. Enter a positive value to slant the text to the right, and a negative value to slant it to the left.
Kern	Adjust the space (in pixels) between all the letters in the text.

Selecting Objects

You must select an object before you can edit it. In Edit mode, you select objects in three different ways:

- Click on an object to select it.
- Use the Selected field to select individual objects.
- Use the All command to select all objects at once.

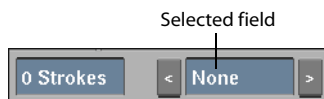
Using the Selected Field

As you add objects to the image, they are numbered sequentially, starting at 1. You can select an object by specifying the number of the object in the Selected field.

As you change the number in the Selected field, the corresponding object is highlighted by a selection box.

To select a single object using the Selected field:

1. Set the number in the Selected field to the number of the object you want to edit.
You can also click the < and > buttons beside the Selected field to move through the sequence.
2. When the selection box outlines the object, release the cursor. You can now edit it.



Selecting More than One Object

To select an additional object, hold down **SHIFT**, and click on another object. Repeat this step for each object you want to select.

Deselecting Objects

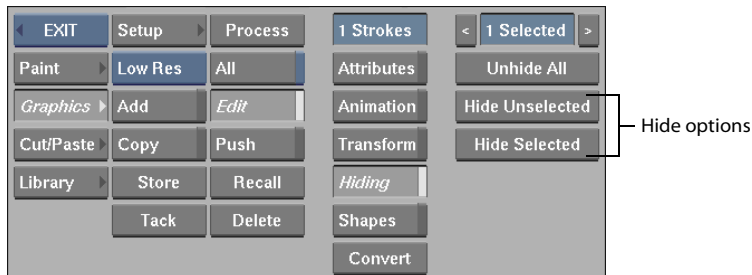
To deselect an object or objects, select a different object or click the Add button.

Selecting or Deselecting All Objects

To select all the objects, click the All button. All is shown in the Selected field. Click it again to deselect them.

Editing Objects

Use the object controls to copy, delete, hide, or layer objects.



Copying Objects

Use the Copy command to create multiple copies of an object. Once a copy is created, it can be selected and edited like any other object.

To copy an object:

1. Click the Copy button in the Graphics menu.
2. Click on the object you want to copy. Do not release the cursor.
A copy of the object is superimposed on the original image. You cannot see the copy until you drag it to a new location.
3. Drag the copy to a new location.
4. When the copy is correctly positioned on the image, release the cursor.

Deleting Objects

Use the Delete command to remove one or more objects from the image.

To delete one or more objects:

1. Select the objects that you want to delete. Click on an individual object or use the Selected box to choose an object. To select multiple objects, click on an object, press **SHIFT**, and click on any other objects you want to delete.
2. Click Delete.
All selected objects are removed from the image.

To delete all objects from the image:

1. Enable All.
2. Click the Delete button.
All objects are deleted from the image.

Hiding Objects

Use the Hide/Unhide commands to hide or unhide objects before they are tacked onto the image.

To hide or unhide one or more objects:

1. Select the objects you want to hide. Click on an individual object or use the Selected box to choose an object. To select multiple objects, click on an object, press **SHIFT**, and click on any other objects you want to hide.
2. Click Hiding.
The Hide options appear.
3. Click Hide Sel to hide all selected objects.
4. Click Hide Unsel to hide all objects that are not selected.
5. Click Unhide All to show all hidden objects.

Changing the Order of Overlapping Objects

Objects can be drawn so that they overlap other objects to produce a stack of objects. An object can be sent to the bottom of the stack using the Push command.

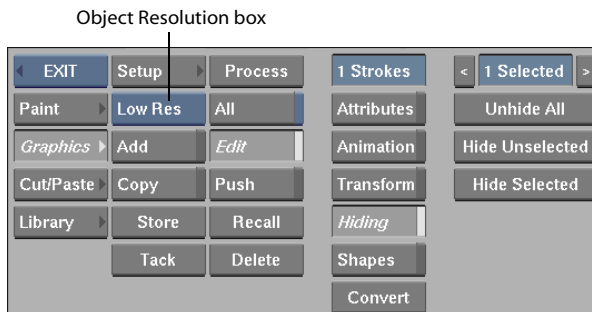
To send an object to the bottom of the stack:

1. Select the object.
2. Click Push in Graphics menu and click the object until the object is at the bottom of the stack.

NOTE: To bring the object back to the front, click the object until it moves to the front.

Setting the Object Resolution and Display

Use the Object Resolution box to set the resolution while drawing or editing objects. The objects are regenerated at high resolution once you release the cursor at the end of each editing operation. Use the various options in the Setup menu to affect the display of the graphics before they are tacked down onto the image.



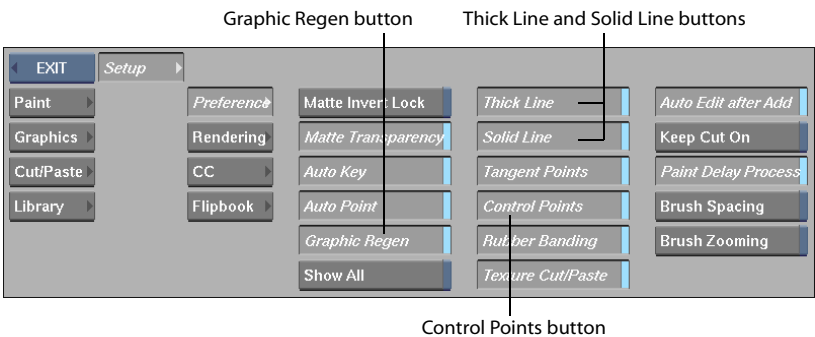
To set the object resolution:

- Select an option from the Object Resolution box.

Select:	To:
Full Res Raster	Display the object at full resolution when updating. This is helpful for displaying gradients, but slows down the rate at which image display is refreshed.
Low Res Raster	Display the object at low resolution when updating.
Wire Frame	Display the object as a wireframe when updating.

Wire Frame Display Options

The Thick Line and Solid Line buttons in the Preferences Setup menu determine the thickness and continuity of the wire frame.



Enabling and disabling the Thick Line and Solid Line buttons affects the wireframe as follows:

- When the Thick Line button is enabled, the wire frame has a thickness of 2 pixels.
- When the Thick Line button is disabled, the wire frame has a thickness of 1 pixel.
- When the Solid Line button is enabled, the wire frame is unbroken.
- When the Solid Line button is disabled, the wire frame is dashed.

Increasing Editing Speed

To speed up editing, disable the Graphic Regen button in the Preferences Setup menu. Instead of waiting for the graphics objects to be regenerated at the end of an editing operation, the objects appear at the display resolution. This can be a useful time-saving feature.

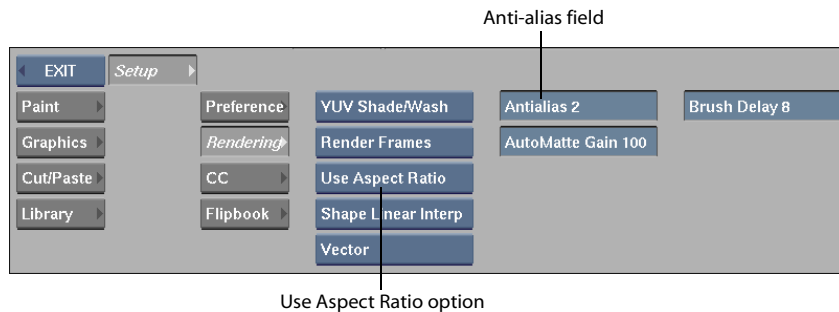
Hiding the Object Control Points

Each object is defined by two or more control points. These control points, or edit handles, appear while you are drawing or editing the object.

Disable the Control Points button in the Preferences Setup menu to turn off the display of the control points while you are drawing or editing the object. This feature is especially useful if you want to trace the outline of a figure.

Displaying Anamorphic Geometry

Select Use Aspect Ratio in the Rendering menu to draw and display anamorphic geometry. For normal display, click Use Aspect Ratio and switch to Use Square Pixel.



Anti-Aliasing for Objects

The jagged edges that can occur along diagonal and curved lines in geometry are caused by aliasing, or insufficient spatial sampling of the image. The process of minimizing the jagged edges by increasing the sampling rate is called anti-aliasing.

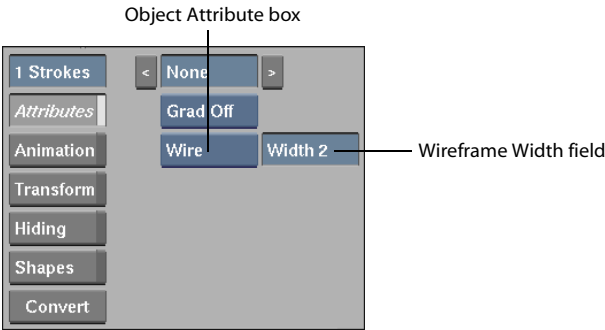
The Anti-alias field in the Rendering Setup menu enables you to specify an anti-aliasing factor for geometry. When the value is set to 1, no anti-aliasing occurs. The best results are obtained at a value of 4.

Setting the Object Attribute

Use the Object Attribute box to set the display mode for the object that you add to the image.

To change the object attribute:

1. Click Edit in the Graphics menu.
2. Select the object that you want to edit.
3. Click Attributes.
4. Select an option from the Object Attribute box.



Select:	To:
Solid	Draw a filled object with a well-defined edge. Use the Brush Opacity field to set the object's transparency. No other brush attributes or types are available. Line objects cannot be set to Solid.
Outline	Draw an outline of the object. Use the Brush Attribute buttons to set the appearance of the outline. For more information, see "Changing the Object's Appearance" on page 1132.
Fuzzy	Draw a solid object with a soft edge. Use the Brush Attribute fields to set the appearance of the outline.
Wire	Draw a wireframe object. Use the Brush Opacity field to set the object's transparency. No other brush attributes or types are available. Use the Wireframe Width field located to the right of the attribute box to set the line width of the wireframe.

Changing the Object's Appearance

You can change the appearance of an object by changing any of the following display attributes:

- Current colour
- Brush attribute values or modes
- Special Effects media or filters
- Colour gradient

Changing display attributes affects all currently selected objects. For information on selecting objects, see “Selecting Objects” on page 1126.

Setting Brush Attributes and Modes

In the Graphics menu, you can only change brush attributes and modes when using the Outline or Fuzzy attribute. For more information, see Chapter 50, “Paint: Setting Brush Attributes and Modes.” The Brush Attribute controls may not be visible. Swipe the cursor at the bottom of the screen once or twice to display the Brush Attribute controls.

Use the Current Colour field to set the graphic's colour, and the Opacity field to set the opacity of the object, or the effect a graphic has on the image.

Using Filters and Special Effects Media with Graphics

Filters and Special Effects media can be used with objects and any filter can be used. Only the Paint, Clone, Reveal, Wash, and Shade media can be used in the Graphics menu.

For more information, see Chapter 50, “Paint: Setting Brush Attributes and Modes.”

Changing the Object Gradient

All objects, except the fill object, can have a gradient. You can edit a gradient or apply a gradient to an object that does not have one. A selected object with a gradient displays a gradient control bar that you can use to edit the orientation of the gradient.

To edit a colour gradient:

1. Click Edit in the Graphics menu.
2. Select the object you want to edit.
3. Click Attributes.
4. Select an option from the Gradient box:

Select:	To:
Gradient Rect	Use a rectangular gradient in which colour changes from top to bottom.
Gradient Circ	Use a circular gradient in which colour changes from centre to edge.
Gradient Off	Turn off the gradient and use a solid colour.

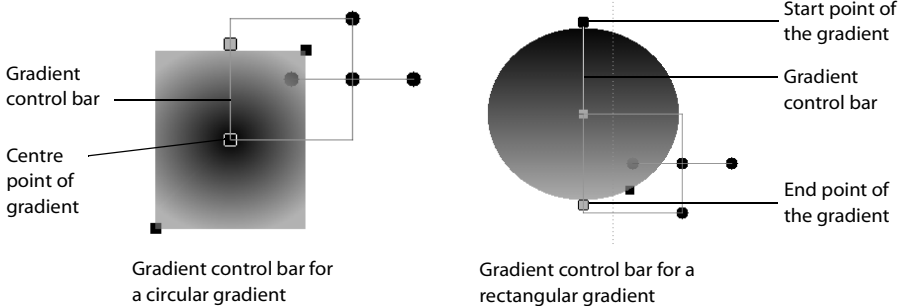
5. Set the colours for the gradient.

For more information, see “Using the Colour Gradient Bar” on page 1065.

6. Use the gradient control bar to change the object's gradient.

Using the Gradient Control Bar

When you select an object with a gradient, Paint displays a gradient control bar in addition to the object transformation box. The gradient control bar has two handles at each end that show the colours of the gradient. In a rectangular gradient, the bar also indicates the direction of the gradient.



NOTE: If you want to move the Transformation box, hold **M** and click on the location where you want to move the box.

To use the gradient control bar:

1. Select an object with a gradient, or add a gradient to an object.

Paint displays the object's transformation box and the gradient bar.

2. Move the handles of the gradient control bar to change the orientation and location of the gradient.

In a rectangular gradient, this affects the gradient orientation. For example, the default orientation for a rectangular gradient is top to bottom. If you drag one of the handles left or right, you change the orientation to diagonal. If you drag the handles so the gradient bar is horizontal, you get a rectangular gradient with the colours blended from side to side.

If you select the centre colour handle in a circular gradient, you can move the gradient's centre inside the object or even outside it.

3. To decrease the amount of one colour visible in an object, move the handle to the edge or outside of the object.

Changing the Size or Position of an Object

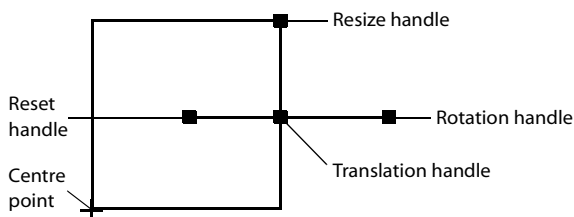
Use the transformation box, or the transformation controls in the Transforms menu to change the size or position of an object. You can use the transformation box and controls to modify the object in the following ways:

- Move the object in any direction on the image
- Rotate the object about its centre point
- Change the size of the object

You can translate, resize, or rotate more than one object at the same time by selecting all the objects that you want to edit.

Using the Transformation Box

To display the transformation box, click the Edit button in the Graphics menu and click the object you want to edit.



The position of the transformation box is saved for each object. To move the transformation box, hold **M** and click on the destination.

The Resize Handle

To change the size of an object, drag the resize handle in any direction until you are satisfied. To maintain an object's proportions, hold **P** as you drag the resize handle.

The Centre Point

Move the centre point to change the point around which the object rotates.

The Rotation Handle

Drag the rotation handle to rotate the object about its centre point.

The Translation Handle

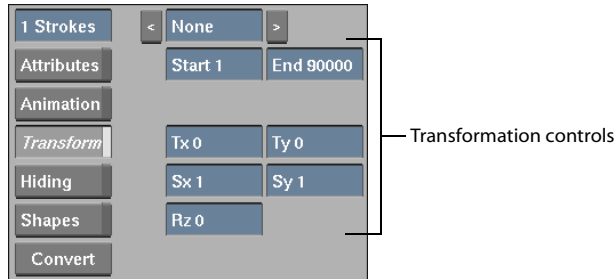
Drag the translation handle to move the object in any direction on the image. The transformation box moves with the object.

The Reset Handle

Click the reset handle to restore the object to its initial size and position on the image.

Using the Transformation Controls

Use the transformation controls to translate, rotate, and resize a selected object. To display the transformation controls, select an object and click the Transform button.



Horizontal Translation

Use the Tx (translate x) field to move the object along the horizontal axis (X-axis).

Vertical Translation

Use the Ty (translate y) field to move the object along the vertical axis (Y-axis).

Horizontal Resizing

Use the Sx (scale x) field to change the size of the object along the horizontal axis.

Vertical Resizing

Use the Sy (scale y) field to change the size of the object along the vertical axis.

Rotation

Use the Rz (rotate z) field to rotate the object about the Z-axis. The object is always rotated about its centre point.

Reset

The Reset button resets the object to its initial size and position.

Setting the Start and End Points for an Object

Use the Start and End numeric fields to set when the object appears in the clip.

Changing the Shape of an Object

To change the shape of an object, such as a polygon or a line, you must add, delete, or move a control point on the object.

To change the shape of an object:

1. Click Edit in the Graphics menu.
2. Select the object you want to edit.
The object control points and the transformation box appear on the selected object. For more information, see “Changing the Size or Position of an Object” on page 1134.
3. Press the control point you want to edit and drag it to a new location. When you are satisfied with the new position, release the cursor.

Adding and Deleting Vertices on a Polygon or Line

Change the shape of a polygon or line by changing its number of vertices.

To add a vertex to a polygon or line:

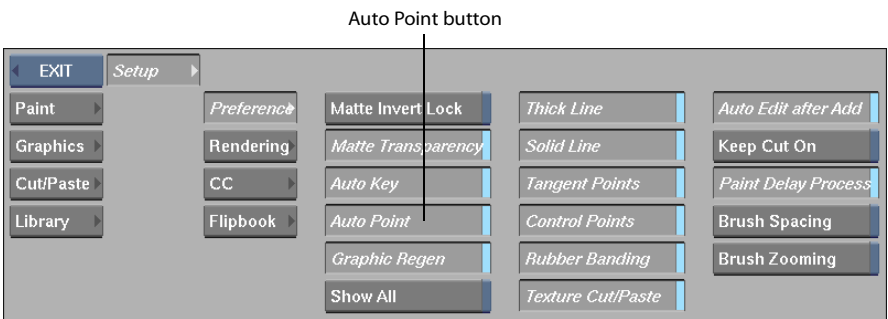
1. Click Edit in the Graphics menu.
2. Select the polygon or line you want to edit.
The control points for that object are selected.
3. Press **A**.
4. Press on one of the existing control points, and drag outward. A control point is added counterclockwise to the selected point.
To add a point clockwise to the selected point, press **CTRL+A**.
5. Repeat steps 3 and 4 for each new vertex you want to add to the object.

To delete a vertex from a polygon or line:

1. Click the Edit button in the Graphics menu.
2. Select the polygon or line you want to edit.
The control points for that object are selected.
3. Press **D**.
4. Click the control point you want to delete.
The selected control point is deleted from the object.
5. Repeat steps 3 and 4 for each control point you want to delete on the object.

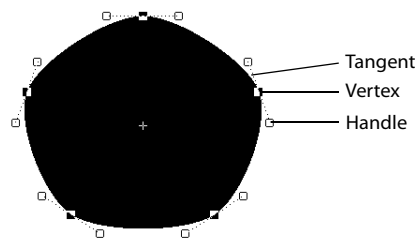
The Auto Point Command

When you add or delete a control point on a polygon or line, the same point is added or deleted on all shapes for that object. You can disable this option in the Preferences Setup menu. When the Auto Point button is disabled, any point that you add or delete on a shape is added/deleted in the current shape key only.

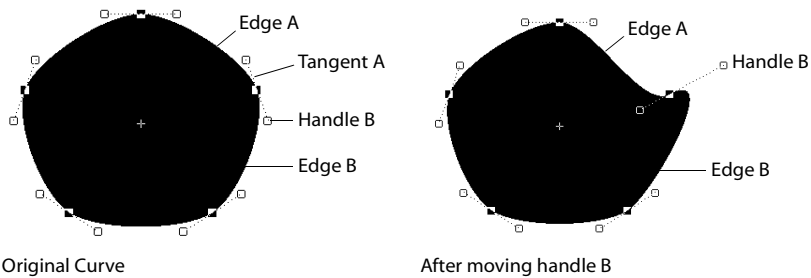


Editing a Bezier Curve

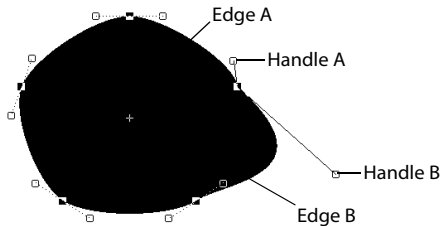
When you use the Bezier option to draw a polygon, each vertex of the polygon has two tangents. Each tangent ends with a handle.



Use Edit mode to move a tangent handle and adjust the slope of the adjacent side of the polygon. When you move one tangent handle, the tangent's reciprocal handle moves in the opposite direction because handles A and B are joined to keep the joint at the intersection of the two edges smooth.

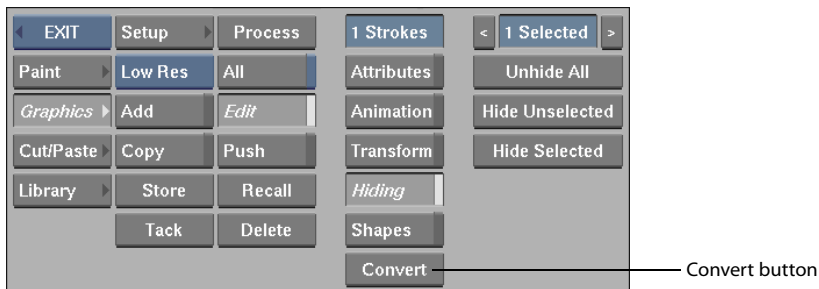


To adjust the slope on one side of the vertex only, press **B** and click the vertex. This breaks the tangent so that you can move its handles independently. To reset the tangent, press **B** and click the vertex again.



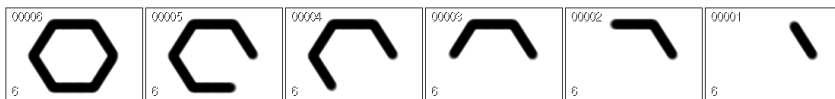
Converting Graphics to Autopaint Strokes

Use the Convert command to convert objects into Autopaint strokes.



You can play back the paint strokes using the Autopaint controls in the Paint menu. For more information, see Chapter 52, “Paint: Using Wipe, Fill, and AutoPaint.”

The following shows how a hexagon is drawn over six frames. One edge is drawn in each frame. After the object is converted to Autopaint strokes, it is always drawn as an outline regardless of its attributes in the Graphics menu.



To determine how much of an object to draw in each frame, **flame** divides the total number of edges in the object by the number of frames in the clip. The polygons are drawn one after the other in the order in which they were drawn in the Graphics menu.

To convert an object to Autopaint strokes:

1. Select the object.
2. Click Convert and Confirm.

Animating Graphics

Use the Channel Editor and Shape Animation controls to animate Paint graphics. You can animate the position, display, and shape of a graphic. To display the Channel Editor, click the Animate button and swipe the cursor across the bottom of the screen.

For more information on using the Channel Editor, see Chapter 9, “Animation.”

Animation Parameters

The following parameters can be animated for every object:

- Position, rotation, and size
- Brush attributes
- Gradient orientation, transparency, and colour
- Colour (red, green, and blue channels)
- Wireframe width
- Range and Softness of a fill object
- Transparency
- Size, Kerning and Italics for a text object

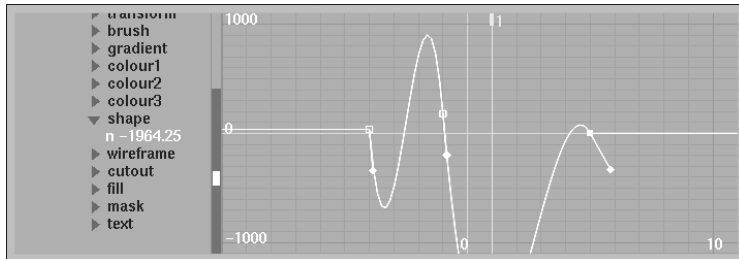
NOTE: In Paint, the commands in the Edit Mode menu can be performed on animation curves only. To modify a keyframe by editing the object in the image window, you must use the editing commands in the Graphics menu.

Shape Animation

To animate the shape of an object, you must create keyframes with the Shape Animation controls. Each shape you define becomes a keyframe in the Shape channel of the Channel Editor. The difference between the keyframes is interpolated and the shape animation is created.

The Shape channel is used to identify the number and location of shape keyframes in a clip. Use the shape curve to control the rate at which an object changes to a new shape. The following

figure shows the shape channel for an object that changes from shape 1 in frame 1, to shape 2 in frame 8, and shape 1 in frame 15.



To animate the shape of an object:

1. Create an object.

This original shape becomes shape keyframe 1 in the Channel Editor.

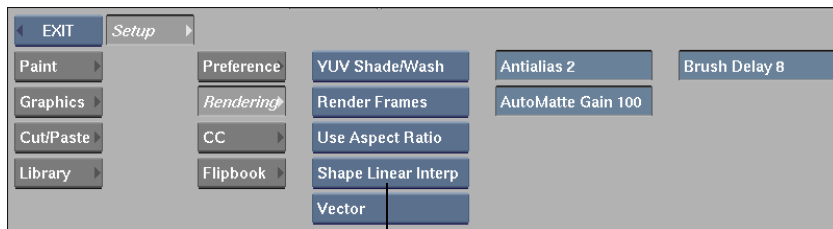
2. Advance to another frame in the clip.

3. Select the object, and change its shape.

To change its shape, you must move, add, or delete a control point on the object. This new shape is shape keyframe 2 in the Channel Editor.

Shape Interpolation

Use the Shape Interpolation box in the Rendering Setup menu to specify the interpolation between shapes in the animation.



Shape Interpolation box

Select:

Shape Linear Interp

Shape Cardinal Interp

To:

Produce sudden transitions between shapes.

Produce smooth transitions between shapes.

NOTE: The type of shape interpolation you use is independent of the type of polygon interpolation you use.

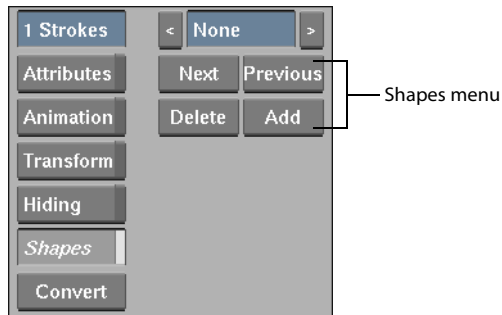
Moving between Shape Keyframes

Use Next and Prev to move between shape keyframes in an animation.

To move between keyframes:

1. Select the object you want to modify.
2. Click Shapes.

The Shapes menu appears.



3. Click Next to advance to the next shape keyframe.
4. Click Prev to go to the previous shape keyframe.

Deleting Keyframes

Use Delete to delete a shape keyframe from the animation.

To delete a shape keyframe:

1. Select the object.
2. Click Shapes.

The Shapes menu appears.

3. Select the keyframe you want to delete.
4. Click Delete in the Shapes menu.

The selected shape is deleted.

Adding Keyframes

Use Add to create a shape keyframe. This can be useful for creating animations that start and end with the same shape.

To add a shape key using Add:

1. Select the object you want to animate.
2. Go to the frame where you want to add the shape keyframe.

3. Click Shapes.

The Shapes menu appears.

4. Click Add.

A new keyframe is added to the shape channel in the Channel Editor.

Gradient Animation

Use the Channel Editor to animate the colours, orientation, and direction of a colour gradient. The Gradient folder includes Start and End folders for the two colours in the gradient. The Start and End folders each contain channels for the X and Y position of the colour, as well as the R, G, B, and alpha values.

Although you can animate the gradient independently, the gradient will by default follow any transformations of the object.

To animate a gradient:

1. Select the object with the gradient you want to animate.
2. Click Animation. If necessary, swipe the cursor across the bottom of the screen to display the Channel Editor.
3. Click the Geometry and Stroke folders to expand them.
If you selected more than one object, a Stroke folder appears for each selected object. Click the Stroke folder for the object you want to animate.
4. Click the Gradient folder to expand it, and click on the Start and End folders.
The Start folder contains the X and Y channels and a Colour folder for the gradient's colour
1. The End folder contains the X and Y channels and a Colour folder for the gradient's colour
2. Click to expand each Colour folder.

Use: To:

x	Move colour origin left or right. This is the same as moving the gradient bar handle.
y	Move colour origin up or down. This is the same as moving the gradient bar handle.
r	Change percentage of red in the colour.
g	Change percentage of green in the colour.
b	Change percentage of blue in the colour.
a	Change percentage of the colour's alpha channel. Use this channel to create a gradient matte or to change the opacity of the colour.

5. Change the values for the channels in different frames to create a gradient animation.

Processing the Animation

To process the animation, click Process in the Graphics menu. Make sure you are at the first frame of the animation since the clip will be processed from the currently displayed frame until the end of the clip.

Saving and Restoring Objects

Use the Store and Recall commands to save and restore the objects to the image. Use the Library menu to save or load graphics from the graphics library.

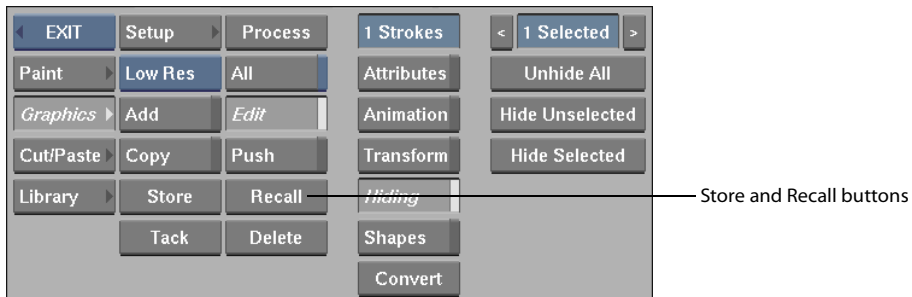
To save or load a graphic:

1. Click Library in the Paint menu.
2. Click Load or SaveAs.
3. Select Geometry.
4. Select a file or type the name of the graphic to be loaded or saved.

The Store and Recall Commands

To save the selected graphic in the buffer, click Store in the Graphics menu. The current object replaces the graphic saved in the buffer.

The Store command does not save the result image. This means that if you tacked the objects onto the image, they are not saved using the Store command. To restore the geometry to the image, click Recall.



HINT: Use Store and Recall to add graphics to clips loaded at a later time.

Tacking Down Objects

An object can be selected, deleted, edited, and transformed until it is tacked down on the image. As soon as it is tacked down, the object becomes part of the image and can no longer be manipulated.

There are several ways to tack down an object:

- You can select a Tack mode so that an object is either tacked down as soon as it is drawn, or as soon as the next object is drawn.
- You can tack down selected objects only.
- You can tack down all objects simultaneously using the Tack All command.

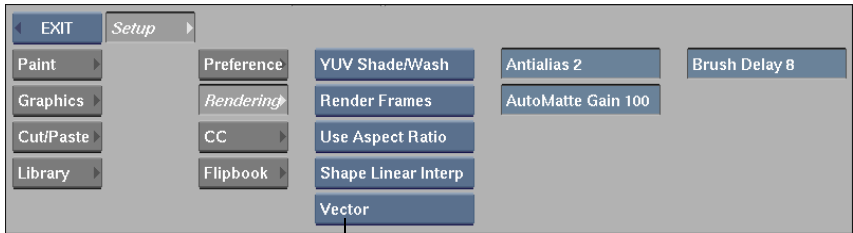
The Tack Modes

The Tack mode is selected with the Tack Mode box in the Rendering Setup menu.

To set the Tack mode:

1. Click Setup in the Paint menu.
2. Click Rendering.

The Rendering menu appears.



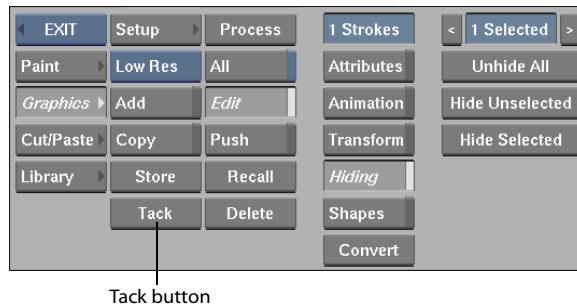
Tack Mode box

3. Select an option in the Tack Mode box.

Select:	To:
Vector	Explicitly tack down the objects using the Tack command.
Edit Last	Edit the last object drawn until the next object is added to the image. As soon as the next object is added, the last object is tacked down.
Raster	Tack down an object as soon as it is added to the image.

Tacking Down Selected Objects

When the Tack mode is set to Vector, objects must be explicitly tacked down on the image.



Tack button

To tack down one or more objects:

1. Select the objects you want to tack down.
2. Click Tack.
All selected objects are tacked down on the image.

To tack down all objects on the image:

1. Enable All in the Graphics menu.
2. Click Tack.
All objects are tacked down on the image.

This image shows a full page of white paper with horizontal dashed lines. The lines are evenly spaced and run across the width of the page, providing a guide for handwriting practice. There are no margins, text, or other markings on the page.

Paint: Cut and Paste

Stick up

Use the Cut and Paste tools to remove portions of your image or the entire frame, tack down portions of the result clip, and add effects to your cutouts.

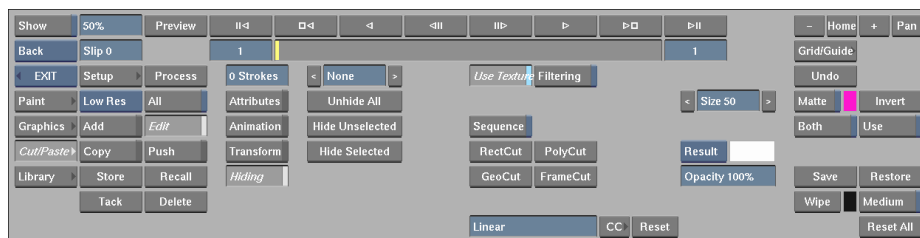
Summary

In this chapter, you learn about:

- “Accessing the Cut and Paste Menu” on page 1147
- “Creating a Cutout” on page 1148
- “Using the Cutout Commands” on page 1149
- “Increasing Editing Speed” on page 1151
- “Using the Paste Mode Options” on page 1152
- “Colour Correcting Cutouts” on page 1156
- “Animating Cutouts” on page 1156
- “Loading and Saving Cutouts” on page 1156

Accessing the Cut and Paste Menu

To access the Cut/Paste menu, click Cut/Paste in the Paint menu.



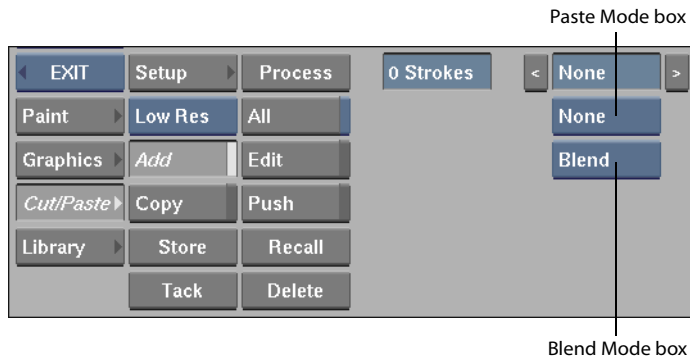
Creating a Cutout

Creating a cutout is similar to creating a graphic. In the Cut/Paste menu, use the same controls as you would in the Graphics menu to select, copy, hide, move, rotate, resize, delete, store, recall, push and tack cutouts. For more information, see Chapter 54, “Paint: Using Graphics.”

To create a cutout:

1. Click Cut/Paste.
2. Click Add.

The Paste Mode and Blend Mode boxes appear.



3. Enable Sequence if you want to create a cutout that spans the entire clip.
4. Enable Use in the Matte controls to limit the area of the cutout with a matte.

For more information, see Chapter 53, “Paint: Using Mattes.”

5. Select an option from the Paste Mode box.

The Paste Mode box determines how the cutout will be pasted onto the result clip. For more information, see “Using the Paste Mode Options” on page 1152.

6. Click one of the four Cutout commands.

For more information, see “Using the Cutout Commands” on page 1149.

NOTE: For the Geocut option, select all of the objects you want to use in the cutout in the Graphics menu before clicking the Geocut button.

7. Draw the cutout on the image.
 8. Set the resolution to be used when updating the object.
- For more information, see “Setting the Object Resolution and Display” on page 1128. The object appears at full resolution when it is drawn.

9. Set the cutout’s position using the Transformation box or controls.

For more information, see “Changing the Size or Position of an Object” on page 1134.

10. Set the cutout's colour using the Current Colour field.

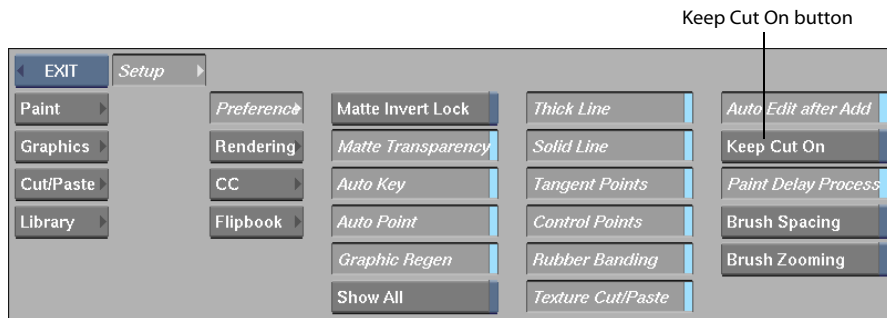
For more information, see Chapter 48, “Paint: Overview.”

11. Click Tack to permanently place the object on the result clip.

Once the object is tacked down, it cannot be moved, rotated, deleted, resized, or copied.

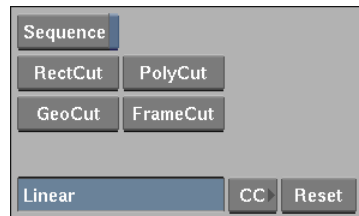
Keep Cut On

By default, **flame** automatically switches from Cut mode to Edit mode after you add a cutout to the image. You can disable this behaviour by enabling Keep Cut On in the Preferences menu.



Using the Cutout Commands

Four commands are used to create cutouts: FrameCut, RectCut, PolyCut, and GeoCut. FrameCut, RectCut, and PolyCut create cutouts using shapes you define in the Cut/Paste menu. Geocut creates cutouts using objects you select in the Graphics menu.



The FrameCut Command

Use the FrameCut command to make a cutout of the entire frame.

To create a cutout of an entire frame:

1. Click Add in the Cut and Paste menu.
2. Click FrameCut.

Paint cuts out the entire frame and pastes the cutout onto the image.

The RectCut Command

The RectCut command is used to cut and paste a single rectangular area of the image.

To use the RectCut command:

1. Click Add in the Cut and Paste menu.
2. Click RectCut.
3. Draw a rectangle over the area of the image that you want to cut out.
The image area defined by the rectangle is copied and pasted onto the image.

The PolyCut Command

The PolyCut command is used to cut and paste a single polygonal area of the image.

To use the PolyCut command:

1. Click Add in the Cut and Paste menu.
2. Click PolyCut.
3. Draw a polygon over the area of the image that you want to cut out.
The image area defined by the polygon is copied and pasted onto the image.

The GeoCut Command

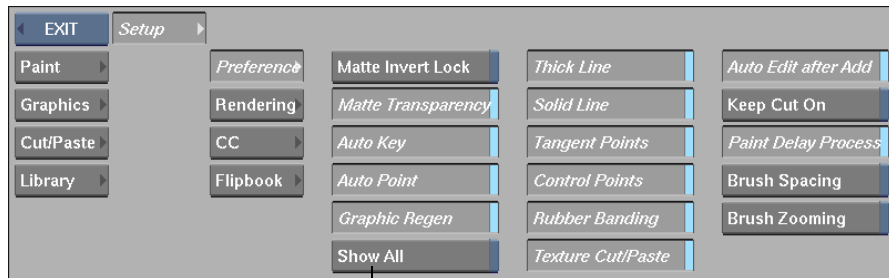
Use the GeoCut command to create a cutout with objects you selected in the Graphics menu.

To create a cutout using GeoCut:

1. Click Graphics in the Paint menu.
The Graphics menu appears.
2. Create any number of objects.
For more information, see Chapter 54, “Paint: Using Graphics.”
3. Select all the objects that you want to use to make the cutout.
4. Set the opacity of the object(s) using the Opacity field.
The opacity of the object(s) used in the cutout determines the transparency of the cutout. Use objects with a low opacity value to create an opaque cutout. Use objects with a high opacity value to create a transparent cutout.
5. Click Cut/Paste in the Paint menu.
The Cut and Paste menu appears.
6. Click GeoCut.
The image area defined by the selected objects is copied and pasted onto the image.

Show All

To display both the cutouts and the graphics objects in the Cut and Paste menu, enable Show All in the Preferences menu. Although the graphics objects appear, they cannot be edited. Enabling Show All also displays both the graphics objects and the cutouts in the Graphics menu. Although the cutouts appear, they cannot be edited.



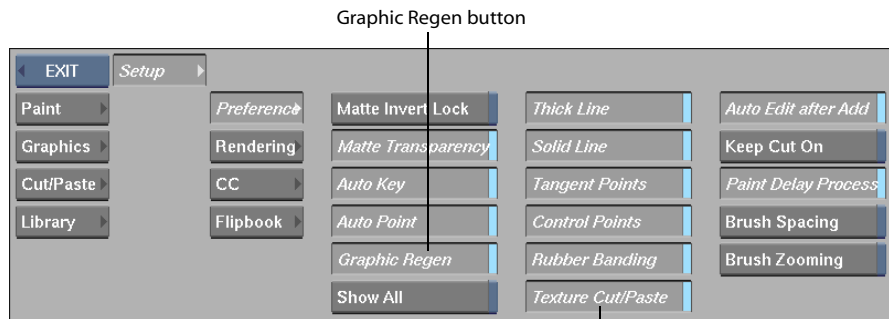
Show All button

Increasing Editing Speed

To streamline cut and paste functionality, use Graphic Regen and Texture Cut/Paste in the Preferences menu, and Use Texture.

The Graphic Regen Button

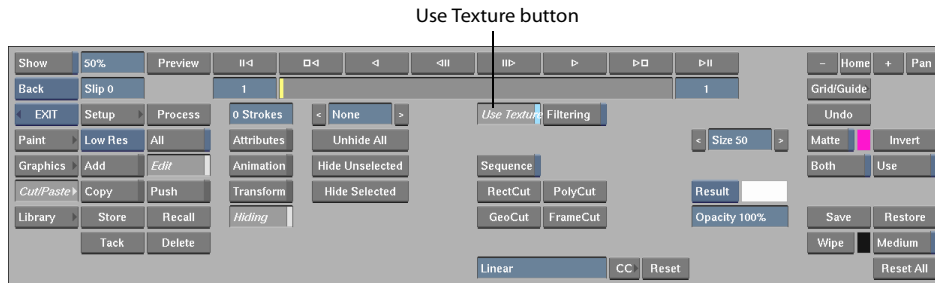
Disable Graphic Regen (regeneration) in the Preferences menu to ensure the cutout always appears at the display resolution.



Texture Cut/Paste button

Texture Cut/Paste and Use Texture button

Enable Texture Cut/Paste in the Preferences menu or Use Texture in the Cut and Paste menu to speed up cut and paste operations. These preferences are designed for platforms that support fast texture mapping.



The Filtering button appears when Use Texture is enabled. Use Filtering to avoid image degradation as a result of recursive cutting and pasting. When Filtering is enabled, the image should not degrade.

Using the Paste Mode Options

Four options are available for pasting a cutout onto the image.

Select:	To:
Emboss	Create an embossed cutout.
Extrude	Create cutout extrusions.
None	Use the blend functions available for pasting cutouts.
Shadow	Add a drop shadow to the cutout.

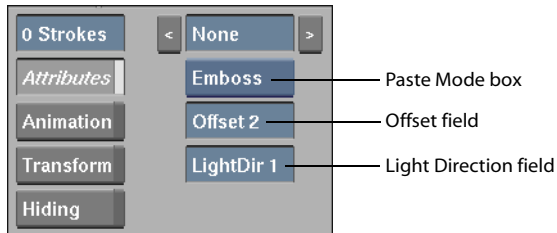
Using the Emboss Option

The Emboss option makes the cutout appear etched into the image. Shadows and highlights are added to simulate the effect of light shining on a raised object. Three layers are used to produce this effect:

- The top layer is a copy of the cutout in which the luminance values have been increased by 50%. The top layer adds the highlights to the cutout.
- The middle layer is the cutout.
- The bottom layer is a copy of the cutout in which the luminance values have been decreased by 50%. The bottom layer adds the shadow to the cutout.

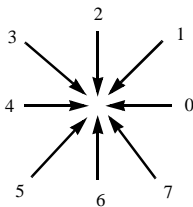
Offset Field

Use the Offset field to specify the offset value in pixels for the three layers. The best results are obtained with values of 1 or 2.



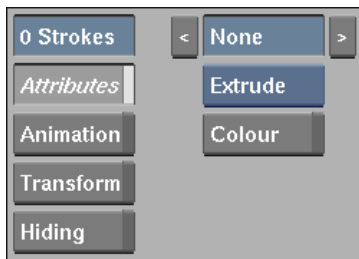
Light Direction Field

Use the Light Direction field to specify the angle of the simulated incident light. The values range from 0 to 7, and specify the following angles of incident light:



Using the Extrude Option

The Extrude option makes the cutout appear to pop out of the image by redrawing the cutout one layer on top of another. Each layer in the stack is offset slightly from the one below so that only the edges of the layer are visible.



Editing the Layers

When you create a cutout using the Extrude option, two layers are created to produce the effect. There are control points at the center of each layer for manipulating the layer. You can change the depth and direction of the extrusion by moving either of the layers.

Changing the Colour of the Layers

Enable Colour to apply the current colour to the extrusion layers. The cutout image is used for the top layer only. When Colour is enabled, use the Opacity field to set the transparency of the extrusion layers.

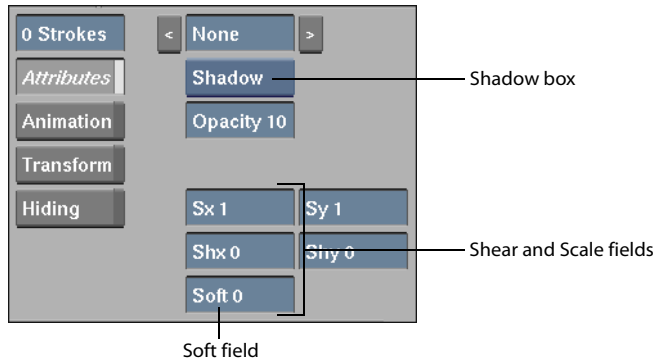
Using the Blend Options

The Blend options are similar to the commands in the Processing menu. They use the RGB values of the cutout and those of the result image. Select the Blend option in the Blend Mode box.

Select:	To:
Blend	Paste the cutout onto the result image.
Add	Add the RGB values of the corresponding pixels in the cutout and the result image. Values greater than 255 are clipped.
Subtract	Subtract the RGB values of pixels in the image from those of the cutout. Value less than 0 are clipped.
Multiply	Multiply the RGB values of corresponding pixels in the cutout and the result image. The value is normalized by dividing the result by 255.
Black	Paste a black object in the shape of the cutout on the result image.
White	Paste a white object in the shape of the cutout on the result image.
Colour	Paste a coloured object in the shape of the cutout on the result image. The current colour is used for the object.
NAddMin	Compare the brightness values of corresponding pixels in the cutout and the result image, and use the pixel with the smaller value in the tacked down cutout.
NAddMax	Compare the brightness values of corresponding pixels in the cutout and the result image, and use the pixel with the greater value in the tacked down cutout.

Using the Shadow Option

You can add a drop shadow to a cutout by selecting the Shadow option in the Paste Mode box. The drop shadow's colour is taken from the Current Colour field.



Shadow Field

Use the Shadow field to adjust the transparency of the drop shadow. Decrease the value in this field to increase the transparency.

Soft Field

Use the Soft field to adjust the edge softness of the drop shadow. Increase the value in this field to increase the softness of the shadow.

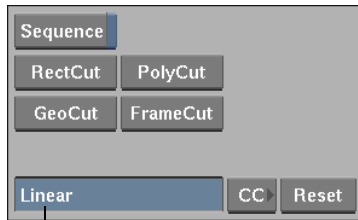
Shear and Scale Fields

Use the Shx and Shy fields to shear the drop shadow along the X- or Y-axis. The Sx and Sy fields are used to scale the drop shadow along the X- and Y-axis. These fields can be used to add lighting perspective to the image.

Colour Correcting Cutouts

Apply colour correction to the cutouts using the Colour Correction field and CC in the Cut and Paste menu. Click the Colour Correction field to load a setup directly from the colour correction library. Click the CC to load the cutout into the Colour Corrector.

The default colour correction setup is Linear. When you load a different setup, the name of the setup appears in the Colour Correction field. To reset the colour correction setup, click Reset in the Cut and Paste menu.



Colour Correction box

Animating Cutouts

Use the Channel Editor to animate cutouts. To display the Channel Editor, click Animate and swipe the cursor across the bottom of the screen.

For more information on using the Channel Editor, see Chapter 9, “Animation.”

Animation Parameters

The following parameters can be animated for every cutout:

- Position, rotation, and shearing
- Brush attributes
- Colour (red, green, and blue channels)
- Emboss options
- Shadow options

Loading and Saving Cutouts

Use the Cutout Library to load and save cutouts.

For more information, see Chapter 56, “Paint: Creating, Loading, and Saving Setups.”

To save or load a cutout:

1. Click Library in the Paint menu.
2. Click Save or Load.
3. Click CutOut.
4. Select a file or enter the name of the cutout to be saved or loaded.

Paint: Creating, Loading, and Saving Setups

Set it up

Create your own setups in Paint, save them to the appropriate setups directory, and load them during a later session.

Summary

In this chapter, you learn about:

- “Loading Setups” on page 1159
- “Removing Setups” on page 1160
- “Creating and Saving Brush Setups” on page 1160
- “Creating a Custom Brush” on page 1161
- “Using Colour Correction Setups” on page 1166

About Paint Setups

Both the procedure and the interface involved in saving Paint setups and preferences differ slightly from those involved in saving setups within most other modules. For more information on saving setups and preferences in most other modules, see Chapter 7, “Saving Setups and Preferences.”

NOTE: Custom brushes, brush sets, colour pots, and palette setups that you create in Paint are stored in your user directory. These resources are loaded when you specify a user in the Project Management menu.

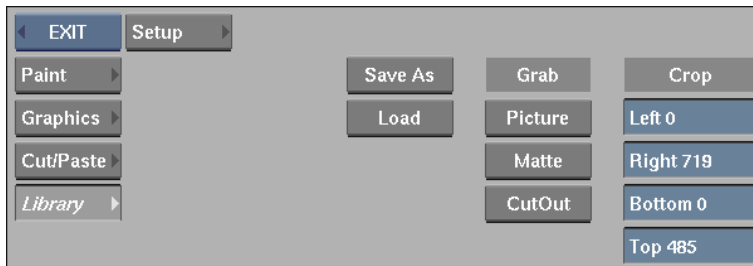
About Saving Setups

Use the Library menu to save pictures, preferences, mattes, palettes, brush groups, brushes, stamps, geometry created in the Graphics menu, cutouts, cut sequences, and recorded strokes in Autopaint.

To save a setup:

1. Click Library in the Paint menu.

The Library menu appears.



2. Click Save As.

This has the effect of toggling from Load to Save mode within the library. You can now save desired setups and preferences.

3. Click the directory button corresponding to the item you want to save.

Click:	To:
Picture	Save an image created in Paint.
Matte	Save the matte.
Palette	Save the current colour palette.
BrushSet	Save the current set of brushes.
Brush	Save the brush that is currently active.
Stamp	Save the Stamp currently in the Stamp window.
Geometry	Save the geometry created in the Graphics menu. This also includes all cutouts.
Cutout	Save the cutouts created in the Cut and Paste menu.
CutSequence	Save the cutout sequence created in the Cut and Paste menu.
AutoPaint	Save the last series of recorded strokes.

The file browser appears, listing any existing setups for that item.

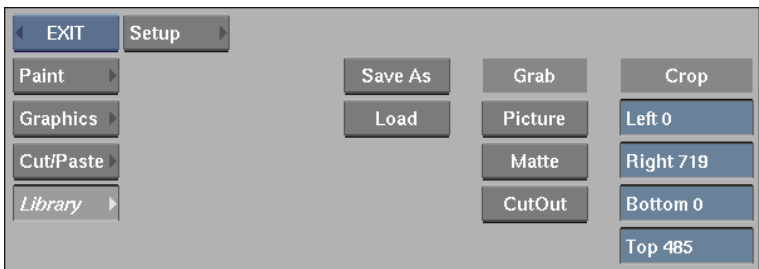
4. Press **ENTER** to save the setup.

Cropping a Setup

Use the crop box to limit the area of the picture, matte, cutout, or graphic to be saved in the setups directory.

To grab a matte, image, graphic, or cutout:

1. Click Library.



The Grab menu appears. Here you can specify which element from the image you want to save.

2. Draw the crop box on the canvas by pressing and dragging the cursor diagonally across the screen. Alternatively, use the left, right, top, and bottom Crop fields to set the boundaries of the crop box.

NOTE: Click Reset to reset the crop box values.

3. Name the setup in the keyboard display.
If you want to overwrite an existing setup, select the file name from the file browser.
4. Press **ENTER** to save the setup.

Loading Setups

Use the Load button to load setups in Paint.

To load a setup:

1. Click Library.
The Library menu appears.
2. Click Load.
3. Click the button corresponding to the item you want to load.
A list of existing setups appears.
4. Select the title or proxy of the setup you want to load.

Removing Setups

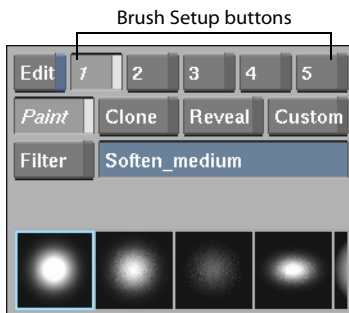
Use the Remove button to remove setups from the setup directory.

To remove a setup:

1. Click Library.
The Library menu appears.
2. Click Remove.
3. Click the button corresponding to the item you want to remove.
A list of existing setups appears on the screen.
4. Select the title or proxy of the setup you want to remove.

Creating and Saving Brush Setups

Within a given brush set, there are five individual brushes. You can create and save five different brush setups using the buttons numbered 1 to 5 in the Paint menu.



To create and save a brush setup:

1. Click one of the five Brush Setup buttons.
2. Enable Edit next to the Brush Setup buttons.
3. Set the brush attributes.
Each brush setup saves the brush type, attribute values and modes, Special Effects medium, filter, fill parameters, and colour correction setups.
4. Disable Edit to save the changes to the selected brush setup.

To save a set of five brush setups, use BrushSet in the Library menu. See “About Saving Setups” on page 1158.

Copying Brush Setups

You can copy brush setups using the setup buttons.

To copy brush setups:

1. Select a setup using the Brush Setup buttons and enable Edit.
2. Press **CTRL** and click another Brush Setup button.
3. Disable Edit to save the setup.

Creating a Custom Brush

Paint allows you to create your own custom brushes for specialized applications. You can modify an existing brush, draw your own brush shape, or grab a part of the image to use as a custom brush.

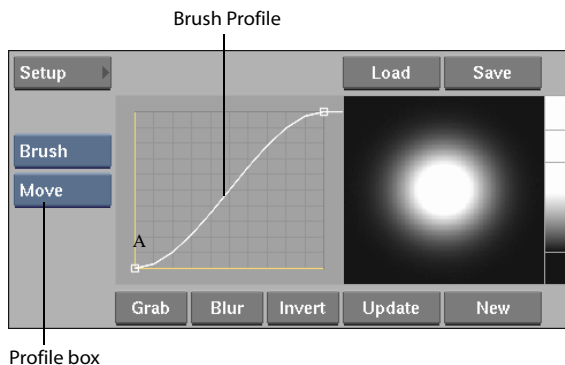
Use the Custom Brush menu to create your own brush. To display the Custom Brush menu, double-click one of the brush icons in the Brushes window. An enlarged view of the selected brush appears in the editing panel.

There are three ways to create a custom brush:

- Using the brush profile
- Drawing a brush shape in the editing panel
- Grabbing an area of the image

Changing the Brush Profile

The brush profile describes the size and edge softness of the custom brush. The default S-curve defines a circular airbrush. You can change the curve by manipulating the two points that define the curve, or you can add points to the curve.



The profile does not affect the brush in the editing panel until you modify the curve or click the Update button.

Changing the Curve

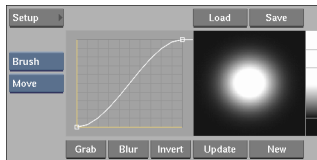
By default, there are two points that define the brush profile. For the following table and examples, these points will be named A and B. To move the points that define the curve, select the Move option from the Profile box.

Move:	To:
Point A up	Darken the brush.
Point B down	Lighten the brush.
Point A right	Harden the outer edge of the brush.
Point B left	Soften the middle of the brush.

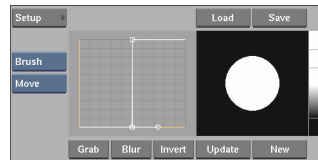
To move the points only on the X-axis, select the Xscale option from the Profile box. To move the points only on the Y-axis, select the Yscale option from the profile box. To break a point and manipulate its tangent, select the Break option from the Profile box and click on a point.

Adding Points to the Curve

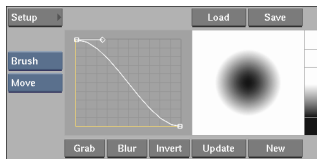
To add points to the curve, select the Add option from the Profile box and click on the curve. To delete points from the curve, select the Delete option from the Profile box and click on one of the points on the curve.



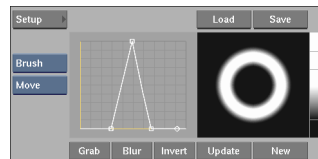
Default S-shaped curve



Hard edge curve



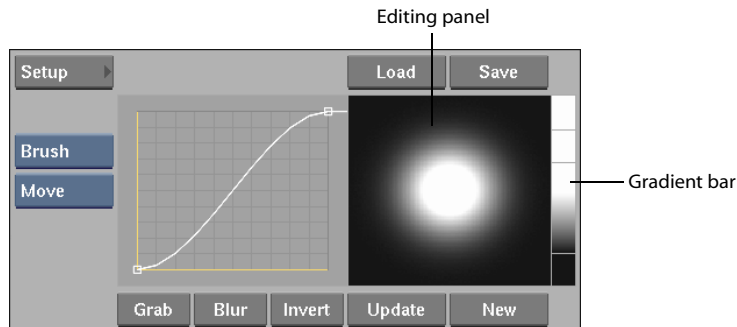
Invert brush curve



Ring-shaped brush curve

Drawing and Updating a Brush Shape

You can draw a new brush shape or update the current brush shape in the editing panel. The new shape is drawn using the brush that you selected to open the Custom Brush menu.



NOTE: You cannot use the Undo command when drawing or updating a custom brush.

To draw a brush shape:

1. Erase the existing brush shape by painting over the editing panel if you want to create a new shape.
2. Draw the new shape.

If you want to invert the brush shape, click the Invert button in the Custom Brush menu.

If you want to blur the brush shape, click the Blur button in the Custom Brush menu. Each time you click the Blur button, a 3x3 filter is applied to the brush image.

Selecting an Area of the Image

You can select an area of the image to use as the custom brush. The brush created from the selected area will be a square monochrome brush.

To grab an area of the image:

1. Click the Grab button in the Custom Brush menu.
2. Click on the image and drag across the screen to define the selection box.

To select a square area, hold down the **P** key as you sweep out the selection box.

The selected area appears as a monochrome image in the editing panel. If the area that you select is not square, then the selected area is resized to fit in the square editing panel.

Updating an Existing Brush

To update an existing brush, click the Update button. The changes are applied to the brush icon that you selected to open the Custom Brush menu.

Creating a New Brush

To create a new brush, click the New button. This creates an icon for the new brush shape, and adds it to the Brushes window.

Saving the Custom Brush

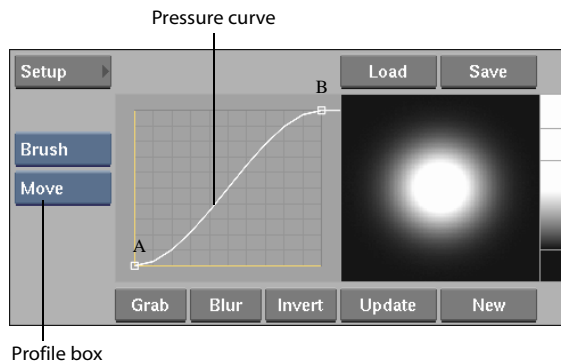
Custom brushes are not saved automatically when you exit Paint. To save the updated or new brush for use in another work session, click the Save button. This opens the Brush setups directory. Use the keyboard to enter the name of the brush, and click the Enter button. The new brush is saved in the Brush setups directory.

Loading a Custom Brush

Click the Load button to load a custom brush from the Brush setups directory.

Customizing the Pressure Profile

Adjust the pressure profile to change the amount of pressure needed to paint with a given brush. The slope of the curve indicates how quickly paint is applied as you press on the tablet. If the curve is steep, a small amount of pressure applies full paint to the canvas. If the curve is soft and rounded, you must press harder and longer to apply full paint.



You use all the same options in the Profile box to change and add points to the pressure curve. For more information, see “Changing the Curve” on page 1162. Click Load to load a Pressure profile. Click Save to save a custom pressure profile.

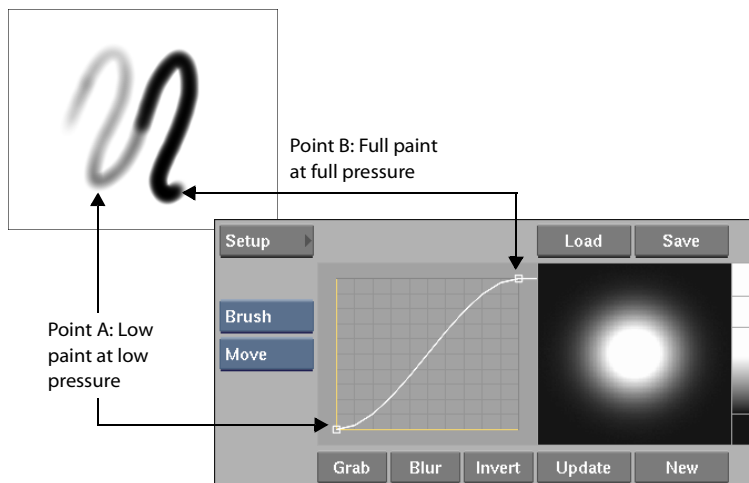
To edit the pressure profile:

1. Set the brush attribute mode to Pressure.
2. Double-click a brush icon in the Brushes window to display the Custom Brush menu.
3. Click the Pressure button to display the pressure profile.

4. Select the Move option in the Edit Mode box.
5. Press the left point (A) and drag it to its new position.
This changes the amount of paint the brush applies when you press lightly on the tablet.
6. Press the right point (B) and drag it to its new position. This changes the amount of paint the brush applies when you press down firmly on the tablet.
7. Click Update to update the pressure profile for the brush.
The pressure settings will be used whenever you paint with the brush during the current work session.

Default Pressure Profile

Pressure is mapped along the horizontal axis (X-axis) of the curve, and the amount of paint applied is mapped along the vertical axis (Y-axis). If you are using the default S-shaped curve and you hardly press on the stylus, very little paint is applied. As you press harder, more paint is applied.



Maximum Pressure Curve

In the following figure, point A has been moved to the maximum position on the Y-axis. This curve results in full paint applied all the time, no matter how much or how little pressure you use.

Soft Airbrush

In the figure below, Point B has been moved to a point midway on the vertical axis. This curve gives you a soft airbrush, even at full pressure.

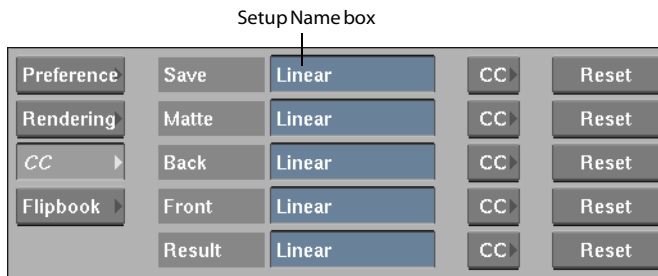
Using Colour Correction Setups

Use the Colour Correction Setup menu to apply colour correction setups to the clips loaded into Paint. You can bring the clip into the Colour Corrector or you can load a setup from the Colour Correction setups directory into Paint.

To load a clip into the Colour Corrector:

1. Click Setup.
2. Click CC.

The Colour Correction menu appears.



3. Click CC next to the clip you want to load into the Colour Corrector.

The Colour Corrector appears.

4. Use the Colour Corrector as described in Chapter 26, “Colour Corrector.”
5. Save the Setup to the Colour Correction setups directory.
6. Click Exit.

The name of the Colour Correction setup appears in the Setup Name field beside the clip label for the selected clip.

To load a setup directly from the Colour Correction setups directory:

1. Click the Setup Name field for the clip you want to colour correct.

The file browser appears, displaying the contents of the Colour Correction setups directory.

2. Select the Colour Correction setup you want to use.

To reset the colour correction for a clip:

1. Click Reset beside the clip you want to reset.
2. Click the Confirm button.

Move images in and out of your framestore without tying up the application. Use the framestore utilities to transfer images between your framestore and local or remote file systems from the command line.

Summary

In this chapter, you learn about:

- “Opening an IRIX shell” on page 1168
- “Requirements” on page 1168
- “Using the Utilities” on page 1169
- “Parameters” on page 1169
- “list_framestore” on page 1171
- “to_framestore and from_framestore” on page 1173
- “Restrictions” on page 1174
- “imcopy” on page 1175

About Utilities

You use command line utilities to move clips between image files stored on local or remote file systems and the clip libraries stored on your framestore. You can also use the *from_framestore* and *to_framestore* utilities to copy image files and clips:

- The *from_framestore* utility copies a clip stored in a clip library on the framestore of the local machine to a sequence of image files in a file system on a local or remote system.
- The *to_framestore* utility copies image files from a local or remote file system to the framestore of the local machine.
- The *list_framestore* utility lists the contents of the specified framestore partition or clip library.
- The *imcopy* utility can be used to apply certain operations to image files such as cropping, resizing, or format conversion. It can also import and export images in a more flexible way than *from_framestore* and *to_framestore*, but requires that the Network File System (NFS) be installed and configured in order to access images on remote systems.

These command line utilities can be used in the background without affecting the normal operation of **flame**.

Opening an IRIX shell

All commands described in this chapter are invoked from an IRIX shell.

To open an IRIX shell:

1. If **flame** is already running, press **F5** (or the key you have mapped to the Lower Window function) to lower the **flame** window and show the windows hidden behind it.
2. With the left mouse button, press and hold the Tools option in the Toolchest window, and choose the Shell option.
A shell window is displayed.
3. When you are finished using your IRIX shell, type **exit** to close it.
4. Position the cursor above an exposed section of the **flame** window and press **F6** (or the key you mapped to the Raise Window function) to bring the **flame** window back on top.

Requirements

To use the *to_framestore* and *from_framestore* utilities, you must be able to access the remote machine without having to reenter a password. Otherwise, the message `Permission denied` will appear when you try to invoke either utility. To access a remote machine, you can use a *.rhosts* file.

To use an *.rhosts* file:

1. Log in to the account from which you will be running **flame**.
2. Open an IRIX shell (see “Opening an IRIX shell” on page 1168).
3. Type **cd ~** to change the directory to your home directory.
4. Type **ls -a** to list all files (including hidden files) in your home directory.
5. Use any text editor to create a new *.hosts* file or edit an existing one. If it is not already present, add the line:

```
+ <user_name>
```

where *<user_name>* is the user name you are currently logged in as.
6. Save the file if you have modified it and exit from the text editor.
7. Type **ls -l .rhosts** to view the file permissions of the *.rhosts* file. You must be the only user that can write to the *.rhosts* file.
8. Save the *.rhosts* file.

Using the Utilities

To use any utility, open an IRIX shell (see “Opening an IRIX shell” on page 1168) and enter the appropriate command using the format specified on the following pages.

Parameters

The following sections describe the parameters required in the *list_framestore*, *from_framestore*, *to_framestore* and *imcopy* utilities.

Clip Directory Pathname

The pathname of the clip directory (the directory that contains references to the framestore images) must be entered as **/usr/discreet/clip** in the *list_framestore*, *from_framestore*, *to_framestore* and *imcopy* utilities.

Framestore and Partition Names

In the *list_framestore* utility, enter the name of the framestore and the name of the partition. To find this information, open the project configuration file corresponding to the project from which you want to access images and look for the `Framestore Definition` heading. For example, in this line:

```
FrameStore stonefs, "NTSC"
```

`stonefs` is the name of the framestore, and `NTSC` is the name of the partition.

NOTE: By default, the configuration file is located in the `/usr/discreet/project/flare_7.0/PROJECTNAME/cfg/PROJECTNAME.cfg` directory.

Clip Name Format

The following clip naming formats are used to specify the location of clips in the *list_framestore*, *to_framestore*, *from_framestore* and *imcopy* utilities.

Absolute Paths

An initial `/` indicates that the path for the clip is anchored at the root of the library. The following three formats can be used to find or list clips:

```
/Desktop1/Reel1/Clip1
```

where Clip1 is on the reel named Reel1 on Desktop1.

```
/Reel2/Clip2
```

where Clip2 is on the reel named Reel2.

```
/Clip3
```

where Clip3 is located at the root of the library.

Relative Paths

If you do not know the absolute path, you can omit the initial `/` and **flame** will search the library for the reel and/or clip. The following two formats can be used to find clips if you do not know the absolute path:

Reel3/Clip3

where Clip3 is on Reel3, which is located inside the library.

Clip4

where Clip4 is located inside the library.

Untitled Desktops, Reels, and Clips

flame uses one or more `/` to refer to an unnamed clip, desktops or reel.

`/`

is an unnamed clip in the library.

`//`

is an unnamed clip in an unnamed reel in the library.

`///`

is an unnamed clip in an unnamed reel in an unnamed desktop.

`//Reel15/Clip5`

Clip5 is on Reel5 in an unnamed desktop.

`/Desktop6//Clip6`

Clip6 is on an unnamed reel on Desktop6.

`/Desktop7/Reel7/`

is an unnamed clip on Reel7 which is located on Desktop7.

The File Template

In the *from_framestore* utility, you can use a file template to specify a pattern from which the name of each destination image file will be derived. Similarly, in the *to_framestore* utility, you can use a template to specify a sequence of source image files. In the *imcopy* utility, both the source and destination can be image sequences specified by a file template.

The *from_framestore* utility (as well as *imcopy* when used to export from a framestore clip) stores each frame of a clip as a separate image file in a local or remote file system.

Image files are numbered sequentially so that the order of frames in the clip can be maintained in transfers. Specify the file name and the numbering format for the image files with the following file template:

<clip name>%04d.sgi

The `4` instructs the utility to use four digits when numbering the files; the `0` instructs it to use leading zeros before the file number.

For example, if you use the file template *example%04d.sgi* in the *from_framestore* utility, the following image files are stored in the specified destination directory:

example0001.sgi

example0002.sgi,

example0003.sgi, etc.

The *to_framestore* utility looks for matching file names in the specified source directory.

list_framestore

The *list_framestore* utility lists the contents of each clip library that you create in a framestore partition. You can use the *list_framestore* utility to list the contents of the framestore before using the *to_framestore*, *from_framestore* or *imcopy* utility.

The listing shows the absolute path to the clip (see “Absolute Paths” on page 1169), the clip name, its creation date and time, and its description. For a source clip, the description is the timecode and the number of frames.

Syntax

```
list_framestore <clip directory pathname>/<framestore name>/
<partition name>/<library name>
```

Where:

<clip directory
pathname>

Is:

The clip directory pathname (*usr/discreet/clip*).

<framestore name>

The name of the framestore, for example: *xl/v0, stonefs*.

<partition name>

The name of the partition, for example: *NTSC*.

<library name>

The name of a library (optional). If this is omitted, all libraries are listed.

Example

```
list_framestore /usr/discreet/clip/stonefs/NTSC
```

This command lists the contents of the framestore partition WARSHAW. For example:

LIBRARY /usr/discreet/clip/stonefs/WARSHAW/Default.library

/MARTIN	99/05/03 09:53:52	01:07:50:07 - 111
/Chasser	99/05/03 09:56:57	01:07:53:15 - 1
/52min_SUBTITLED_OUTPUT/	99/05/03 17:06:42	10;20;07;03 - 197
/52min_SUBTITLED_OUTPUT/	99/05/03 17:07:05	10;20;29;09 - 149
/47min_SUB_OUTPUT/	99/05/03 18:31:48	10;25;01;16 - 197
/47min_SUB_OUTPUT/	99/05/03 18:32:33	10;25;23;22 - 149

LIBRARY /usr/discreet/clip/stonefs/WARSHAW/WARSHAW_ON_THE_MAIN.library

/MASTER_4mail999/47MIN_textless	99/05/04 07:32:40	09;58;30;00 - 110590 A
/MASTER_4mail999/52MIN_textless	99/05/04 07:35:39	09;58;30;00 - 98004 A
/MASTER_4mail999/39MIN_textless	99/05/04 07:34:53	09;58;30;00 - 76426 A
/MASTER_4mail999/39MIN_subtitled	99/03/25 11:16:43	09;58;30;00 - 76426 A
/MASTER_4mail999/47MIN_subtitled	99/03/25 11:20:50	09;58;30;00 - 110590 A
/MASTER_4mail999/52MIN_subtitled	99/03/25 11:19:04	09;58;30;00 - 98004 A
/FINAL_AUDIO/52min_MIX	99/05/03 07:52:33	09;58;30;00 - 98004 A
/FINAL_AUDIO/47min_MIX	99/05/03 08:57:02	09;58;30;00 - 110590 A
/FINAL_AUDIO/39min_MIX	99/05/03 09:42:57	09;58;30;00 - 76426 A
/EDL/warshaw52min_corr	99/03/25 09:24:40	10;00;00;00 - 93507 A
/EDL/warshaw47_45min_corr	99/03/25 09:24:21	10;00;00;00 - 87362 A
/EDL/warshaw39min_corr	99/03/25 09:24:05	10;00;00;00 - 72228 A
/FLAME_GRAPHICS/Warshaw_HeroCART	99/03/16 16:56:35	00:00:00:00 - 113
/FLAME_GRAPHICS/Warshaw_HeroFrench	99/03/16 16:57:48	00:00:00:00 - 113
/FLAME_GRAPHICS/Warshaw_Hero*	99/03/16 16:58:53	00:00:00:00 - 113
/FLAME_GRAPHICS/Warshaw_HeroREV	99/03/16 17:02:01	00:00:00:00 - 113
/FLAME_GRAPHICS/painted_WARP	99/02/20 16:30:02	00:00:00:00 - 149
/FLAME_GRAPHICS/FilmBy_CART	99/03/16 17:10:29	00:00:00:00 - 105
/LOGOS/LOGO	99/03/15 09:49:52	00:00:00:00 - 1
/LOGOS/720fr-2	99/03/15 09:34:15	00:00:00:00 - 1
/LOGOS/720fr-1	99/03/15 09:34:15	00:00:00:00 - 1
/LOGOS/720eng-2	99/03/15 09:34:15	00:00:00:00 - 1
/ROLL_MATERIAL/FINAL47min_roll	99/04/07 13:21:50	00:00:00:00 - 958/
/ROLL_MATERIAL/FINAL39min_roll	99/04/07 14:18:51	00:00:00:00 - 1422

to_framestore and from_framestore

This section describes the `to_framestore` and `from_framestore` utilities.

to_framestore

The `to_framestore` utility copies image files from the specified directory to the framestore.

Once the transfer is complete, the clip is available in the clip library. You can use the Load Clip command to load the clip to the work reels. For more information, see Chapter 12, “Clip Libraries.”

from_framestore

The `from_framestore` utility copies clip information files from the framestore on the local machine and saves them as image files in the specified destination directory.

If you transfer two or more clips with the same name, the most recent clip is used unless you specify otherwise.

Before using the `from_framestore` utility, make sure the clip has been saved in the clip library. For information on saving a clip, see Chapter 12, “Clip Libraries.”

Syntax

```
<command name> <clip library pathname>@<clip name> <hostname>:/
<directory pathname>/<file template> [<starting frame
number> [<ending frame number>]
```

Where:	Is:
<command name>	either from_framestore or to_framestore
<clip library pathname>	the directory pathname of the clip library. This pathname is displayed in the LIBRARY line of the list_framestore listing. For example: <code>/usr/discreet/clip/stonefs/NTSC/paint.library</code>
<clip name>	the name of the clip as shown in the first column of the list_framestore output.
<hostname>	the hostname of the local or remote machine where the destination file system resides
<directory pathname>	the path of the destination directory on the local or remote machine
<file template>	the template used to name and number the image files. For example: <code>example%04d.sgi</code>
<starting frame number>	the number of the first frame in the clip that you want to copy (optional)
<ending frame number>	the number of the last frame in the clip that you want to copy (optional)

NOTE: If the message `remuser too long` appears when you initiate `from_framestore` or `to_framestore`, the current user name exceeds 8 characters. In this case, use the **su** (substitute user) command to switch to another user's identity with a user name of 8 characters or less. Note that you must enter the corresponding user password.

Example

```
from_framestore /usr/discreet/clip/stonefs/NTSC/paint@jaune  
sgi2:/usr/tmp/jaune%04d.sgi
```

This command retrieves the clip information for all frames in the clip *jaune* from the clip library */usr/discreet/clip/stonefs/NTSC/paint.library*, and creates the image files *jaune0001.sgi*, *jaune0002.sgi*, and so on in the directory */usr/tmp* on the remote machine *sgi2*.

Using 12-bit partitions

You can use the `from_framestore` and `to_framestore` utilities with 12-bit partitions. Use the **-l** command to import or export 16-bit SGI images using the 12 least significant bits. For additional information, see “Import Image Options” on page 327.

Example

```
from_framestore -l /usr/inferno/clip/swr00/Film_Res/  
<libraryname@clipname>
```

Restrictions

The following restrictions apply to the use of the `from_framestore` and `to_framestore` utilities:

- These utilities work with the raw framestore only. Your system must include a disk array with a raw disk store.
- Only RGB non-RLE encoded SGI images can be used. If you have SGI images that use RLE encoding, you can use the */usr/sbin/verbatim* utility (which is installed as part of the *ee.sw.imagetools* IRIX subsystem) to convert these to non-RLE encoding.
- The image files to be transferred must be the same size as the framestore images.
- The `from_framestore` or `to_framestore` utility must be invoked from the machine on which the framestore resides.

If you specify the name of an existing clip as the destination clip name, `to_framestore` will create a new clip with the same name. You cannot overwrite or replace an existing framestore clip. For more information on the file template, see “The File Template” on page 1170.

imcopy

In addition to the *to_framestore* and *from_framestore* utilities, *imcopy* is another image transfer/conversion utility installed with **flame** as part of the *DLtools.libio* package. *imcopy* is located with the other utilities in */usr/discreet/io/bin* and can be used to import and export images to and from the framestore. The command **versions DLtools** will verify that this package has been installed on your system.

The *from_framestore* and *to_framestore* utilities use the *rsh* protocol when accessing images on remote machines, which requires little additional configuration (see “Requirements” on page 1168). *imcopy* requires that remote file systems be accessible via NFS (the Network File System). However, these utilities must conform to the restrictions outlined in “Restrictions” on page 1174.

imcopy is a command-line utility that lets you extract either single images or image sequences from the framestore and convert them to any image format supported by **flame**. It can also import single images or image sequences into the framestore. The *from_framestore* and *to_framestore* utilities can only understand uncompressed SGI image files, however *imcopy* understands all image file formats supported by **flame**. *imcopy* can also apply croppings and scalings, and select output compression. This utility must have direct access to the image files, whether on a local file system or on a remote host accessed via NFS.

NOTE: *imcopy* cannot access a remote framestore unless its directory has been mounted with NFS. Otherwise, *imcopy* must be working with the framestore connected to the system on which it has been installed.

With the *imcopy* utility, you can:

- Convert image formats supported by **flame**. *imcopy* supports 8 bits per component or greater for image formats such as Cineon, SGI, Wavefront, TIFF, Tdi/Maya, and DPX.
- Apply a LUT (Look-Up Table) for linear-to-log conversion.
- Crop and resize your image upon import/conversion.
- Extract the alpha channel from an RGBA image, and output it as a separate file.

To start imcopy:

1. Make sure that */usr/discreet/io/bin* is in your path.

2. Type:

```
imcopy <options> <infile> <outfile> [start index] [stop index]
```

Syntax:

```
imcopy [-f format] [-c chan] [-m compress] [-o x,y]
[-s width,height] [-d width,height] [-l inlut] [-w outlut]
[-v] [-b r,g,b,a] [-i nbit] [-e nbit] [-k croptype] [-q zoomqual]
[-j jpegfactor] [-t] [-a alphasource] [-x alphadest] [-p x,y]
```

```
infile outfile [start index] [stop index] [step factor]
```

Example:

```
imcopy -k fitall -d 2048,1556 -q quality -f Cineon
/usr/discreet/clip/stonefs/NTSC/video@clip_name
/usr/tmp/clip_name.%04d.cineon
```

NOTE: %04d means use 4 digits padded with zeroes; %d means do not use any padding for the numeric part of the name.

To view the available **imcopy** options:

1. Make sure that */usr/discreet/io/bin* is in your path.
2. Type **imcopy**

Options:

Option:	Function:	Default:
-f	Output file format	Same as source image
-c	Output channel format (BW, RGB, or RGBA)	Same as source image
-m	Output image compression	Same as source image
-o	(x,y) source origin in <i>infile</i>	0,0
-s	Source width and height in <i>infile</i>	Whole file
-d	Destination image width and height	same as source image
-l	Map input pixels through lookup tables in the <i>inlut</i> file	No mapping
-w	Map output pixels through lookup tables in <i>outlut</i> file	No mapping
-v	Verbose	Non-verbose
-i	Bits per pixel (bit-depth) in the internal buffer (8 or 12)	8
-e	Bits per pixel in the output image	8
-b	R,G,B, or a background colour between 0.0 and 1.0	0.0, 0.0, 0.0, 0.0
-k	Cropping type (crop, fit, fitall or fitboth)	Crop
-q	Zoom quality (coarse, medium or quality)	Quality
-j	JPEG compression factor (0-100)	75
-t	Use 12 LSBs of 16 bit SGI files	12 MSBs

Option:	Function:	Default:
-f	Output file format	Same as source image
-a	Combine the alpha (matte) file and RGB file (matte file name)	None
-x	"invert" or "noinvert", extracts the alpha channel to <filename>_alpha	None

Supported formats (-f option):

Image Format:	Default Extension:
Alias	als
Cineon	(none)
DPX	(none)
Pixar	picio
Pict	pict
JPEG	jpeg
SGI	sgi
Softimage	pic
Targa	tga
TdiMaya	iff
Tiff	tif
Wavefront	rla

Converting File Formats

To convert an image from one format to another using *imcopy*, you must specify which file to convert, where the file is located, the format it should be converted to, and where to store the file on output.

Syntax:

```
imcopy -f <output_format> <infile> <outfile> [start index]
[stop index]
```

Example:

```
imcopy -f tiff /usr/tmp/SGI/source%04d.sgi /usr/tmp/TIFF/
dest%04d.tif 1 100
```

This operation converts the SGI images into TIFF images.

Input:	Output:
<code>/usr/tmp/SGI/ source0001.sgi</code>	<code>/usr/tmp/TIFF/source/dest0001.tif</code>
<code>/usr/tmp/SGI/ source0002.sgi</code>	<code>/usr/tmp/TIFF/source/dest0002.tif</code>
<code>/usr/tmp/SGI/ source0100.sgi</code>	<code>/usr/tmp/TIFF/source/dest0100.tif</code>

Converting 10-bit Cineon Images

Cineon images contain 10-bit log data. If you import Cineon images directly into **flame**, you may get undesirable results. You need to create a Look-Up Table (LUT) to convert from 10-bit log to 8-bit or 12-bit linear, depending on the selected partition. In the **flame** Import Image menu, select the Cineon format. Click the Create LUT button that appears to enter the LUT builder module, and create a LUT to convert from 10-bit log to 8-bit or 12-bit linear. For more information on LUTs, see “Using a Lookup Table” on page 330.

The *imcopy* utility can be used to import Cineon files directly into the framestore from the command line, and can utilize the same LUTs that you create in **flame**.

Extracting an Alpha Channel from an RGBA Image

You can use the **-x** option to extract the alpha channel from your RGBA image. The RGBA infile will be split into two RGB outfiles; the *destination_alpha_filename* file will contain only the alpha channel of the *inRGBA_file*, and the *outRGB_file* will contain only the RGB components of the *inRGBA_file*.

Syntax :

```
imcopy -x <destination_alpha_filename> <inRGBA_file>
<outRGB_file>
```

Example:

```
imcopy -x /usr/discreet/clip/stonefs/NTSC/MyLib@myclip_alpha
/usr/tmp/SGI/source%04d.sgi /usr/discreet/clip/stonefs/NTSC/
MyLib@myclip 1 100
```

This example takes the SGI RGBA images in the sequence and creates two clips in the *MyLib* library on the *NTSC* partition of the framestore *stonefs*.

Input:	Output:
<code>/usr/tmp/SGI/ source0001.sgi</code>	RGB in frame 1 of myclip, Alpha in frame 1 of myclip_alpha

Input:

```
/usr/tmp/SGI/
source0002.sgi
```

Output:

RGB in frame 2 of *myclip*, Alpha in frame 2 of *myclip_alpha*

```
/usr/tmp/SGI/
source0100.sgi
```

RGB in frame 100 of *myclip*, Alpha in frame 100 of *myclip_alpha*

On output, the clip *myclip* contains the RGB image, and the clip *myclip_alpha* contains the alpha channel of the original image.

Integrating an Alpha Channel into an RGB Image

You can use the **-a** option to combine an alpha (matte) infile and an RGB infile to produce an RGBA outfile.

NOTE: When reading a matte from the Inferno RGB framestore, imcopy will only take information from the image's red channel.

Syntax:

```
imcopy -a <in_mattefile> <in_RGBfile> <out_RGBAfile>
```

Example:

```
imcopy -a /usr/discreet/clip/stonefs/NTSC/MyLib@myclip_alpha
/usr/discreet/clip/stonefs/NTSC/MyLib@myclip /usr/tmp/SGI/
dest%04d.sgi 1 100
```

This example takes the RGB data from *myclip* in the *MyLib* clip library in partition *NTSC* on framestore *stonefs*, combines it with the matte clip *myclip_alpha*, and generates SGI RGBA for the first 100 frames.

Input:

```
RGB from frame 1 of myclip and Alpha from frame 1
of myclip_alpha
```

Output:

```
/usr/tmp/SGI/
dest0001.sgi
```

```
RGB from frame 2 of myclip and Alpha from
frame 2 of myclip_alpha
```

```
/usr/tmp/SGI/
dest0002.sgi
```

```
RGB from frame 100 of myclip and Alpha
from frame 100 of myclip_alpha
```

```
/usr/tmp/SGI/
dest0100.sgi
```

Cropping an Image

If you want to import a section of your source image only, you can specify the origin of this section with the **-o** (x position, y position) option. You can specify the size of this section with the **-s** (width, height) option.

The size of the internal buffer is specified with the **-d** (width, height) option. By default, the size of the internal buffer is the same as the width and height of the source image. If the buffer

size differs from the source image, resize the selected rectangle from the source image into the internal buffer size using the **-k** (cropping type) option. The cropping options are:

Option:	Function:
-k crop	Crops the image. -b r, g, b, or a can be used to specify the color for the cropped part of the image or set the default to black.
-k fit	Scales the longest edge to fit the frame size. Unused parts of the frame are filled with black.
-k fitall	Scale the shortest edge to fit the frame size and crop the longest edge. The frame will be filled and part of the original image will be missing.
-k fitboth	Stretch or squeeze the image to fill the new resolution. This option scales the X and Y dimensions of the image non-proportionality to make it fit in the current frame. This changes the aspect ratio, which results in a distorted image.

The **-q** option can be used to specify the quality of resizing: coarse, medium, quality or bicubic.

NOTE: If you are importing an image to the framestore, the specified width and height parameters of the framestore are used to define the destination size of your image.

Adjusting Bit Depths

By default, the internal buffer has a depth of 8 bits. If your source image has a depth greater than 8-bit, you can specify an internal buffer depth of 12 bits using the **-i 12** option. This changes the buffer depth from 8-bit to 12-bit.

Once the desired area of the source image has been cropped and resized and the bit depth has been converted into the internal buffer, the image is written to the destination. If the destination is a clip on a framestore, the output size and bit depth is adjusted accordingly. If the destination is an image file, the image file format will be the same as the input format. You can use the **-f** (image format) option to specify a different image format. By default, *imcopy* outputs images with 8-bit depth. Use the **-e** (bit depth) option to override the default setting.

Source Image Bit-Depth:	Option Parameter:
16-bit (TIFF, SGI, Wavefront, Tdi/Maya)	-e 16
10-bit (Cineon, DPX)	-e 10

NOTE: By default, if you read an image file with more than 8 bits per component into an 8-bit internal buffer, *imcopy* will only preserve the 8 most significant bits. To avoid this, specify an input LUT with the **-l** option to convert the source image bit-depth to the internal buffer bit-depth. Using the default 8 bit internal buffer depth (-i 8), you will need a 10log to 8 LUT. Using a 12-bit internal buffer depth (-i 12), you will need a 10log to 12 LUT. For more information on LUTs, see "Using a Lookup Table" on page 330.

glossary

Glossary

This glossary contains terms that are found within this user's guide, as well as industry terms, and product terms that apply to this and other Discreet products.

Note: Words that appear in boldface within a definition are also defined by entries within this glossary.

1080p/24	1080p/24 is an HDTV format. Commonly described as the universal mastering format, it is 1920 pixels wide by 1080 pixels in height, progressively scanned at 24 frames per second.
2K	A film image scanned into a computer file at a resolution of approximately 2000 (usually 2048) horizontal pixels.
3:2 pulldown	3:2 pulldown is a standardized procedure to generate 60 fields of video from 24 frames of film.
4K	A film image scanned into a computer file at a resolution of approximately 4000 (usually 4096) horizontal pixels. 4K is considered to be a full-resolution scan of 35mm film.
4:2:2	The sampling ratio used in the ITU-R BT.601-4 digital video signal. For every 4 samples of luma, there are 2 samples each of R-Y (Red minus Luma) and B-Y (Blue minus Luma).
4:4:4	A sampling ratio that has equal amounts of both the luma and the chroma channels. Usually used to describe an RGB signal.
additive mix	A mixing process that adds the colour value of a pixel in one source clip to the value of a corresponding pixel in a second source clip, and assigns the resulting value to the corresponding pixel in the output clip. <i>See also</i> non-additive mix <i>and</i> inverse non-additive mix.
AIFF	Audio Interchange File Format. This is the format for both compressed and uncompressed audio data.
aliasing	Defects in the picture caused by a sampling frequency too low, or by poor filtering. <i>See also</i> anti-aliasing.

alpha channel	1. A TV signal that defines a portion of an image or clip to be keyed, or overlaid with a second image. 2. A matte or mask that defines a portion of an image or clip. 3. A raster channel defined by a paint or brush stroke.
ambient light	Natural, or surrounding light in a clip.
analogue	Information that is represented electronically as a continuously varying electronic signal.
anamorphic	Distortion in viewing of images or geometry; related to the difference between computer monitor screen aspect ratio, in which pixels are square, and broadcast, projected, or frame aspect ratio, in which the height differs from the width of image pixels.
animation curve	A curve depicting the interpolation between keyframes; can be viewed and edited within the Channel Editor.
anti-aliasing	Smoothing, reducing, or removing jagged edges along the lines and curves in text, images, or geometry. <i>See also</i> aliasing.
ASCII	American Standard Code for Information Interchange. Unformatted text characters that are interchangeable across platforms and applications. Also referred to as “plain text”.
aspect ratio	The ratio of picture height to width. Theatre screens generally have an aspect ratio of 1.85 to 1; widescreen television (16 x 9) of 1.77 to 1; and normal television (4 x 3) or 1.33 to 1.
ATSC	The Advanced Television Systems Committee, established in 1982, is an international organization developing voluntary standards for emerging technologies in the broadcast industry. In 1996, the Federal Communications Committee (FCC) of the United States adopted the principal elements of the ATSC DTV Standards.
audio subframe	There are 100 subframes of audio for every frame of video.
axis	The element of an object or layer that can be manipulated to determine the object's two or three dimensional space, position, and movement.
bandwidth	The amount of information that can be transferred in a given time. The greater the bandwidth, the greater the image detail.
batch	An automated process in which clips are processed according to a schematic set up in the Batch Processing module.

bezier	<p>1. A curve that connects the vertices of a polygon; each vertex has two tangents, or handles, which you can use to adjust the slope of the adjacent curve or side of a polygon.</p> <p>2. One of the Polygon Mode options.</p>
bicubic surface	A surface with four control handles that can be added to a layer for creating non-linear effects. <i>See also</i> extended bicubics.
bilinear surface	A surface with four control handles that can be added to a layer and used for four-point tracking.
bit	One binary digit. An 8-bit byte can define 256 brightness or colour values.
bitmap	A pixel-by-pixel digital translation of an image. Bitmap images are also referred to as Raster images.
bit rate	The amount of data transferred in a given amount of time, usually defined in Mega (million) bits per second. Bit rate is one measurement of compression used on a video signal.
brightness	<p>1. A visual sensation that an area of an image seems to emit more or less light. Brightness is a perceptual quantity, and can have no true objective measure.</p> <p>2. In NTSC and PAL, brightness information at a particular instance in a picture is conveyed by instantaneous DC level of active video.</p>
broadcast monitor	Television set without receiving circuitry, wired directly to a VTR or other output device.
Bspline	<p>1. A smooth curve that passes on the inner side of the vertices of a polygon to connect the vertices to interpolate or draw the polygon.</p> <p>2. One of the Polygon Mode options.</p> <p>3. A curve used to define a motion path. <i>See also</i> spline.</p>
byte	A sequence of bits. In general, the amount of memory needed for one character (for example, the letter A) of a specified size, usually 8 or 16 bits.
channel	<p>1. A parameter whose values can be adjusted in several keyframes, and animated over a sequence of frames.</p> <p>2. Image colour information. For example, an RGB image has three channels: red, green, and blue.</p>
Channel Editor	The tool used to set keyframes and modify the animation curves of various channels.
channel hierarchy	A set of animation parameters arranged and displayed in a logical group within the Channel Editor. A group, or upper-level, channel is called a folder. For example, the Camera folder contains channels for camera settings, such as position, interest, and focal length.

chroma	The element of the video signal that contains hue and saturation information.
clip	A sequence of images or frames loaded from an external device such as a VTR, stored in digital format, and displayed in the clip library and on the desktop. You can edit, process, and record clips to external devices.
CMYK	A colour model which combines cyan, magenta, yellow, and black to represent the colour spectrum.
colour correction	Changing, adjusting, substituting, or suppressing colour or colour elements in an image.
colour palette	A tool used to store and select colours in Paint.
colour picker	A tool used to plot colours in an image.
component video	A video signal in which the luma and chroma signals are kept separate. Requires a greater bandwidth, but yields a higher-quality picture.
composite	A combination of image elements from two clips used to produce an output image. For example, you can composite a clip that combines background elements from one clip, and foreground elements from another clip.
compression	A method of removing information from a video signal to reduce the image volume. Compression can cause data to be lost.
compression rate	The ratio of the amount of data in the original video compared to the amount of data in the compressed video. The greater the compression, the higher the ratio.
configuration file	Contains a list of the device parameters, resource directory pathnames, and image file extensions that are required to run the software and all external devices properly.
constant	An option for the interpolation and/or extrapolation of an animation curve that produces a square or stepped curve.
contrast	Gradation between light and dark areas of an image. “Contrast” is the general term for the property called “gamma”.
control points	Small square handles displayed when you draw or edit geometric shapes that can be manipulated to change the shape.
crop box	A boundary box that, superimposed on an image, restricts colour corrections, key setups, or other processing options to the area of the clip within the box.
crossfade	A dissolve transition that occurs between a fade-out clip and a fade-in clip over a specified number of frames, and has a specified midpoint.

cue mark	Marks made on a clip to indicate frames of interest.
cut	To remove frames from a clip, or to split a clip into separate clips.
cycles	An option for the extrapolation curve that produces cycles within the curve, based on the values of the first and last keyframes in the curve.
D1	A digital VTR format that conforms to ITU Rec. 601 (ITU-R BT.601-4) 4:2:2 standards, and uses 19-mm tape.
DDR	Digital Disk Recorder. A machine that converts analogue video signals into digital information.
decibel	A unit of measure applied to both sound and electrical signals, based on a logarithmic scale. Also referred to as “db”.
deinterlace	A command in the Image Options menu that separates fields 1 and 2 in a source clip, producing a new clip twice as long as the original. <i>See also</i> interlacing.
diffuse light	Widely spread lighting that reflects equally in all directions, producing a matte (flat), reflection on an object. The reflection intensity depends on the position of the light source relative to the surface of the object.
digital	A system whereby a continuously variable (analogue) signal is broken down and encoded into discrete binary bits that represent a mathematical model of the original signal.
digitize	To convert an analogue signal into digital form for storage and processing purposes.
directory	A logical or physical portion of a hard disk drive where the operating system stores files. IRIX uses a hierarchical tree structure to organize directories. The root directory, identified by the forward slash, contains all the other directories on the disk. When you install flame , the installation program creates a directory on the hard disk for the product, such as <i>/usr/discreet</i> . In this example, <i>/usr/discreet</i> is a subdirectory of the <i>/usr</i> directory, and <i>usr</i> is a subdirectory of the root (<i>/</i>) directory.
Discreet Filesystem (STONEFS)	Discreet framestore management software for creating dynamic soft partitions of different resolutions on the Discreet Storage unit.
Discreet Network (WIRE)	Discreet networking utility for transferring clips from a volume on a remote framestore to a volume on a local framestore.
Discreet Storage (STONE)	Discreet high-performance disk array that provides real-time video playback, resolution-independence, and protection against loss of stored images.

disk array	Combination of multiple hard drive assemblies into a single hard volume to deliver optimal performance and high-capacity storage.
displacement mapping	The adding of a 3D effect to a 2D image. Also called surface displacement.
displacement source	The source image used to generate the displacement of pixels in another image.
dissolve	A transition effect in which the outgoing image blends into the incoming image, creating a blend from one to the other. The effect is created optically by running two clips concurrently through two projectors.
DLT	A high-capacity data tape format.
down-converting	Down-conversion is the process of transferring images of a higher resolution to images of a lower resolution. Because the initial resolution is high, the image quality may be superior than images acquired at this target resolution.
drop shadow	The silhouette of an image based on the matte of the image.
DVE	Digital Video Effect.
edit sequence	A timeline reference to a section of a clip where it has been fused with another.
EDL	Edit Decision List. A list of commands used to describe a series of edits. An EDL uses SMPTE timecode to interchange information between offline and online editing systems.
Ethernet	A form of local area network (LAN) used to interconnect computers and peripheral devices. Ethernet is a standardized system; many manufacturers supply hardware and software for Ethernet networks.
exabyte	An 8mm data tape format, popular for storing graphics due to its low cost and high capacity (8 - 40 gigabytes).
extended bicubic	A surface with four control handles that can be added to a layer for creating advanced non-linear effects, such as deformations.
extrapolation	A mode that defines the shape of an animation curve before the first keyframe, and after the last keyframe. Only apparent if there are frames which precede and follow the keyframes.
fade	A transition effect in a clip in which it disappears into blackness (fade-out) or appears from black (fade-in).
field	One half of a complete video image (frame) containing all of either the odd or even scanning lines. <i>See also</i> frame.

field dominance	Setting in the Preferences menu that determines whether field 1 or 2 of a frame is dominant (i.e., is displayed first). Commands that render in “field” mode use the specified field dominance.
fill	1. Solid colour or a reference image that is used to fill areas of an image. 2. A command in the Retouch menu that is used to fill images with colour or a reference image.
filter	A computerized tool used to alter images. Filters can be used, for example, to soften jagged edges, sharpen blurred contours, and eliminate colour banding.
fit-to-fill	An insert edit where an incoming source clip replaces an existing segment (or gap) in the record clip. A fit-to-fill edit functions like a swap shot edit except that the edit sequence does not ripple. If the source clip has a different length than the segment it replaces, the source clip is shortened or lengthened proportionally to fit the duration of the original segment.
flex files	A flex files is a log of the Telecine transfer. It contains information about film keykode, video timecode, audio source timecode, and the A-frame start of the 3:2 sequence.
frame	An image that you load into flame from an external source, such as a VTR or framestore. A clip is made of a sequence of frames.
framerate	Framerate is the number of images displayed within a given time period, normally defined in frames per second.
framestore	1. A general term for hard-disk space used to store and retrieve images. Framestores can be disk arrays, a single disk, or a single file in a file system. 2. A disk array.
gain	A colour correction parameter that adjusts colour by multiplying pixel colour values by a percentage value.
gamma	A parameter that adjusts the midlevel grey values in an image. <i>See also</i> brightness.
gamma correction	A process to correct brightness and internal microcontrast within a computer image.
gradient	A blended mix of two or three colours that is used to draw or fill objects.
hard commit	Removing the soft edit properties of an edit sequence. Hard commits are different from soft commits in that hard commits cannot be restored—the fusion of the clips is permanent. Hard commits also force a render on the selected elements. <i>See also</i> soft commit.

hardware inventory	An IRIX command (<i>hinv</i>) that is used to list the hardware, memory, and peripheral equipment in, or connected to, a computer.
head	Video or audio material that has been trimmed out of the front (leading) end of a clip. This material may remain in the clip to be used in a dissolve transition.
hermite	An option for the interpolation of an animation curve that produces a smooth curve by assigning a slope to each control point on the curve. Each control point has a tangent handle that can be used to adjust its slope.
hierarchy	A structure of levels that organizes component elements. For example, the IRIX operating system uses a tree-like hierarchy to organize directories on a hard drive.
highlights	Light areas in an image.
histogram	A bar graph in the Keyer that is used to adjust the values of the red, green, blue, and luma channels of an image when creating a matte.
HLS	A colour model based on human perception of colours. Its primary components are hue, lightness, and saturation.
hue	One of the main characteristics that distinguishes one colour from another. Hue values define a colour by its position on the spectrum from red through yellow and blue.
hybrid splice	Edits or film splices that occur on a "B" or "C" frame in the 3:2 pulldown sequence.
image window	The image viewer that is seen at the top half of the screen when working in modules such as the Keyer or the Colour Corrector.
in point	In points (and out points) are used in editing to determine where and how edits are inserted into the record clip, and to determine what part of a source clip is used in an insert or overwrite. <i>See also</i> out point.
insert edit	The insertion of a source clip into a record clip, in which the record clip edit sequence ripples (the duration changes to accommodate the inserted clip). The material before and after the insert edit is not affected.
interlace	A command in the Image Options menu that recombines deinterlaced fields in a clip, producing a new clip half as long as the original. The process combines, or interlaces, all the fields numbered 1 in the first frame with all the fields numbered 2 in the second frame and produces a new single frame in the new clip. <i>See also</i> deinterlace.

interlaced scan	Interlaced scan systems display all the odd lines, and then all the even lines of an image, creating two fields each made up of half the entire image.
interlacing	The means by which traditional television picture tubes create images on-screen. An interlaced-scanning tube sends information to each pixel in the even-numbered rows of pixels on a screen; left to right, then top to bottom. It then sends information to odd-numbered rows. The even and odd lines of each frame are referred to as fields.
interpolation	A mode that automatically defines the shape of an animation curve between keyframes on the curve. <i>See also</i> extrapolation.
inverse non-additive mix	A mixing process that compares the colour values of the corresponding pixels in the two source clips, and assigns the higher value to the corresponding pixel in the output clip. <i>See also</i> additive mix and non-additive mix.
IRIX	A version of the UNIX operating system that is used by SGI machines.
key	A process in which a hole is cut in a clip, or undesired background elements are removed, using a matte; used to create a composite image by combining images, texts, or other elements from another clip with elements of the image that have not been keyed out (removed).
keycode	A keycode is the bar code printed along the edge of negative film stock. The bar code contains information on manufacturer, film type, emulsion number, date, and footage count. Keycode can be machine read to provide accurate cutting lists.
Keyer	A tool that is used to create a composite from a background and foreground clip by using an input key-in clip to determine how the clips are combined. The input key-in clip is used to create a black and white matte that defines which areas of the foreground and background clips are used in the result clip.
keyframe	A frame within a clip to which unique parameters have been assigned. Once a keyframe is set, these parameters can be animated over the frames between one keyframe and the next (to which unique parameters have also been assigned) using interpolation. Colour correction, morphing, and timewarps are among the effects that can be animated.
layer	1. A combination of a front and matte clip on top of a background clip. 2. Images, objects, scenes, et cetera, that are stacked on top of each other in a frame. Layers are used in Action, Paint, Text, and other modules.

linear	1. A mode of interpolation or extrapolation of an animation curve in which the keyframes are either connected by straight lines (interpolation) or the curve is continued in a straight line before the first keyframe and after the last keyframe (extrapolation). 2. Editing based on the sequence of frames recorded on a tape. <i>See also</i> hermite, cycles, and constant.
logical operation	1. A mathematical calculation performed using the corresponding pixel values of two source clips. The calculation may be addition, subtraction, multiplication, or comparison of minimum or maximum values. 2. A command in the Processing menu which performs any of these mathematical calculations.
logical volume (XLV)	A system of framestore management which is used for third-party disk arrays. Does not guarantee real-time playback, is resolution-dependent, and requires pre-allocation of storage space for each partition created. (only available on Onyx/IR, 02, and Indigo2 Impact systems). <i>See also</i> Discreet Filesystem.
logical volume table (lvtab)	A description of the memory blocks disks used for the framestore.
Look-Up Table (LUT)	Files used to convert colour information in an image.
luma	Brightness; the value that determines the black and white element in an image.
match	Matching individual frames in assembled clips to the corresponding frames in the source clip.
matte	The cutout that is used in the creation of a composite image. The solid black part of the matte represents the area of the resulting image where the background appears. The solid white part of the matte represents that area of the result image where the foreground appears. <i>See also</i> alpha channel.
mesh	A grid that is superimposed over an image for deformations such as morphing or warping.
midtone	Midlevel greys in an image.
mix	The act of combining two video sources.
monochrome	A black and white image, or an image of a single colour.
morph	To transform one image into another image by warping and dissolving between the two images over a series of frames.
motion blur	A digital effect that simulates the blur of fast-moving objects.
motion jitter	Jerky movements in a clip, often caused by gate slip when film is converted into video.

motion path	A Bspline displayed in the image window and a timing curve displayed in the Animation Curve window that defines the translation, or motion, of an object, camera, camera point of interest, displayed axes, or light sources.
node	An object and its axis as represented in Schematic view.
noise	1. Aberrant specks that appear in a video image that increase with each generation of the video. 2. An effect that can be simulated in the Processing menu.
non-additive mix	A mixing process that compares the colour values of the corresponding pixels in the two source clips and assigns the higher value to the corresponding pixel in the output clip. <i>See also</i> additive mix and inverse non-additive mix.
NTSC	National Television Standards Committee. Standards for colour broadcasting used in North America and parts of South America. The NTSC format uses 525 horizontal lines per frame, with two fields per frame of 262.5 lines each. Each field refreshes at 60 Hz. <i>See also</i> PAL.
offline edit	Roughcut editing used to produce an EDL.
offset	1. The horizontal and vertical displacement of a clip. 2. Reference numbers that indicate the change, in terms of frames, that takes place when you trim.
OMFI	Open Media Framework Interchange. A standard file format used for the exchange of digital multimedia data between applications and across platforms.
online edit	Final edit, using the original source material and an EDL. The online edit produces a finished product ready for distribution.
online table of contents	A file listing the contents of an archive. The online table of contents is created automatically by the software and saved in the UNIX file system.
orbit	The rotation of the camera eye around the point of interest.
out point	Out points (and in points) are used in editing to determine where and how edits are inserted into the record clip, and to determine what part of a source clip is used in an insert or overwrite. <i>See also</i> in point.
overwrite edit	The addition of a source clip into a record clip, where the record clip edit sequence does not ripple (the duration does not change). The source clip overwrites an equal number of frames on the edit sequence.

PAL	Phase Alternate Line. The PAL standard uses a total of 625 lines per frame, with two fields per frame of 312.5 lines each. Each field refreshes at 50 Hz. PAL-B (PAL-I) is a European colour broadcasting standard; also used in China, Malaysia, Australia, New Zealand, The Middle East, and parts of Africa. PAL-M is a Brazilian colour broadcasting system. <i>See also</i> NTSC.
pan	1. To reposition an image horizontally or vertically in the image window in order to focus on a particular area of the image. 2. In the Channel Editor, to scroll through the Curves window, move the camera eye and point of interest, or scroll through the scene in the image window.
pan & scan	The process of extracting a 4 x 3 image from a program with a larger aspect ratio; it is usually performed at Telecine transfer. Universal Mastering necessitates pan & scan functions in on-line finishing.
parent	A hierarchal relationship in the Action and Batch Processing modules in which one object is linked to the other, and controls or affects its behaviour.
particles	A stream of 3D models with varying size, speed, and distribution, originating from a surface, a light source, or another 3D model in Action.
particle bouncer	An object or node in Action that influences the behaviour of particles when they come in contact with a surface.
particle generator	An object or node in Action that creates a stream of particles when attached to a light source, 3D model, or surface.
particle manipulator	An object or node in Action that influences either the position or the speed of particles.
partition	A subdivision of the total capacity of a storage disk that creates two or more virtual disks from a single physical disk. In the case of disk arrays, a partition is a virtual array within the whole array.
pathname	A description of a file, program, or directory location on a hard disk drive that incorporates the names of directories and subdirectories. For example, <i>/usr/discreet</i> is an example of the full pathname for the directory that contains flame .
Perspective view	A view of the animation scene from the camera eye, equivalent to the viewing frustum of the camera. In Perspective view, an object's size depends on its distance from the camera eye.
pixel	A word created through the blending of two words: "picture" and "element". As a standard unit of colour: one dot on a screen or monitor. As a picture element or picture cell: one sample of picture information. Pixels are the most basic unit from which video/computer images are created.

point of interest	The focal point of the scene; the portion or area of a scene on which the camera focuses.
priority	The order in which surfaces, 3D models, and 3D text are drawn in the scene in Action.
progressive scan	Progressive scan systems display each line of an image sequentially.
proxy	A scaled-down version of an image stored in the framestore.
rate	A Paint attribute that determines the rate at which brush images are deposited on the canvas.
reel	A section of the work area that displays clips. It includes controls that mimic a VTR.
rendering	The translation of the abstract description of a 3D object into 2D pixel information.
resolution	1. The amount of detail in an image. Higher-resolution images have greater detail. 2. A measurement of image size, usually in pixels.
resolution-independent	A term to describe equipment that can work in more than one resolution. Most equipment can work in film resolution or video resolution, but not both. Resolution-independent equipment can work in both.
reverse	A command used to reverse the order of frames in a clip.
RGB	Red, Green, Blue. An industry colour standard used to describe colour components or colour space. Computers and some analogue component devices use separate Red, Green, and Blue channels to retain full bandwidth, and provide the highest-quality picture.
ripple	A setting that determines how the overall length of the edit sequence is affected when a clip is inserted. When ripple is on, the timecode and frame count increase or decrease (ripple) to accommodate the added or removed material.
rotoscope	The procedure involved in removing an unwanted object from a shot, such as the wires that make an actress appear to fly.
sampling frequency (rate)	The number of sample measurements taken from an analogue signal in a given period of time. These samples are then converted into numerical values stored in bytes to create the digital signal.
saturation	1. The intensity of colours. 2. The degree by which the eye perceives a colour as departing from a grey or white scale of the same brightness.

scene	An image window view in the Action module in which you can see and manipulate objects, axes, lights, and the camera.
Schematic view	An illustration in the Action module that depicts the different relationships between various objects and layers. <i>See also</i> Perspective view.
SCSI	Small Computer System Interface. A standard connection for computers, disk drives, and peripheral devices.
SECAM	SEquential Colour And Memory. A broadcast standard used in Eastern Europe and France that is partially compatible with the PAL standard but incompatible with NTSC. The SECAM standard uses a 625-line 25-frame-per-second signal.
setup files	Files containing customized menu settings, animation channels, key information, and other data. You create and save setup files in the UNIX file system, and can load them for use during another work session.
shadows	Dark areas in an image created by light source direction.
shot	A sequence of images and/or clips.
SMPTE	Society of Motion Picture and Television Engineers
soft clip	A clip assembled using soft edits.
soft commit	Removing the soft edit properties of an edit sequence. The soft properties can be restored at any time by using the U (uncommit) button in the Timeline menu.
soft edit	An electronic edit that maintains source clips in memory and tracking processes so that edits can be modified without starting from scratch.
softness	A blending or mixing along lines or edges in an image.
sourcing	To separate the front and matte clips of a layer in Action so different transformations can be applied to each.
Sparks	Third-party software plug-ins.
spline	A curved line.
stabilize	<ol style="list-style-type: none"> 1. To remove motion jitter and unwanted camera movement from a clip. 2. To track an image in a clip in the Stabilizer.
surface	A representation of a layer that contains material attributes in the Action module.

surface displacement	The adding of a 3D effect to a 2D image by displacing the pixels of the image; also called displacement mapping.
swap shot	An insert edit where the segment of an edit sequence that lies between two transitions is swapped for the incoming source clip. Swap shots ripple, meaning the edit sequence duration changes if the source clip is of a different length than the segment it replaces.
tail	Video or audio material that has been trimmed out of the back (trailing) end of a clip. This material may remain to be used in a transition. <i>See also</i> head.
Telecine	A Telecine is a device that transfers film to video. Telecines are designed to transfer negative, print, duplicate negative or interpositive film. They can transfer film to video at various frame rates and various resolutions.
texture mapping	Attributing a surface quality, such as a colour, roughness, smoothness, or volume to a 3D model.
TIFF	Tagged Image File Format. A tag-based format for storing and interchanging Bitmap (Raster) images.
tiling	A technique for displaying high-resolution images that divides images into portions (tiles) and loads the portions into memory as needed for display on screen.
timecode	The indexing method used for timing and editing video and audio material. The numbers in a timecode denote hours, minutes, seconds, and frames (00:00:00:00) elapsed on a videotape.
timeline	The graphical representation of time in flame ; located just below the image window controls.
timewarp	Speeding up or slowing down the action in a clip by decreasing or increasing the number of frames in a clip.
tolerance	A range of colour values for an operation such as keying or colour correction.
track	1. To pan the camera in order to maintain alignment with a moving object. 2. To attach an axis to a moving background.
transition	A smooth passage from one segment of video or audio to another with a visual effect, creating a segue. A transition can be in the form of a splice, dissolve, or wipe.
transition rate	The number of frames over which a dissolve or wipe transition occurs.

up-converting	Up conversion is the process of transferring images of a lower resolution to images of a higher resolution. It may cause image artifacts if varying frame rates or image filtering is introduced during the conversion process.
vector image	An image described by basic geometric shapes like rectangles, polygons, circles, ellipses, lines, and curves.
view	A perspective of the animation scene from the camera eye, equivalent to the viewing frustum of the camera. In Perspective view, an object's size depends on its distance from the camera.
virtual source	A source clip that generates new frames as needed; it has no real beginning or end. Virtual sources can be trimmed to any extent.
VTR	Video Tape Recorder. Betacam and D1 are VTRs.
warp	To distort or change the shape of an image in a free-form manner.
wipe	1. To apply colour, filter, or an effect to an entire image. 2. An effect that uses a pattern to progressively reveal a new clip and obscure the current clip during a transition.
wireframe	A display option where solid or filled objects are represented by mesh lines and/or curves.
$Y'C_B C_R$	A colour signal which is divided into luma (Y), and the colour difference signals, B-Y and R-Y. Tape uses $Y'C_B C_R$. Synonymous with YUV.
write-on	A series of recorded paint strokes played back in a clip.
zoom	1. To increase or decrease the percentage of an image that you can see in an image window. It gives the impression of moving closer to or farther from an image. 2. To reduce or enlarge the display of the animation curves in the Channel Editor.

index

Numerics

- 2-3 and 3-2 pulldown
 - adding 621
 - removing 618
- 2-3 configurations 312
 - updating 315
- 2-3 Insertion button 307
- 2-3 Removal button 307
- 2-3 Removal Mode 311
- 24p Mastering 257
- 24p mastering 305
 - advantages 305
 - capturing with an EDL 310
 - Engineering menu options 307
 - inputting clips with 2-3 removal 309
 - outputting clips with 2-3 insertion 309
 - PAL mastering 319
 - playing 24p clips 317
- 24p projects
 - archiving 319
 - clip input and output 306
- 2D Histogram node 677
- 3D handles buttons 1004
- 3D histogram 705
 - display options 718
 - displaying 707
 - plotting a colour value in 727, 738
- 3D Keyer 701
 - animating key elements 733
 - grey areas, removing 714, 715, 735
 - hot keys 738
 - how it calculates keyed-out area 708
 - minimizing noise 711
 - overview 702
 - plotting a range of colour values 728
 - plotting a single colour value 726
 - preferences 738
 - RGB Viewer, described 705
 - sampling patches 715
 - sampling softness 709
 - sampling tolerance 706
 - selecting key elements 722
 - speeding up interaction during sampling 738
 - steps for pulling a key 703
- 3D models
 - exploding 1051
 - importing 985
 - importing in Action 983
 - projected textures 996

- sort 991
- specular highlight 990
- texture mapping 992
- transparency 991
- 3d studio max, Action import 983
- 3D text
 - Bevel curve 986
 - beveling 987
 - Create button 986
 - creating in Action 985
 - Depth field 986
 - Font field 986
 - Frame button 986
 - Italic field 986
 - Kern field 986
 - Replace button 986
 - Reset button 986
 - Size field 986
 - Text field 986
 - Undo button 986
- 3D Text Editor controls 986

A

- A&V Synch box in Player 80
- AccLine manipulator 1038
- AccPlane manipulator 1039
- AccPoint manipulator 1037
- Action
 - 3D models, importing 983
 - 3d studio max, importing 983
 - 3D text, creating 985
 - Add/Blend options 922
 - adding elements 957
 - adding lights 968
 - Analyze command 929
 - Apply button 919
 - axes 940
 - camera 945
 - creating branches 963
 - displacement mapping 999
 - drawing order 928
 - Edit Offsets button 925
 - Front view 949
 - grids 944
 - hiding elements 962
 - image resolution 898
 - Import Geometry 984
 - keying 915
 - layers 910, 919

- Layers menu 908
- loading the clips 892
- locking the front clip 911
- naming elements 959
- offsetting surfaces 924
- Perspective view 949
- planes 943
- scene 949
- Schematic view 955
- Shading button 898, 974
- shadows 978
- shaping surfaces 925
- Side view 949
- Source nodes 931–937
- Sources, motion blur 935
- stabilizing from 808
- Surface Geometry controls 989
- surface order 928
- surface transparency 922
- surfaces, characteristics of 907
- Texture button 898
- texture mapping 992
- Top view 949
- Z-Buffer box 981
- Z-buffer option 930
- Z-sort command 929
- See also* light sources
- Action setup, removing 901
- Active button 548, 753
- Add and Blend box, 3D models 990
- Add blend, projector 998
- Add button, in Graphics 1116
- Add mode, in Channel Editor 134
- Add, Replace, and Rename prompt
 - while archiving 368
 - while saving clips 202
- additive key
 - creating in Keyer 653
 - creating in Modular Keyer 688
- additive mix 422
- adjusting EQ 470
- advanced gradient 758
 - removing gradient for hard edges 760
 - softness borders 759
 - softness vertices 759
 - using with pickers 760
- A-Frame Timecode field 307
- alpha channels, extracting 1178
- Alt View button, in Batch 564, 583
- Ambient, 3D models 989
- Analyze command, in Action 929
- analyzing clips 791
 - one frame at a time 818
 - speeding up the process 804
- anamorphic, geometry 1130
- animating
 - Paint strokes 1106
 - text 872
- animation
 - animating shapes 1139
 - animating the front clip 778
 - camera 945
 - changing the timing 163
 - Channel Editor 129
 - Channel Hierarchy 130
 - explicit 148
 - expressions 165
 - in Warper 847
 - motion paths 141, 142
 - of garbage masks manually 750
 - of garbage masks with Tracker 754
 - of offsets in four-point tracking 804
 - of selected vertices on garbage masks with Tracker 755
 - of settings in Keyer 652
 - previewing in Paint 1065
 - setting keyframes 151
- animation curves
 - Break mode 135
 - changing the colour 158
 - changing the Y scale 140
 - copying 156
 - defined 128
 - extrapolation mode 154
 - interpolation mode 152
 - making discontinuous 153
 - moving 137, 141, 160
 - negating 160
 - resetting tangents 135
 - reversing 160
 - selecting 138
 - tangents, breaking and resetting 135
 - X scale, changing 139
 - Y scale, changing 140
- Animation menu
 - Expr button 167
 - Link button 168
- anti-aliasing 784, 903
 - in Warper 850
- Anti-Aliasing box, Source nodes 935
- Anti-Aliasing Softness field, Source nodes 935
- Apply button 919
- archives
 - Add 368
 - ASCII table of contents 354
 - Clear/Load button 370
 - Clear/Load in Library button 371
 - clip library 364
 - closing 372
 - compatibility 352
 - creating 356
 - file 360
 - file opening 363
 - Format button 359
 - Keep button 372
 - loading a D1 Archive from flame 3.9.10 371
 - loading clips from 369
 - menu 352
 - online table of contents 354
 - opening 362

- recovering 372
 - removing clips from 371
 - Rename 368
 - Replace 368
 - saving clips, desktop, reels, soft edits 364
 - structure 354
 - tape 359, 363
 - tips for VTR archiving 358
 - viewing 374
 - VTR 356, 362
 - archiving 24p projects 319
 - archiving process, aborting 366
 - arrow cursor 37
 - arrow, in key elements, controlling 725
 - ASCII table of contents 354
 - recovering an archive 372
 - viewing an archive with 374
 - aspect ratio 123
 - changing 92
 - projector 996
 - audio 451
 - avoiding clicks and pops 461
 - changing sync between monitors 472
 - copying audio tracks 418
 - cutting clips with 398
 - deleting from clips 415
 - Discreet 464
 - editing 383, 385
 - editing gesturally 384
 - editing in Edit Reel 387
 - importing 453, 464
 - input and output delay 245
 - inputting clips with 232
 - moving audio tracks 417
 - outputting clips with 245
 - playback fidelity 80
 - preferences 125
 - removing files 466
 - rendering in Edit Reel 388
 - sampling rates 461
 - scrubbing 458
 - scrubbing in modules 458
 - setting audio delay 459, 472
 - SGI 452
 - sparks 483
 - starting product with 452
 - sync issues 461
 - synchronization 80
 - timewarps 481
 - waveforms 463, 475, 478
 - word synd 476
 - Audio editor 386
 - audio patching, in an EDL 276
 - audio preferences, setting for clip I/O 245
 - Audio TimeStretch 481
 - audio tracks
 - selecting for input 240
 - selecting for output 249
 - Auto Edit after Add button 1117
 - Auto Key
 - Automatic Keyframing 532
 - button 149, 151, 788, 896
 - in 3D Keyer 738
 - in Modular Keyer 698
 - timewarps 447
 - Auto Matte command 500, 1113
 - Auto mode, in Channel Editor 135
 - Auto Pan option 821
 - Auto Scale, timewarps 447
 - Auto Tangents mode, for garbage masks 743
 - Auto Update button, in Batch 580, 590
 - auto-logging clips 234
 - AutoPaint 1101
 - playing recorded strokes 1105
 - Random mode 1103
 - recording brush strokes 1104
 - tracking 1105
 - User mode 1104
 - using Graphics 1138
 - Wipe mode 1101
 - AutoPoint button 1137
 - AutoSave timing 122
 - Average option 162
 - Avid Video Resolution (AVR) 348
 - Axis menu, in Garbage Mask menu 748
 - axis system 940
- ## B
- back matte, adjusting 644
 - background
 - changing the brightness 118
 - changing the brightness in Colour Corrector 518
 - changing the contrast 118
 - backing up a key setup 676
 - BackLUT 644
 - Backup button and field
 - in Batch 591
 - Basic Blend 693, 694
 - Basics menu
 - in Colour Corrector 522
 - in Colour Warper 542
 - Batch
 - Alt View button 564
 - Auto Update button 590
 - branch 574
 - bypassing nodes 578
 - Cache menu 564
 - copying nodes 576
 - CurBack view 564
 - CurFront view 564
 - CurMatte view 564
 - CurResult view 564
 - deleting nodes 577
 - execution by command prompt 589
 - frames, specifying 587
 - Graph, selecting 574
 - grouping nodes 575
 - introduction 563
 - Keyer node 603

- Library node 604
- LUT Editor node 605
- Modular Keyer node 609
- Monochrome node 609
- moving nodes 576
- multiple node connections 570
- Negative node 610
- Node bars 564
- Node menu 564
- nodes and clips, connecting 568
- Optics node 610
- Output node 611
- output or export node name 587
- Posterize node 611
- Process tree menu 564
- process trees 593
- processing the tree 586
- Quick Composite node 612
- Reels node 612
- reference box 582
- rendering setups from UNIX shell 20
- Restore Prev button 590
- saving setups 589
- Schematic view 564
- Setup menu 564, 589
- setups, loading 590
- Spark node 613
- Status Webpage button 590
- Text Log button 590
- Unbypass button 590
- variables in scripts 588
- Viewing buttons 564
- viewing in-context 582
- Batch menu 564
- Bevel curve 986
- Bezier curves
 - for polygons 1122, 1137
- bicubic scaling quality 329
- bicubic surfaces 917, 918
 - changing the shape 925
- bilinear surfaces 917
 - changing the shape 925
- black level, setting monitor 175
- Black Lv field
 - adjusting contrast 547
 - matching luminance 542
- blend modes, projector 996, 998
- blend nodes 662, 690
 - adding mattes 693
 - Basic Blend 693
 - blending front and back images 694
 - bypassing pipes 698
 - CBlend node 691
 - changing the order of layers 697
 - Curves Blend 694
 - MBlend node 692
 - multiple layers, using 696
 - pipes 662, 690
 - renaming layers 697
- Blend, projector 998
- blending layer components, in Modular Keyer
 - Basic Blend 694
 - Curves Blend 695
- blue arrow cursor 37, 51
- blur filter
 - in Keyer 648
 - in Modular Keyer 677, 684
- Blur medium 1086
- Blur tool, in Keyer
 - softening edges of matte with 646
- Blur tool, in Modular Keyer 682
- blurring
 - colour images in Modular Keyer 677
 - edges of matte, in Keyer 647
 - edges, in Modular Keyer 682, 684
- bold font, for buttons 118
- bouncers, particle 1017, 1047
 - behaviour 1047
 - customizing 1047
 - friction 1048
 - MaxDist 1048
 - resilience 1048
 - types 1047
- Box filter, blurring matte with 648
- branches, defined 892
- Break mode 135
- breaking tangents 750
- brightness, changing background 118
- browser, file 61
- brush attributes 1076
 - Direction 1079, 1081
 - Jitter 1078
 - modes 1080
 - Opacity 1076
 - Pressure 1078, 1082
 - Rate 1077
 - Roll 1080
 - Size 1077
 - using reference images 1081
- brush setups, saving and loading 1160
- Brush Spacing 1082
- brush types 1070, 1116
 - custom 1161
- brushes
 - selecting 1070
 - See also* special effects media
- Bspline curves, for polygons 1122
- Burn In TC command 616, 626
- button types 53
- buttons
 - bold fonts 118
 - cloning 101
 - determining hot key for 98
- C
- cache
 - Batch, using 584
 - cache tab on node 568
- Cache button
 - Batch menu 564

- cache tab
 - states 584
- calculator 56
- calibrating
 - the monitor 177
- camcorder
 - inputting material from 250
- camera
 - animating in Compositor 776
 - clipping planes 945, 948
 - depth of field 906
 - eye 945, 948
 - field of view (fov) 946
 - Focal Length 946, 947
 - frustum 947
 - F-Stop 946
 - panning 953
 - physical Cam button 946
 - point of interest (poi) 945, 948
 - roll 946
 - rotating 953
 - zooming 953
- camera, inputting material from 250
- Canvas button 1069
- canvas, in RGB Viewer, displaying 719
- capturing
 - for 24p mastering 309
 - from a VCR, camera or camcorder 250
 - from a VTR 232
 - single frames 235
- Cardinal curves, for polygons 1122
- CBlend node 691
 - adding a matte 693
- CDs, importing tracks from 454, 466
- centre guide 71
- CHAN menu. *See* Channel menu
- Channel Editor 129
 - animating key elements in 3D Keyer 733, 735
 - expressions 165
 - Info view 167
 - keyframe indicators 151
 - linking channels with expressions 168
 - particle parameters 1025
 - setting Shape keyframes in 3D Keyer 734
 - tabular format 167
 - zoom mode 141
- Channel Hierarchy 130
- Channel menu 633
 - custom colour option 634
 - using a primary colour 633
- Channel View box 133
- channels
 - defined 127
 - selecting 138
- channels, video and audio
 - selecting for input 240
- characters, text layers 857
- chroma suppression 526
- Cineon, converting 1178
- circles, painting 1074, 1120
- Clean button, in Tracer 769
- Clear button, in Colour Warper 535
- Clear/Load button 214, 370
- Clear/Load in Library button 371
- clip
 - defined 38
 - reference 81
- clip compatibility 223
- clip duration, displaying 121
- clip input
 - dropping frames during 234, 265
 - troubleshooting 265
 - with 2-3 removal 309
- Clip input and output
 - Troubleshooting 265
- clip libraries
 - archiving 352, 364
 - Batch, accessing from 604
 - creating 218
 - default 203
 - entries 203, 227
 - fields in 203, 227
 - loading clips, reels, desktops 213
 - loading discreet effects clips into editing products 225
 - managing 217
 - opening 199
 - organizing contents 220
 - removing 218
 - removing entries from 222
 - saving clips to 201
 - selecting entries 212
 - size of 200
 - structure 203
 - structure of 203
 - viewing modes 205
- clip management. *See* clip libraries
- Clip Name field 238
- clip name, displaying 120
- clip output
 - to VTR 245
 - troubleshooting 265
 - with 2-3 insertion 309
- clips
 - adding glow effects 786
 - archiving 352, 364
 - changing field dominance 622
 - collapsing 39
 - combining channels from different 618
 - compatibility between Discreet products 223
 - compressing 618
 - copying 418
 - copying frames in 410
 - copying in a clip library 221
 - cutting 396
 - degraining 504
 - deinterlacing fields 620
 - deleting 30, 414, 415
 - dividing into smaller clips 619
 - duration 39

- expanding to 30 FPS 621
- formatting 622
- frame numbers 39
- frames for processing 64
- generating monochrome clips 625
- incoming 380
- inputting from DDR 252
- inputting from DDR in emulation mode 232
- inputting from tape device 264
- inputting from VCR, camera or camcorder 250
- inputting from VTR 232
- inputting with 2-3 removal 309
- inserting frames into clips 403
- inserting into other clips 401, 404
- interlacing 623
- interleaving 623
- key-in 632
- loading 213
- matching 47
- matching source 408
- mixing 429
- moving 416, 417
- names 39
- naming 416
- navigating desktop reel 43
- outgoing 380
- outputting to DDR 254
- outputting to VCR 251
- outputting to VTR 245
- outputting with 2-3 insertion 309
- playing 24p clips 317
- playing, in a module 80
- playing, on the desktop 40
- recording to tape drive 263
- regraining 506
- replacing frames in 406
- reversing field dominance 625
- reversing frame order 411
- saving to clip library 201
- scrubbing through 40
- searching for 46
- selecting 39
- sharing 25
- specifying location in projects 25
- splicing 400
- timecodes 39
- working with 38
- Clone medium 1087
- Cloud Opacity field, setting 721
- collapsing clips 39
- collapsing vertices in Channel Editor 744
- colour
 - adjusting balance 526
 - adjusting in Modular Keyer 678, 691
 - bars 503
 - curves 531
 - remapping values 530
 - sampling 520
 - See also* colours.
- colour clamping
 - in Colour Warper 536
- Colour Correct node 679
 - in Batch 595
- colour correction
 - in Action 915
 - in Modular Keyer 679
 - in Paint 1166
 - of front clip when keying 651
- Colour Corrector
 - accessing from Compositor 782
 - accessing from Modular Keyer 679
 - adjusting the gain 525
 - adjusting the gamma 525
 - adjusting the hue 523
 - adjusting the offset 525
 - animating setup values 532
 - Basics Menu 522
 - C1, C2, and !C buttons 679
 - changing the background brightness 518
 - chroma suppression 526
 - clip loading from other components 516
 - colour balance 526
 - Context button 915
 - contrast 523
 - crop box 518
 - cutouts 1156
 - displaying clips in 517
 - dynamic updating (Regen) 518
 - gain 526
 - hiding the menu 516
 - histogram 528
 - hue 526
 - input levels 528
 - Invert Matte button 519
 - loading clips into 515
 - matching colours 531
 - Monochrome command 524
 - Negative command 524
 - output levels 529
 - previewing the changes 517
 - reducing contrast 530
 - reference buffer 81
 - remapping colour values 530
 - rewiring RGB channels 524
 - sampling colours 520
 - sampling from the front and back clips 521
 - saturation 523
 - saving setups 519
 - shadows, midtones, highlights 522
 - viewing Action result 679
- colour curves in Keyer, editing 650
- Colour Curves node 678
- colour matching
 - in Colour Corrector 531
 - in Colour Warper 541
- colour model
 - selecting to key with 636
- colour palette 1063
 - colour gradient 1065
 - mixing colours 1064

- using and customizing 60
- colour picker 57
- colour sampling
 - processing order 521
- colour spill
 - adjusting saturation 650
 - suppressing 649
- colour temperature 177
- Colour Warper 533
 - accessing 534
 - node 534
 - reference buffer 81
- Colour Warper node, in Batch 596
- Coloured Frame command 503
- Coloured Frame node, in Batch 596
- colours
 - channel values, viewing and setting 59
 - mixing 58
 - sampling for an average 58
 - selecting 57
- Combine command 618
- combustion
 - compatibility of setups 112
- command prompt
 - Batch, executing 589
- commands
 - selecting clips for 51
 - startup options 18
 - undoing 52
- Commit command 391
- compact archives 364
- compatibility
 - archives 352
 - of clips from different Discreet products 223
 - of setups between products 112
- compositing with Quick Composite 786
- Compositor
 - animating the camera 776
 - animating the front clip 778
 - colour correcting from 782
 - loading the clips 773
 - motion paths 780
 - setting preferences 783
 - shadows 779
 - tracking 781
- Compound command 493
- Compound Over value 493
- configuration file
 - declaring DDRs 255
 - FontMapping keyword 857
- Consolidate command 414
- Constant
 - extrapolation 155
 - interpolation 152
- Constant Shape button 747
- constant timewarps 444
- context points
 - setting 672
 - viewing 673
- contrast 523
 - changing background 118
 - decreasing 529
 - increasing 528
- conventions, in User's Guide 16
- Convert button 1138
- converting Cineon images 1178
- converting timecodes to frame numbers 56
- convertmacfont utility 879
- convex hull
 - reshaping 731
 - softness 711
 - tolerance 708
- Copy button
 - in Batch 573
 - in Paint 1127
- Copy command 418
- copying
 - nodes 667
 - preferences 27, 28
 - setups 25
- copying clips 418
 - in a clip library 221
- crawl, creating in Text 871
- Create button 986
- Create User menu 27
- creating
 - clip libraries 218
 - projects 25–26
 - soft partitions 89
 - user name 27–28
- credits
 - creating a text roll 869
- Crop Auto option 774, 775
- crop box
 - in Colour Corrector 518
 - in Compositor 774
 - in Keyer 632
 - in Modular Keyer 675
- Crop Edges option 822
- cropping
 - front clip 774
 - key area 632, 675
- cropping images
 - when importing or exporting 328
- cross cursor, yellow 37
- crossfades 422, 425
- CurBack, in Batch 564
- CurFront, in Batch 564
- CurMatte, in Batch 564
- current colour, in Paint 1063
- Current Layer proxy 696
- Current Result proxy 661
- CurResult
 - in Batch 564
- CurResult button
 - in Batch 581, 596
- cursor, solving jitter problem 125
- cursors 37
 - gestural editing 382
 - moving using the stylus 37

- trim 436
 - Curves Blend 694, 695
 - Curves menu
 - in Colour Corrector 530
 - custom menus 125
 - loading 126
 - removing setups 126
 - saving 126
 - custom particles 1026
 - customer support, contacting 16
 - Cut mode 964
 - cutouts
 - adding drop shadows 1152
 - animating 1156
 - blending 1154
 - colour correcting 1156
 - drop shadows 1155
 - embossing 1152
 - extruding 1153
 - FrameCut 1149
 - GeoCut 1150
 - in editing 379, 390
 - Keep cut On 1149
 - PolyCut 1150
 - RectCut 1150
 - cutting
 - at splices 397
 - audio gesturally 400
 - clips gesturally 399
 - Cut command 396
 - every nth frame 397
 - gesturally 399
 - with audio tracks 398
 - with dissolves and timewarps 398
- D**
- Data tape
 - Input and output 263
 - DDRs
 - defining in configuration file 255
 - inputting clips 252
 - inputting clips in background 253
 - inputting from in VTR emulation mode 232
 - outputting clips 254
 - outputting clips in background 254
 - Deal command 619
 - Decimation field
 - in Warper 850
 - default
 - directories 106
 - preferences 111
 - project 23
 - user 23
 - Define Style button 873
 - Degrain node 685
 - in Batch 597
 - degraining clips 504
 - in Modular Keyer 685
 - Deinterlace command 620
 - DeInterlace node
 - in Batch 598
 - Delete button
 - in Batch 573
 - Delete command 414
 - in Graphics 1127
 - Delete mode
 - in Batch 572
 - in Channel Editor 136
 - deleting
 - all clips on desktop 415
 - clips 30, 415
 - clips gesturally 415
 - nodes 667
 - projects 30
 - setups 30
 - users 31
 - Depth field 986
 - Depth of Field button 906
 - desaturating highlights
 - in Colour Warper 552
 - description (project)
 - modifying 29
 - specifying 25
 - Desk Cache option 124
 - desktop 38
 - archiving 364
 - deleting all clips 414
 - loading 213
 - naming 42
 - saving 42, 201
 - desktop preferences 118
 - desktop reel
 - defined 38
 - naming 42
 - navigating clips on 43
 - reel controls 43
 - saving 42
 - scrolling through 44
 - setting direction 123
 - setting number to display 123
 - setting orientation 123
 - working with 38
 - destination reel
 - selecting 51
 - diagnostics 116
 - Difference command 498
 - Difference Matte node 680
 - Diffuse, 3D models 989
 - Direction brush attribute 1079, 1081
 - directories 106
 - accessing 106
 - default 106
 - loading setups and preferences to 108
 - removing multiple setups from 110
 - removing setup and preferences from 109
 - saving setups and preferences to 108
 - Directory box 61
 - Discreet Audio 464
 - and sync 475
 - playing 480

- preferences 471
 - record and playback options 468
 - working with waveforms 477
- displacement mapping 999
- display LUTs 117
- Dissolve field 847
- dissolves 422
 - alignment options 423
 - creating complex 428
 - cutting 398
 - keeping or removing source clips 424
 - modifying in Edit Reel 432
 - rates 423
 - render options (EDL Import) 293
 - rendering options 423
 - types 422
- Do Not Render option 423
- documentation
 - online 4
 - set 4
- Don't Map Sources 447
- drag and drop
 - editing operations 120
 - in clip libraries 220
- Drag medium 1088
- Draw Border button 761
- Draw Pickers button 761
- Draw Spline button 761
- drop frame material
 - inputting with 2-3 removal 308
- Drop Frame Reference Timecode field 308
- drop frame tape
 - outputting to with 2-3 insertion 309
- drop shadows
 - adding to cutouts 1152
 - for cutouts 1155
 - in Action 978
- dropping frames during clip input 265
- Dst button 539
- duration
 - displaying 121
- dynamic animation 165
- E**
 - Edge node 681
 - edge-detection filters 560
 - in Modular Keyer 681
 - Edit Configuration button 312
 - edit decision lists. *See* EDLs
 - Edit Mode box
 - in Batch 572
 - Edit Offsets button 803, 925
 - Edit Reel 386, 387
 - audio rendering 388
 - playing clips 388
 - virtual sources 388
 - Edit Reel setups 388
 - editing
 - audio and video together 383
 - clips 396
 - commands 380
 - menu 380
 - timecodes 390
 - See also* gestural editing
 - editing audio 387
 - gesturally 384
 - with editing commands 385
 - edits (EDL)
 - defined 272
 - EDLs
 - assembling 292
 - assembling multiple 273
 - audio patching 276
 - autocapture 287
 - Auto-Capture status boxes 291
 - auto-capturing 287
 - capture handles 290
 - capturing for 24p projects 310
 - dissolve render options (EDL Import) 293
 - editing 274
 - editing transitions 277
 - event translation 298
 - exporting 296, 299
 - exporting timewarps 300
 - Fix Match Frame 273
 - generating 296
 - hotkeys 302
 - importing through network 302
 - Library Menu 273
 - limit handles 300
 - loading 272
 - loading from PC 273
 - logging clips to, from Input Clip menu 234
 - menu 270
 - saving generated 297
 - searching 294
 - selecting tracks for EDL generation 297
 - selective capture 289
 - setting default reel 298
 - setting output options 299
 - setting Save options for export 297
 - skipping clips 290
 - slipping events 280
 - timewarp render options (EDL Import) 293
 - using 2-3 removal mode 311
 - EDLs preferences 285
 - EDLs timewarps 281
 - Effects menu
 - overview 629
 - Spark buttons 885
 - ellipses
 - painting 1120
 - ellipsoids
 - scaling 735
 - selecting 722
 - setting display options 719
 - softness 710
 - tolerance 707
 - See also* key elements
 - enabling a matte 1109

- entries (EDL)
 - defined 272
- EQ
 - adjusting 470
 - resetting 471
- erasing a matte 1112
- Erode tool
 - in Keyer 646, 647
 - in Modular Keyer 682
- eroding edges, in Modular Keyer 682
- events (EDL)
 - defined 272
- example expression setups 171
- example particle setups 1056
- Exit command 118
- exiting 20
- expanding vertices in Channel Editor 744
- explicit animation 148
- Export node
 - in Batch 599
 - process trees 572
- exporting
 - clips to OMF 345
 - images 321, 338
 - LUTs 331
- exporting EDLs 296
- Expr button
 - Animation menu 167
- Expression button 133
- expressions
 - animation 165
 - Expr button 167
 - function reference 169
 - linking channels 168
 - setups 171
- extended bicubics 1002
- Extra Tangents button 761
- extracting an alpha channel 1178
- extrapolation mode 154
- F
 - fade-ins and fade-outs 422, 427
 - field dominance
 - setting for clip input and output 241
 - Field Dominance box 622
 - field information, preserving 620
 - field jitter
 - removing effects caused by 624
 - Field Merge command 624
 - Field Merge node
 - in Batch 600
 - Field mode
 - using in Stabilizer 819
 - field of view 946
 - fields
 - deinterlacing in clips 620
 - dominance 622
 - entering numeric values in
 - entering text in
 - merging 624
 - recombining after deinterlacing 623
 - reversing dominant 625
 - file archive
 - creating 360
 - opening 363
 - file browser 61
 - Filter tool 62
 - loading files 63
 - selecting files 63
 - file formats 322
 - File list 62
 - Fill command 1097
 - fill objects
 - drawing 1124
 - Fill option 822
 - film
 - compressing to 24 FPS 618
 - expanding to 30 FPS 621
 - Film Compress command 618
 - Film Compress node, in Batch 601
 - Film Expand command 621
 - Film Expand node, in Batch 601
 - film-based material, 24p mastering 305
 - Filter library 557
 - Filter node, in Batch 600
 - Filter tool (in file browser) 62
 - filters
 - applying to a clip 556
 - blurring 560
 - custom 558
 - edge detection 560
 - in Graphics 1132
 - in Paint 1084
 - inverting 560
 - matrix 558
 - opening the Filter menu 555
 - Prewitt 560
 - procedural 560
 - Sobel 560
 - using to blur mattes 648, 684
 - Fit button 985
 - fit method
 - applying when importing or exporting images 328
 - Fix Match Frame (EDL Import) 273
 - Fixed Reference button 820
 - Fixed X button 806, 811
 - Fixed Y button 806, 811
 - Flip command 494
 - Flip node, in Batch 601
 - Flipbook 1065
 - flipping images 494
 - Flush button
 - in Batch Setup menu 591
 - in Colour Warper 541
 - Focus Point option box 432
 - Font field 986
 - FontMapping keyword 857
 - fonts
 - convertmacfont* utility 879

- importing 879
 - installing new 19
 - loading (in Text) 878
 - Force Render option 392, 423
 - Format button, in Archive 359
 - Format commands 622
 - Format menu 615
 - four-point tracking 801
 - positioning the trackers 802
 - using offsets 803
 - FOV
 - projector 996
 - Frame
 - Increment field 238
 - mode 819
 - Frame All button 133
 - Frame button 986
 - Frame Chn button 133
 - frame depth
 - changing 92
 - Frame Grab, using 235
 - Frame Number field in image window controls 65
 - frame numbers
 - converting to timecodes 56
 - displaying 120
 - frame rate
 - changing 618, 621
 - frame, defined 38
 - FrameCut 1149
 - frames 64
 - changing the FPS rate 618, 621
 - combining 493
 - copying 410, 418
 - deleting 414
 - for processing 64
 - inserting in clips 401
 - matching 47
 - matching source 408
 - moving 416
 - replacing 406
 - reversing 411
 - selecting 39
 - unrendered 443
 - framestore
 - basics 198
 - listing contents 1171
 - setup 85
 - transferring clips 1173
 - See also* partitions
 - framestore utilities
 - from_framestore 1173
 - list_framestore 1171
 - remote access 1168
 - restrictions 1174
 - to_framestore 1173
 - free guide 70
 - FrmSnap option
 - in Channel Editor 162
 - from_framestore utility 1173
 - remote access 1168
 - restrictions 1174
 - front clip
 - adding a shadow to 779
 - animating in Compositor 778
 - cropping 774
 - Front pipe
 - of CBlend node 691
 - of MBlend node 692
 - FrontLUT 644
 - frustum 947
 - Full Resolution/Proxies box in Player 78
 - Full Screen Player 78
 - full-resolution film images
 - painting 1066
 - functions
 - expressions, using with 169
- ## G
- gain
 - adjusting 525
 - adjusting luminance of key with 644
 - gamma
 - adjusting 525
 - monitor 179
 - Gamma fields, in Colour Warper 547
 - Garbage Mask menu
 - opening from Keyer 742
 - opening from Modular Keyer 756
 - garbage masks 741
 - active and inactive vertices 752
 - adding and removing vertices 752
 - animating 750
 - animating in Tracer 765
 - animating selected vertices with Tracker 755
 - animating with Tracker 754
 - changing the drawing order 748
 - creating 745
 - deleting vertices from 753
 - determining use by plotting regions 729
 - drawing freehand 743, 745
 - hot keys 770
 - modifying position, shape and size of 748
 - moving vertices and tangent handles 749
 - reference buffer 81
 - ROI (Region of Interest) 757
 - selecting vertices and tangent handles 749
 - setting drawing options 743
 - setting the opacity 746
 - setting the softness 747
 - turning on and off 747
 - using straight edges 750
 - using the Tracer 758
 - Gaussian blur
 - applying to colour images in Modular Keyer 677
 - blurring matte with 648, 684
 - generated EDLs, saving 297
 - generating monochrome clips 524
 - generators, particle 1016
 - 3D model 1017, 1020

- changing parameters 1023
- exploding 3D models 1051
- exploding surfaces 1051
- light source 1017
- stream types 1021
- surface 1017, 1019
- Update button 1023
- GeoCut 1150
- Geometry
 - Add and Blend box 990
 - Ambient 989
 - Diffuse 989
 - painting with 1072
 - Shine 989
 - Sort button 990
 - Sort Order box 990
 - Specular 989
 - Subdivide 990
 - Transp field 989
 - Use Submaterials 990
 - Wireframe 990
- gestural editing 382
 - cursors 382
 - enabling 383
 - insert, splice, replace 408
 - options 119
- glow effects
 - adding to clips 786
 - animating 788
- GMask node
 - in Batch 602
- Goto
 - button 41
 - hot key 41
- Grab button
 - in Batch Setup menu 591
 - in Colour Warper 541
- gradient mattes 1112
- gradients, animating 1142
- grain, processing 504
- graininess
 - removing from softened areas of a key 711
- Graph
 - Batch, selecting 574
- Graphic Regen button 1129
- graphics
 - adding graphics 1116
 - animating gradients 1142
 - animating graphics 1139
 - blending 1154
 - brush 1116
 - deleting graphics 1127
 - drawing graphics 1117
 - drop shadows 1155
 - editing graphics 1127, 1134
 - embossing 1152
 - extruding 1153
 - Fill tool 1124
 - Fuzzy option 1131, 1132
 - gradients 1132, 1142
 - Graphic Regen button 1129
 - hiding control points 1130
 - hiding graphics 1128
 - menu 1116
 - moving graphics 1134, 1135
 - object attributes 1131
 - Outline option 1131, 1132
 - Push command 1128
 - Recall command 1143
 - resizing graphics 1134, 1135
 - resolution 1128
 - rotating graphics 1134, 1135
 - rubber banding 1123
 - saving graphics 1143
 - selecting graphics 1126
 - shape animation 1139
 - shapes 1136
 - Store command 1143
 - tacking graphics 1144
 - text 1125
 - Transformation Box 1134
 - using AutoPaint with 1138
 - Wire option 1131
 - wireframe 1129, 1131
- Gravity manipulator 1034
- green arrow cursor 37, 51
- grid 72
 - defined 66
 - in Action 944
 - in Paint module 75
- grid and guides
 - turning on and off 76
- Grid and Guides menu 66
 - in Paint module 75
 - Miscellaneous controls 73
 - Preferences 74
 - Quick Off button 76
 - Save Zoom button 74
- Grid and Snap buttons
 - in Batch 591
- Grid/Guide box 66
- Group button
 - in Batch 573
- guide
 - centre 71
 - free 70
 - ratio 68
 - safe area 70
 - working area 67
- H
 - HDCAM Codec 256
 - clip input 257
 - clip output 261
 - Hardware configuration 257
 - Manual decoding and encoding 260
 - Regenerating Proxies 261
 - head and tail clips 380
 - head and tail offsets 437
 - help, getting 16

- Hermite interpolation 153
 - hiding vertex channels 744
 - Hig button 552
 - Highlights trackball 543
 - Histo option box, in Colour Warper 540
 - histogram
 - 2D histogram in Modular Keyer 677
 - 3D histogram in 3D Keyer 705, 718
 - in Keyer 641
 - histogram, luma
 - in Colour Warper 543
 - HLS colour model 636
 - Home button
 - hue cube in Colour Warper 541
 - vectorscope in Colour Warper 541
 - Hot Key editor 97
 - hot keys
 - 3D Keyer 738
 - creating 100
 - determining for buttons 98
 - editing 100
 - in Colour Warper 540
 - in Warper 851
 - local, shared, global 99
 - managing 101
 - Modular Keyer 700
 - predefined 99
 - reference buffer 83
 - RGB Viewer 739
 - house sync 476
 - hue 523
 - hue cube, in Colour Warper 550
 - Hue field, in Colour Warper 543
 - hue shift, creating 651
- I
- Image Export controls, in Batch 599
 - image files
 - exporting 338
 - formats 322
 - importing 322
 - lookup tables 330
 - numbering format 342
 - Image Import controls, in Batch 602
 - image resolution
 - in Action and Compositor 902
 - image surfaces 917
 - image tracking, *see* tracking
 - image window 64
 - display controls 66
 - Pan button 66
 - play controls 65
 - zoom 66
 - images
 - applying colours and filters 1095
 - applying random paint strokes 1103
 - Import from Setup button, Source nodes 936
 - Import Geometry menu 984
 - Fit button 985
 - Rotate Axis button 985
 - Separate Nodes button 984
 - Smooth button 984
 - Import node 566
 - in Batch 602
 - importing
 - audio 453, 464
 - files in Action 899
 - fonts 879
 - images 321, 322
 - LUTs 330
 - Impressionist medium 1088
 - in and out timecodes
 - setting for input 238
 - setting for output 247
 - in Modular Keyer 698
 - incoming clip 380
 - In-Context viewing, in Batch 582
 - Increasing Editing Speed 1151
 - Indirect button 1022
 - Info view
 - expressions, viewing in Channel Editor 167
 - Input Clip menu 232
 - Input Delay 241
 - audio 245
 - inputting clips
 - from DDR 252
 - from tape device 264
 - from VCR, camera or camcorder 250
 - from VTR 232
 - Insert command 401
 - gestural edit 382
 - insert edit 408
 - inserting clips
 - gesturally 405
 - with audio 405
 - inserting frames and clips 403
 - interface
 - changing brightness and contrast 118
 - primary 35
 - Interlace command 623
 - Interlace node, in Batch 598
 - interlaced images, tracking 819
 - interlacing
 - display options 122
 - Interleave command 623
 - Interpolate field 847
 - interpolation
 - expressions, using with 166
 - while deinterlacing 620
 - interpolation mode 152
 - setting for garbage mask animation 753
 - Inventor, Action import 983
 - inverse non-additive mix 423
 - inverting the matte 649, 1111
 - isomorphic LUT
 - LUT editor, in Batch 608
 - Italic field 986
- J
- jitter

- of cursor, removing 125
 - removing 806, 809
 - removing from animation curve 162
- Jitter brush attribute 1078
- Jitter Filter 125
- JL Cooper MCS-3800 102
- JPEG compression 349
- K**
- Keep command 222
- Keep Cut On 1149
- Keep/Remove Sources options 390, 424
- Kern field 986
- kernel, troubleshooting 265
- key elements 716
 - animating 733
 - controlling the arrow 725
 - controlling the display of 719
 - displaying outlines 719
 - selecting 722
 - translating, scaling and rotating 729
 - See also* ellipsoids or patches
- keyboard
 - on-screen 55, 857
 - turning on and off 119
 - using in Text 857
- Keyer
 - additive key, creating 653
 - adjusting edges of key 646
 - adjusting the luminance 641
 - animatable parameters 652
 - colour correcting from 651
 - Comp button 635
 - Context button 915
 - displaying key during processing 653
 - garbage masks, using 741
 - loading the clips 632
 - plotting a colour value 639
 - slip synching 653
 - softening the matte 638
 - suppressing colour spill 649
 - tolerance 639–640
 - viewing key with a solid colour background 635
- Keyer node
 - example 579
 - in Batch 603
- keyframe indicators 151
- keyframes
 - adding 134
 - averaging 162
 - copying 151
 - defined 128, 132
 - deleting 136
 - moving 137
 - removing 151
 - selecting 138
 - setting 151
 - setting in Warper 848
- key-in clips 632
 - analyzing colour mapping 706
 - colour correcting 679
 - colour, adjusting 691
 - softening with blur filter 648
- KeyIn Filter, using to soften matte 648
- keying
 - additive key, creating in Keyer 653
 - additive key, creating in Modular Keyer 688
 - from Action 915
 - plotting colour values 639, 726
 - selecting a keying technique 633
 - steps in the 3D Keyer 703
- keypad. *See* numeric keypad
- keywords
 - FontMapping 857
- Kill process command 20, 21
- L**
- lasso option, in Garbage Mask menu 743
- layers 907
 - applying to surfaces 919
 - blurring 912
 - changing 910, 919
 - colour correcting 915
 - defined 892
 - deleting 914
 - keying 915
 - using in blend nodes 696
- Layers List 908
- Letterbox option 822
- Library menu 196
- Library node 566
 - in Batch 604
- lift, adjusting luminance of key with 644
- light source
 - generators 1017
- lights
 - adding 968
 - enabling 969
 - spotlights 970
- limit handles, EDL export 300
- Linear
 - curves for polygons 1122
 - extrapolation 155
 - interpolation 153
- lines
 - painting 1072, 1118
- Lines button, in 3D Keyer
 - viewing key element outlines 719, 723
- Lines button, in Colour Warper 540
- Link button, Animation menu 168
- list view 208
 - sorting in 209
- list_framestore utility 1171
- Load button
 - in Batch 590
- Load command
 - in Library menu 213
 - on work reels 214
- loading

- files in Action 899
- files in file browser 63
- preferences from other users 32
- setups and preferences 108
- setups from other projects 32
- loading clips 213, 214
 - from DDR 252
 - from tape device 264
 - from VTR 232
 - in Colour Warper 534
- Lock Clip command 114
- Lock Key button 814
- locked reels
 - moving 123
- locking keyframes, in Stabilizer 814
- locking reels 44
- Logic Op node 571, 604, 685
- Logic Ops, using with Curves Blend 695
- logical operations 495
 - in Batch 604
- lookup tables 330
 - creating 331
 - Lookup Table Editor 332
- luma values
 - calculated by two source clips 685
- luminance
 - adjusting in mattes 641, 678
 - creating mattes according to 640
 - mattes, values for 1113
- luminance curves 652
 - adjusting 644, 686
 - calculation 687
- LUT 330, 332
 - exporting 331
 - importing 330
 - loading a custom 332
 - saving a custom 338
 - switchable display 117
- LUT Editor node
 - in Batch 605
 - in Modular Keyer 686
- M**
- magnet 1010
 - Magnet Curve Editor 1007
 - properties, defining 1007
- Main menu 36, 49
- manipulators 1029
 - AccLine 1030, 1038
 - AccPlane 1030, 1039
 - AccPoint 1030, 1037
 - damping 1032
 - Function 1041
 - Gravity 1030, 1034
 - magnitude 1031
 - Path 1030, 1040
 - position influence 1031
 - power 1031
 - speed influence 1031
 - Transform 1030, 1032
 - Vortex 1030, 1036
- manual tracking 817
- Map Source 447
- masks. *See* garbage masks
- Match button
 - in Colour Corrector 531
 - in Colour Warper 541
- Match source command 408
- matching colours
 - in Colour Corrector 531
 - in Colour Warper 541
- matrix filters 558
- matte clips
 - creating 498, 500, 680
- Matte Curves menu 678, 686
 - curves calculation 687
- Matte Curves node 678
- Matte Invert Lock button 1111
- matte operations
 - setting up using MBlend node 692
- Matte pipe
 - of CBlend node 691
 - of MBlend node 692
- Matte Transparency button 1110
- mattes
 - adding to blend nodes 693
 - adjusting the luminance 641
 - Auto Matte command 1113
 - blurring using filters 648
 - blurring with Edge node 684
 - changing colour 1110
 - creating 1111
 - creating according to luminance 640
 - creating gradients 1112
 - edges, blending 682
 - edges, removing pixels 682
 - edges, softening 682
 - enabling 1109
 - erasing 1112
 - filters, applying Erode, Shrink, and Blur 681
 - generating from two clips 680
 - grey areas, removing 714, 715, 735
 - in surface particle generator 1019
 - Inverting 519
 - inverting 649, 1111
 - inverting in Modular Keyer 686
 - loading 1111
 - luminance curves, adjusting 678, 686
 - luminance values 1113
 - negative sampling 714
 - sampling patches 715
 - saving 1112
 - transparency 1110
 - using 1109
 - viewing 1110
- Max blend, projector 998
- MaxLibrarySize token 200
- MBlend node 692
 - adding a matte 693
 - Front and Matte pipes 692

- media. *See* special effects media
- memory, playing from 79
- menu area 49
- menus
 - Archive 352
 - customizing 125
 - EDL 270
 - EDL Library 273
 - Grid and Guides, in Paint module 75
 - Network 226
 - Rendering Setup 1113
 - Spark command 886
- menus (custom)
 - loading 126
 - removing setups 126
 - saving 126
- meshes
 - copying and pasting 839
 - for morphing 842
 - in Warper 828
- message bar 49
- Mid button 552
- midtone triangles
 - in Colour Warper 543
- Midtones trackball 543
- Min blend
 - projector 998
- minimizing noise, in softness 711
- Mix command 429
- Mixed extrapolation 153, 156
- Mixed interpolation 153
- mixing colours 58
- Modify Project menu 29
- Modify User menu 30, 31
- modifying
 - preferences directory 30
 - project description 29
 - project name 29
 - timewarps 447
 - user name 30
- Modular Keyer
 - accessing Colour Corrector from 679
 - accessing from Action 915
 - additive key, creating 688
 - backing up the setup 676
 - blend nodes 690
 - changing the image in image window 660
 - colour boxes, using 674
 - description of tools 655
 - displaying menus 659
 - front and matte luminance curves 686
 - hot keys 700
 - layers, selecting 694
 - nodes corresponding to traditional Keyer 658
 - opening 656
 - playing clips 672
 - processing the composite 689
 - saving the setup 676
 - See also* processing pipeline
- Modular Keyer node 609
- modules
 - selecting clips for 51
- monochrome clips
 - creating 501
- monitor calibration
 - colorimeter for 177
 - comparing 177
 - flowchart 174
 - gamma 179
 - manual 186
 - troubleshooting 192
 - when to calibrate 173
- monochrome
 - generating clips in 625
- Monochrome command 501
- Monochrome node
 - in Batch 609
 - in Modular Keyer 686
- morphing
 - between live-action clips 846
 - between still clips 845
 - defined 825
 - dissolve rate 847
 - interpolation rate 847
 - meshes for 842
 - one image into another 842
 - procedure 843
- motion blur
 - in Action 904
 - using Average 491
- Motion Blur button, Source nodes 935
- Motion Blur curve, Source nodes 936
- motion jitter
 - removing 809
- Motion Path button 821
 - projector 996
- motion paths 141
 - defined 142
 - in Compositor 780
- mouse 37
 - using in Paint 1062
- Move button
 - in Colour Warper 551
- Move command 416
- Move mode
 - in Batch 572
 - in Channel Editor 137
 - moving axes in Action 942
- moving
 - animation curves 137
 - clips 416, 417
- multiple EDLs
 - assembling 273
- Multiply blend
 - projector 998
- music CDs, importing tracks from 454, 466
- N
- name (project)
 - modifying 29

- specifying 25
- Name command 416
- Name field
 - in Batch 573
- Negate blend
 - projector 998
- Negate option 160
- negative clips
 - creating 502
- Negative command 502
 - in Modular Keyer 686
- Negative node 686
 - in Batch 610
- negative sampling, in 3D Keyer 714
- negatives, generating from a source image 686
- Network menu 226
 - library controls 227
- new features, list of 6
- node bars
 - Batch menu 564
 - in Batch 567
- Node button, Batch menu 564
- Node menu 567
- nodes
 - adding in Batch 567
 - adding to pipeline 664
 - blend 662
 - bypassing in Batch 578
 - cache tab 568
 - cached tab, states 584
 - clips and nodes, connecting 568
 - Colour Correct 595
 - Colour Warper 596
 - Coloured Frame 596
 - copying 667
 - copying in Batch 576
 - corresponding to traditional Keyer 658
 - defined 892
 - Degrain 597
 - DeInterlace 598
 - deleting 667
 - deleting in Batch 577
 - Export 599
 - Field Merge 600
 - Film Compress 601
 - Film Expand 601
 - Filter 600
 - Flip 601
 - GMask 602
 - grouping in Batch 575
 - Import 602
 - Input and Output nodes 593
 - Interlace 598
 - Keyer 603
 - Library 604
 - Logic Op 604
 - LUT Editor 605
 - moving in Batch 576
 - moving on pipeline 666
 - multiple input 662

- Negative 610
- Optics 610
- Output 611
 - placing on pipeline 699
- Posterize 611
- Quick Composite 612
- Reels 612
- Regrain 597
- renaming 670
- selecting on pipeline 663
- single input 661
- Spark 613
- tabs described 568
- unbypass 590
- updating 668
- See also* processing pipeline
- noise
 - removing 493
 - removing from softness of key 711
- non-additive mix 423
- normal archives 364
- numeric fields. *See* fields
- numeric keypad
 - using to calculate timecodes 56

O

- offsets
 - adjusting 525
 - head and tail 437
- offsets, in four-point tracking
 - animating 804
 - generating with Stabilizer 803
 - setting manually 803
- OMF files
 - AVR options 348
 - cropping exported images 347
 - cropping imported images 350
 - exporting clips to 345
 - importing clips from 349
 - JPEG compression with 349
- one-point tracking 798
- online table of contents 354
- Opacity brush attribute 1076
- Opacity field, for patches 716
- Open Media Framework. *See* OMF files
- opening an archive 362
- Optics module 786
- Optics node, in Batch 610
- option box, defined 54
- Optisense device 177
- Orbit mode, rotating the camera 953
- OTOC 354
- outgoing clip 380
- Output Delay
 - audio 245
 - VTR 242
- Output node
 - in Batch 611
 - process trees 572
- output options, EDL export 299

- outputting clips
 - to DDR 254
 - to tape drive 263
 - to VCR 251
 - to VTR 245
- Over value 809
- Oversampling button 822
- P**
- Paint 1059
 - animating Paint strokes 1106
 - Auto Matte command 1113
 - AutoPaint 1101
 - brush attributes 1076
 - brush setups 1160
 - brush, selecting 1070
 - colour correcting the clips 1166
 - colour palette 1063
 - copying graphics 1127
 - creating a matte 1111
 - custom brushes 1161
 - Fill command 1097
 - filters 1084
 - Flipbook 1065
 - full-resolution film images 1066
 - Grid and Guides menu 75
 - library 1158
 - loading a matte 1111
 - media. *See* special effects media
 - painting on the canvas 1069
 - recording brush strokes 1104
 - reference images 1081
 - revealing the back image 1089
 - Roll command 1071
 - saving a matte 1112
 - saving and restoring images 1067
 - selecting colours 1063
 - setups 1158
 - tracking with AutoPaint 1105
 - using a matte 1109
 - Wipe command 1095, 1101
 - See also* Graphics
- painting with geometry 1072, 1074
- PAL mastering 319
- Pan button (in image window) 66
- Pan mode
 - in Channel Editor 137
 - panning the camera 953
- panning images, when importing and exporting 329
- paragraphs
 - text layers 857
- Parent mode
 - in Batch 572
 - in schematic view 963
- parent to child
 - defined 892
- particle setups 1056
- particles 1015
 - applying textures to 1028
 - behaviour 1017
 - bouncing 1017, 1047, 1055
 - Channel Editor 1025
 - Colour_V 1025
 - custom 1026
 - Factor field 1024
 - generating streams 1016, 1017
 - generation parameters 1022
 - generators 1016
 - Indirect button 1022
 - Interactive Type box 1023
 - lifetime 1022
 - light source 1017
 - Mass 1025
 - Mass_V 1025
 - number of 1022
 - on particles 1053
 - position 1016
 - Pre field 1021
 - Render Type box 1023
 - rendering streams 1023
 - selecting type 1023
 - Shutter value 1025
 - size 1022
 - speed 1016, 1022
 - Spread 1026
 - Spread_V 1026
 - Subdiv field 1024
 - TailSize 1025
 - TailSize_V 1025
 - Timestep field 1021
 - Trail button 1022
 - TrailSize 1023
 - types 1024
 - varying parameters 1023
- partitions 85
 - adding 89, 90
 - changing 91
 - list 88
- patches 715
 - adjusting transparencies with 737
 - animating 733, 735
 - display options 721
 - opacity, setting 716
 - removing grey areas from matte with 715
 - selecting 722
 - setting the softness of 716
 - See also* key elements
- Path manipulator 1040
- pathname 106
- PDF documentation 4
- Percentage Mix value 430
- Perspective view 949
- Phase field, Source nodes 936
- pickers
 - animating sampled values 766
 - assigning to selected vertices 762
 - defined 759
 - interpolating sampled values between key-frames 767
 - selecting 749

- using with advanced gradient 760
 - working with 758
- picking colours 58
- picking elements, in RGB Viewer 722
- Picking option box, in 3D Keyer 724
- picture level
 - setting monitor 176
- pipeline. *See* processing pipeline
- pipes, in blend nodes 690
 - bypassing 698
- planes
 - in Action 943
- Play Audio Only option in Player 79
- Play from Disk option in Player 79
- Play from Memory option in Player 78
- Play Lock 236
- Play menu area in Player 78
- play to air 80
- Play to Video option in Player 79
- Player 77
 - Full Screen 78
 - Play menu 78
 - setup options 78
- playing
 - a playlist 80
 - clips in the Player 77
 - clips, on the desktop 40
 - Discreet Audio 480
 - SGI Audio 479
 - SGI audio 463
- playing clips
 - in Edit Reel 388
 - in Modular Keyer 672
 - playing 24p clips 317
- Plot button 544
- plotting colour values, in 3D Keyer 726
 - displaying or hiding 722
 - in 3D histogram 727
 - range of values 728
 - single colour value 726
- plotting colour values, in Keyer 639
- plug-ins
 - loading 886
 - using 886
- point of interest 945
 - See Also* camera
- pointer preferences 124
- PolyCut 1150
- polygons, drawing 1121
- Pop button 748
- Posterise command 502
- Posterize node, in Batch 611
- postroll, setting for clip I/O 241
- posttrail 446
- Pre field 1021
- preferences
 - copying 27, 28
 - defined 106
 - Drag & Drop option 120
 - Gestural Editing option 120
 - loading 108
 - loading factory default 111
 - loading from other users 32
 - removing from directories 109
 - saving 108
 - saving as default 111
 - setting in Compositor 783
 - sharing 27, 28
 - specifying 27
- Preferences controls, Grid and Guides menu 74
- preferences directory 28
 - modifying 30
- Preferences menu 118
 - in Colour Warper 540
- preroll, setting for clip I/O 241
- Pressure brush attribute 1078
 - mode 1082
- Pressure Threshold field 124
- pretrail 446
- Preview command
 - in Paint 1067
- primary interface 35
- Priority Editor 928
- procedural filters 560
- process indicator
 - Batch Queue Manager 587
- Process tree menu
 - described 572
 - in Batch 564
- process trees
 - assembling in Batch 565
 - branch 574
 - completing 572
 - connecting nodes 569
 - creating 593
 - definition of branches 574
 - moving nodes 576
 - multiple node connections 570
 - processing 586
- processes, stopping 20
- processing
 - grain 504
 - in Colour Warper 536
 - in Compositor 785
 - in Keyer 652
 - in Modular Keyer 689
 - in Warper 849
- Processing menu
 - in Batch 563
- processing pipeline
 - 2D Histogram node 677
 - 3D Keyer node 701
 - adding clips to 671
 - adding nodes to 664
 - blend nodes 690
 - CBlend node 691
 - Colour Correct node 679
 - Colour Curves node 678
 - context points, setting 672
 - Degrain node 685

- Difference Matte node 680
 - Edge node 681
 - Front, Back and Matte branches 658
 - Logic Op node 685
 - Matte Curves node 678
 - MBlend node 692
 - moving nodes 666
 - Negative node 686
 - panning 674
 - placing nodes 699
 - processing the composite 689
 - Regrain node 685
 - Result node 658, 686
 - RGB Blur node 677
 - structure 657
 - using layers in blend nodes 696
 - See also* Modular Keyer
 - product compatibility
 - setups 112
 - Project box 62
 - projector
 - Action 996
 - position X,Y,Z fields 996
 - Projector menu 996
 - Projector node 992, 997
 - projects
 - clip location, specifying 25
 - clips, deleting 30
 - clips, sharing 25
 - creating 25–26
 - default 23
 - deleting 30
 - description, modifying 29
 - modifying 29
 - name, modifying 29
 - naming 25
 - resolution 24
 - setups, copying 25
 - setups, deleting 30
 - setups, load from other projects 32
 - setups, sharing 25
 - setups, specifying 25
 - switching to another project 23
 - Proxies mode 206
 - proxies, caching in memory 124
 - Proxy Aspect box 123
 - proxy depth, changing 93
 - proxy size, changing 93
 - proxy transparency
 - adjusting 120
 - pulldown
 - adding 621
 - removing 618
 - Punch Back Only option, in Keyer 653
 - Push button 748
- Q**
- quality, setting for image import and export 329
 - Queue Manager
 - Batch, accessing in 587
 - controls, described 587
 - Export node 599
 - Node Name 587
 - scripts in Batch 587
 - Quick Composite module 786
 - Quick Composite node
 - in Batch 612
 - Quick Off button in Grid and Guides menu 76
 - quitting 20
- R**
- R button, in 3D Keyer 711
 - Rate brush attribute 1077
 - ratio guide 68
 - record and playback options
 - Discreet Audio 468
 - SGI audio 456
 - Record Configuration area. 314
 - record timecode 390
 - recording
 - SGI audio 455
 - recording clips
 - to DDR 254
 - to tape drive 263
 - to VTR 245
 - Recover command 115
 - recovering lost work 115
 - rectangles, painting 1073, 1118
 - Rectangular Zoom mode 137
 - RectCut 1150
 - Recursive Clone medium 1090
 - red arrow cursor 37, 51
 - Redo button 52
 - in Colour Warper 536
 - reel controls 43
 - reels
 - archiving 364
 - desktop, defined 38
 - loading 213
 - locking 44
 - naming desktop 42
 - navigating on desktop 43
 - number of displayed 123
 - reel controls 43
 - removing 222
 - saving 42, 201
 - saving desktop 42
 - scrolling locked 45
 - scrolling through 44
 - scrubbing locked 45
 - Reels node 566
 - in Batch 612
 - Ref button 544
 - reference box
 - changing the colour 820
 - defined 790
 - in Batch 573, 582
 - positioning 794
 - resetting 796
 - resizing 794

- setting magnification 822
 - reference buffer 81
 - controls 591
 - Flush button 591
 - Grab button 591
 - hot keys 83
 - in Batch 573
 - in Colour Warper 541
 - reference clip 81
 - reference frame
 - tracking 793
 - reference point 820
 - changing 815
 - selecting 790, 793
 - Regen button 738
 - Regen in Colour Corrector 518
 - regenerate command 117
 - Regrain node 685
 - in Batch 597
 - regraining clips 506
 - in Modular Keyer 685
 - Rem Key button 151
 - RemKey 698
 - Remove button 30
 - removing
 - audio files 466
 - clips 30
 - removing jitter 809
 - setting the Over value 809
 - renaming nodes 670
 - render mode 443
 - rendering options
 - rendering by field or frame 652
 - Repeat command 410
 - Replace button 986
 - Replace command 406
 - gestural replace 383
 - replace edit 408
 - replacing clips 407
 - Reset All button 133
 - Reset button 986
 - in Batch 573
 - in Colour Warper 535
 - Reset button, Source nodes 936
 - Reset Chn button 133
 - Reset Key 698
 - resetting
 - mattes, in Paint 1112
 - reference box 796
 - shift values 797
 - tracker box 796
 - trackers 795
 - resetting EQ 471
 - Resolution value 902
 - in Compositor 784
 - resolution, changing 24
 - Restore button
 - in Paint 1067
 - Restore Prev button 111
 - in Batch 590
 - Result node 658, 686
 - Reveal medium 1089
 - Reverse command
 - in Editing menu 411
 - Reverse Dominance command 625
 - Reverse option 160
 - reversing timewarps 443
 - RGB Blur node 677
 - RGB channels
 - combining 618
 - rewiring 524
 - separating 625
 - RGB colour model 636
 - RGB Softness
 - compared to YUV Softness in 3D Keyer 714
 - RGB Viewer 705
 - black canvas background 706, 719
 - controlling the display of 718
 - displaying outlines in 719
 - hot keys 739
 - manipulating key elements 729
 - plotting colour ranges 728
 - plotting colour values 727, 738
 - resetting 717
 - reshaping convex hull 731
 - rotating, zooming and translating 707, 717
 - selecting key elements 722
 - softness, viewing 710
 - tolerance, viewing 707
 - RGBCMYl model 636
 - rhosts file 1168
 - ROI (Region of Interest) 757
 - Roll brush attribute 1080
 - Roll option 822
 - Rotate Axis button 985
 - Rotate mode
 - rotating axes in Action 943
 - rotoscoping 750, 1062
 - Rubber Banding button 1123
- ## S
- S button, in Colour Warper 550
 - safe area guide, defined 70
 - Same Clips option 52
 - Sample field
 - in Warper 850
 - Sample On button 766
 - Samples field, Source nodes 936
 - sampling
 - setting rendering method 738
 - saturation 523
 - Saturation field
 - in Colour Warper 543
 - Save As button 108
 - in Action 899
 - Save button 110
 - in Batch 589
 - in Library menu 201
 - in Paint 1067
 - save options, EDL export 297

- Save Zoom button, Grid and Guides menu 74
- saving
 - clips 214
 - clips to a clip library 201
 - setups and preferences 108
- Scale mode 749
- Scale Setup 120
- scaling images
 - when importing and exporting 328
- scene
 - defined 892
 - viewing 949
- schematic
 - clips, adding in Batch 566
 - in Batch 564
- Schematic view 955
 - adding icons 957
 - creating branches 963
 - defined 892
 - hiding elements 962
 - naming icons 959
- Screen blend
 - projector 999
- Screen blend, projector 999
- scripts
 - Batch and variables 588
- scrolling locked reels 45
- scrubbing
 - audio 458
 - audio in modules 458
- scrubbing locked reels 45
- Search command 46
- Select mode, in Channel Editor 138
- selecting
 - colours 57
 - colours in Paint 1063
 - files in file browser 63
 - users 24
- Selection Mode box, in Batch 573
- selective colour correction
 - in Colour Warper 547
- Selective menu 548
- Separate command 625
- Separate Nodes button 984
- Set Context button
 - in Batch 573, 583
- Set Key button 151
- SetKey 698
- setting the background brightness and contrast 118
- Setup button
 - Batch menu 564
- Setup menu
 - in Batch 589
 - in Garbage Mask menu 743
- setup options
 - Player 78
- setups
 - Action, deleting 901
 - Action, saving, loading, importing 899
 - archiving 375
 - Batch, loading 590
 - compatibility between products 112
 - copying 25
 - defined 105
 - deleting 30
 - examples 5
 - fast re-save of 110
 - in Modular Keyer 676
 - in Paint 1158
 - load from other projects 32
 - loading 108
 - removing from directories 109
 - removing multiple items 110
 - restoring backup copy of 111
 - saving 108
 - saving, in Batch 589
 - scaling 120
 - sharing 25
 - specifying in projects 25
- SGI audio 452
 - playing 463, 479
 - preferences 459
 - record and playback options 456
 - recording 455, 467
- Shad button 552
- Shade medium 1093
- Shading button in Action 898, 974
- shadows
 - in Action 978
 - in Compositor 779
 - keying using YUV softness 713
- Shadows trackball 543
- shape animation 1139
- Shape keyframe, setting for key elements 734
- Shape menu, in Keyer 746
- shapes
 - in Graphics 1136
- sharing
 - clips 25
 - material with other projects 27
 - preferences 27, 28
 - setups 25
- Sharpness field, in Colour Warper 549
- Shift option 822
- shift values 791
 - copying 823
 - resetting 797
- Shine
 - 3D models 989
- shots 380
 - swapping 412
- Shrink tool
 - in Keyer 646
 - in Modular Keyer 682
- shrinking edges, in Modular Keyer 682
- Shutter field, Source nodes 936
- Simple Add blend
 - projector 999
- Simple Add blend, projector 999
- Simplify option 161

- Size brush attribute 1077
- Size field 986
- slide lock 437
- Slip field, in Batch 566
- slip synching 653
- slipping events
 - in an EDL 280
- Smear medium 1091
- Smooth button 984
- soft cutouts 390
- soft cuts
 - enabling 119
- soft edits 379, 389
 - archiving 352, 364, 366
 - committing 391
 - enabling 119
 - matching with source clips 408
- soft partitions
 - adding 89
 - creating 89
- softness
 - in Action 903
 - in Colour Warper 550
 - in Compositor 784
 - Source nodes 935
- softness border 759
 - removing gradient 760
 - separating from mask border 761
- Softness Scaling fields, in 3D Keyer 737
- softness vertices, on garbage masks 759
 - selecting 749
- softness, in 3D Keyer
 - adjusting for transparencies 710
 - adjusting numerically 737
 - animating 733
 - appearance of ellipsoid, setting 720
 - displaying convex hull 711
 - ellipsoid 710
 - grey areas, removing 714, 735
 - grey areas, removing with patches 715
 - negative sampling 714
 - removing noise from 711
 - sampling 709
 - setting softness range 702
 - YUV softness 713
- softness, in Keyer
 - range 635, 638
- Solid Line button 1129
- sort
 - 3D models 991
- Sort box 62
- Sort button
 - 3D models 990
- Sort Order box
 - 3D models 990
- source clips 379
- Source nodes 931–937
 - Anti-Aliasing box 935
 - Import from Setup button 936
 - Motion Blur button 935
 - Motion Blur curve 936
 - Phase field 936
 - Reset button 936
 - Samples field 936
 - Shutter field 936
 - Softness field 935
- source timecode 390
- Spark command menu 886
- Spark menu 887
- Spark node, in Batch 613
- Sparks
 - loading 886
 - using 886
 - audio 483
- special effects media
 - Blur 1086
 - Clone 1087
 - Drag 1088
 - Impressionist 1088
 - in Graphics 1132
 - Recursive Clone 1090
 - Reveal 1089
 - Shade 1093
 - Smear 1091
 - Stamp 1091
 - Warp 1092
 - Wash 1093
- specular highlight, 3D models 990
- Specular, 3D models 989
- speed curve 142
 - timing curve 444
- Splice command 400
- splicing 408
 - all clips on a reel 401
 - clips with audio 402
 - two clips together 400
- Spline keyframing 751
- split bar 82
 - in Batch 573
- Spotlight
 - slide projector effect 999
- Spotlight Blend effect 999
- spotlights 970
- squares, painting 1118
- Src button 539
- stabilization process 791
- Stabilizer
 - accessing 790
 - adjusting tolerance value 815
 - analyzing backward 816
 - copying shift channels 823
 - correcting errors 812
 - four-point tracking 801
 - importing and exporting data 823
 - locking keyframes 814
 - manual tracking 817
 - one-point tracking 798
 - positioning reference box 794
 - preferences 819
 - rendering options 822

- resizing the image 822
 - selecting a reference point 793
 - shift values 791
 - stabilizing methods 806
 - tracking by axis in Warper 837
 - tracking by vertices in Warper 838
 - tracking live-action clips in Warper 836
 - tracking with 798
 - two-point tracking 798
 - using multiple trackers 804
 - using thru Warper 836
- stabilizing
 - clips with camera roll 808
 - clips with no camera roll 807
 - clips with no pan 807
 - from Action 807
 - from the desktop 807
 - removing border 822
 - removing border from Action 808
- Stamp medium 1091
- Start and End Frames
 - in Batch 587
- starting the software 17
- starting with audio 452
- startup options 18
- Status Webpage button 590
- Step button, in Stabilizer 818
- Step Frame Processing field (in image window) 64
- Step Offset option 123
- streams, particle 1016, 1047
 - combining 1053
 - generating a second stream 1053
 - parameters 1021
 - rendering 1023
 - types 1021
- Styles
 - saving and loading in Text 874
- Styles menu 873
- stylus
 - adjusting the sample rate 124
- Subdirectories list 62
- Subdivide
 - 3D models 990
- Subsetups
 - in Colour Warper 552
- Subtract blend
 - projector 999
- support, customer, contacting 16
- Surface Geometry menu 989
 - Add and Blend box 990
 - Ambient 989
 - Diffuse 989
 - Shine 989
 - Sort button 990
 - Sort Order box 990
 - Specular 989
 - Subdivide 990
 - Transp field 989
 - Use Submaterials 990
 - Wireframe 990
- Surface Projector menu 996
- surface, defined 892
- surfaces 907
 - bicubic 917, 918, 925
 - bilinear 917, 925
 - changing the order 928
 - changing the shape 925
 - changing transparency 922
 - exploding 1052
 - image 917
 - offsetting 924
 - reordering 928
 - selecting 919
 - shadows 978
- Swap option 160
- Swap Shot command 412
- swapping shots
 - with audio 413
- switching to another project 23
- sync
 - house sync 476
 - setting for clip output 243
- sync and Discreet Audio 475
- sync issues
 - audio 461
- system
 - recovering from a failure 115
- T
- T button, in Colour Warper 550
- tablet 37
- tabular format, Channel Editor 167
- tail offsets 437
- tangent handles
 - extended bicubics 1003
- Tangent R and Tangent L options 159
- tangents
 - adjusting adjacent tangents automatically 750
 - breaking and reconnecting 750
 - moving 749
 - removing 750
- tape archive
 - creating 359
 - opening 363
- Tape Library button 264
 - saving clips 263
- tapes
 - with inconsistent pulldown 317
- target clip 379
- Test Disks command 116
- Text
 - adding in Graphics 1125
 - animating 872
 - Asian character sets 857
 - Attributes menu 854
 - background color 861
 - crawling 871
 - creating 3D 985
 - creating roll 869
 - fonts supported 878

- fonts, Asian 857
 - layers 857
 - loading fonts 878
 - on-screen keyboard 857
 - opening the Text module 854
 - outline 860
 - previewing and processing 881
 - roll/crawl, cropping 871
 - shadow 861
 - Styles menu 873
 - styles, loading and saving 874
- text crawl 871
- Text field 986
- text fields. *See* fields
- Text Log button
 - in Batch 590
- texture
 - projection 992
- Texture button in Action 898
- texture mapping 992
 - Cylindrical mode 994
 - in Warper 849
 - planar maps 994
 - reflection maps 993
 - Wrap mode 994
- Thick Line button 1129
- Threshold Test button 125
- Tiff file format 322
- timecodes
 - Burn in TC button 616
 - calculating, with numeric keypad 56
 - converting to frame numbers 56
 - displaying 120
 - setting for clip input 238
 - setting for clip output 247
 - source and record 390
 - stamping 626
- timeline in image window controls 65
- Timestep field 1021
- timewarps 379
 - audio 481
 - Auto Key 447
 - Auto Scale 447
 - complex timing effects 444
 - constant and variable 444
 - creating complex 444
 - creating simple 442
 - cutting 398
 - exporting in EDLs 300
 - in an EDL 281
 - modifying soft edits 447
 - posttrail and pretrail 446
 - render mode 443
 - render options (EDL Import) 293
- Titles mode 205
- To 828
- to_framestore utility
 - remote access 1168
 - restrictions 1174
- tolerance, in 3D Keyer
 - animating 733
 - appearance of ellipsoid, setting 720
 - convex hull, viewing 708
 - ellipsoid 707
 - sampling 706
 - setting tolerance range 702
- tolerance, in Colour Warper 550
- tolerance, in Keyer
 - setting tolerance range 635, 639
- tolerance, in Stabilizer
 - adjusting 815
- tonal range 522
- Total Frames field in image window controls 65
- Tracer
 - advanced gradient mode 758
 - animating sampled picker values 766
 - animating the mask 765
 - Clean button 769
 - fine-tuning the matte 763
 - hot keys 770
 - pickers, defined 759
 - pickers, working with 758
 - pulling keys with 758
 - steps to pull a key with 761
 - tracking with 768
 - troubleshooting 768
 - viewing options 761
 - when to use 758
- Track Editor 163
- trackballs 543
- tracker box
 - changing the colour 820
 - defined 790
 - positioning manually 813
 - preventing from straying 812
 - resetting 796
 - resizing 794, 813
- tracker number 790
- trackers
 - defined 790
 - hiding 821
 - positioning 794
 - resetting 795
 - selecting 793
- tracking 798
 - analyzing backward 816
 - animating garbage masks 754
 - animating selected vertices on garbage masks 755
 - colour corrected copy of clip 819
 - correcting errors 812
 - displaying the tracking path 821
 - four-point 801
 - in Compositor 781
 - in Paint 1105
 - manual tracking 817
 - modifying the path manually 818
 - one frame at a time 818
 - one-point 798
 - two-point 798

- using multiple trackers 804
 - using offsets 799
 - with Tracer 768
 - See also* Stabilizer.
- tracking process 791
- tracks
 - selecting for input 240
 - selecting for output 249
- Trail button 1022
- trail effects 446
- Transform manipulator 1032
- Transition type 432
- transitions 380
 - handles 380
 - in an ED 277
 - replacing shots between 412
 - trimming 439
- Translate mode
 - in Channel Editor 141
- Translate X and Translate Y options 160
- Transp field
 - 3D models 989
- transparencies
 - adjusting softness for 710
 - adjusting with patches 737
 - using YUV softness 713
- transparency
 - 3D models 991
 - projector 996
- Transparency field 1110
- triangles
 - painting 1074, 1119
- trim
 - cursors 436
- trimming
 - shots with audio 438
 - transitions 439
- troubleshooting, clip I/O 265
- 12-bit partitions
 - using 1174
- partitions
 - 12-bit 1174
- two-point tracking 798
- TX and TY fields 924

U

- Unbypass button, in Batch 590
- undo 52
 - clearing the undo buffer 124
 - setting undo levels 124
- Undo and Redo buttons, animation menu 133
- Undo button 52, 986
 - in Colour Warper 536
- Unrendered Frame message 443
- Update button, in Batch 573
- updating nodes 668
- Use Aspect Ratio button 1130
- Use Submaterials
 - 3D models 990
- User's Guide

- conventions 16
 - overview 15
- users
 - creating 27–28
 - default 23
 - deleting 31
 - modifying 30
 - name, modifying 30
 - preferences directory, modifying 30
 - preferences, copying 28
 - preferences, load from other users 32
 - preferences, sharing 28
 - preferences, specifying 27
 - selecting 24
- utilities 1167
 - convertmacfont* 879
 - from_framestore* 1173
 - list_framestore* 1171
 - remote access 1168
 - restrictions 1174
 - to_framestore* 1173

V

- Value Type 103
- variable timewarps 444
- VCRs
 - inputting material from 250
 - outputting material to 251
- vectorscope
 - toggling on or off in Colour Warper 540
- vectorscope preferences
 - setting in Colour Warper 540
- vertex channels
 - expanding and collapsing 744
 - hiding 744
- vertex scaling 735
- vertical reels 123
- vertices, in 3D Keyer
 - adding to convex hull and moving 732
 - deleting from convex hull 733
- vertices, on garbage masks
 - adding and removing from garbage mask 752
 - animating 751
 - deleting from garbage mask 753
 - making active or inactive 753
- video
 - playback fidelity 80
 - rendering in Edit Reel 387
- video images, tracking 819
- video tracks
 - selecting for input 240
 - selecting for output 249
- Viewing buttons
 - Batch menu 564
- viewing modes (in clip libraries) 205
- virtual sources
 - in Edit Reel 388
- Vortex manipulator 1036
- VTR archive
 - creating 356

- opening 362
- VTRs
 - End of Tape status 237
 - inputting material from 232
 - Local status 236
 - No Cassette status 237
 - outputting clips to 245
 - Play Lock status 236
 - Record Inhibit status 237
 - Standby status 237
 - Tape EE status 237
- W
- Warp medium 1092
- Warp|Gam menu 545
- Warper
 - Add button 829
 - adding a control point 833
 - adding a proportional mesh 828
 - adding meshes 828
 - adding patches 829
 - adjusting the Magnet size 834
 - adjusting the magnet size 834
 - Analyze button 837
 - Anim button 826, 847
 - animation curves 847
 - Auto button 832
 - automatically creating destination meshes 847
 - Break button 832
 - clips, managing 827
 - Copy button 840
 - copying and pasting meshes 839
 - copying tracking data 838
 - creating a freehand mesh 834
 - creating meshes 828
 - creating the transition between clips 847
 - customizing the Magnet effects 835
 - Decimation field 850
 - Delete button 829
 - deleting patches 829
 - destination mesh for morphing 842
 - destination meshes 828
 - dissolve rate 847
 - Freehand button 834
 - generating tracking data 837
 - interpolation rate 847
 - keyframes, setting manually 848
 - live-action clips and Stabilizer 836
 - loading clips 825
 - locking the back clip 842
 - Magnet button 834
 - Magnet Curve Editor 835
 - MEditor button 835
 - mesh, customizing colours 848
 - meshes for live-action clips 836
 - modelling tools 829
 - Morf button 826, 843
 - morphing between live-action clips 846
 - morphing between still clips 845
 - morphing one image into another 842
 - Move button 831, 832
 - moving control points 830
 - moving edges 830
 - moving tangent handles 830
 - opening Warper module 825
 - Paste button 840
 - performing a regional warp 842
 - placing meshes 828
 - procedure 840, 843
 - processing options 849
 - Proportional button 828
 - redimensioning meshes 831
 - Reset All button 826
 - resetting broken tangent handles 832
 - rotating meshes 831
 - Samples field 850
 - saving setups 849
 - scaling meshes 831
 - Select button 830
 - selecting multiple control points 830
 - Setup button 826
 - Setup menu 848
 - source interpolation, using 847
 - source mesh for morphing 842
 - source meshes 828
 - Split button 833
 - Stabilizer 837
 - Stabilizer button 838
 - Stabilizer, loading clips 836
 - SubDiv button 833
 - subdividing a patch 833
 - tangent handles, breaking 831
 - tangent handles, unbreaking 832
 - texture mapping 849
 - tracking by axis 837
 - tracking by vertices 838
 - translating meshes 831
 - using animation curves 847
 - using the magnet 834
 - Warp button 826, 840
 - Warper button 825
 - warping a still clip 842
 - warping an image 840
 - warping and morphing, defined 825
 - warping with the Magnet 834
 - Warper menus 826
 - warping, defined 825
 - Wash medium 1093
 - waveforms
 - audio 463, 478
 - Wavefront
 - Action import 983
 - file format 322
 - white arrow cursor 37, 51
 - White Lv field
 - adjusting contrast 547
 - matching luminance 542
 - Wipe command 1095, 1101
 - wire
 - using to access remote systems 226

- Wireframe
 - 3D models 990
- wireframe, of garbage mask
 - hiding or displaying 744
- word sync 476
- Work On box 549
- work reels
 - archiving 352
 - copying clips on 418
 - deleting clips on 414
 - moving clips on 416
- work session, starting 23–24
- workflow 563
 - Batch process tree 565
- working area guide 67
- working with waveforms
 - Discreet Audio 477

X

- X and Y shift values 791
 - resetting 797
- X-Rite DTP 92 colorimeter 177
- XScale mode 139

Y

- yellow cross cursor 37
- YScale mode 140
- YUV colour model 636
- YUV softness, in 3D Keyer 713

Z

- Z-buffer 930
 - extended bicubics 1004
- Z-Buffer box 981
- Zoom button
 - in Colour Warper 551
- zoom in image window display controls 66
- Zoom mode
 - in Channel Editor 141
 - zooming the camera 953
- Zoomed-Out Paint 1066
- Z-sort command
 - in Action 929

Acknowledgments

User's Guide Team:

Written and illustrated by: Jacqueline Arès, Louis Beauregard, Ed Hawco, Jeremy Kerr, Lisa MacDonald, Stephen McLeod, Marika Piehler, Wanda Pierce, Scott Polzen

Edited by: Mylène Pepin

Layout design by: Brenda Barrie

Section cover illustrations by: Alain Dubois

Technical review by: Niki Bern, Oksana Birioukova, Pierre Bourgault, Vince Cerundolo, Jean-Francois Dumais, André Gagnon, Louis Girardin, Martin Hélie, Jean-François Panisset, Daniel Pettigrew, Ricardo Ryan, Christine Tassan, Nicolas Tremblay

flame*

© Discreet Logic Inc., 1992-2000. All Rights Reserved.

This documentation contains proprietary information of Discreet Logic Inc. and its affiliates. No part of this documentation may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, recording or otherwise, without the prior written permission of Discreet Logic.

The End-User Software License Agreement which is included with the software program and this manual specifies the permitted and prohibited uses of the software program. Please read the terms and conditions of the Agreement before commencing use of the software program.

The information in this documentation was prepared by Discreet Logic with all reasonable care and is believed to be accurate. However, Discreet Logic and its affiliates shall not assume responsibility for losses or damages resulting from any omissions, inaccuracies, or errors contained herein. This documentation is subject to change, and revisions or new editions may be issued to incorporate such changes.

RESTRICTED RIGHTS LEGEND: The Software {and Documentation} is provided with RESTRICTED RIGHTS. Use, duplication or disclosure by the United States Government or any agency, department or instrumentality thereof is subject to the restrictions set forth in the Commercial Computer Software — Restricted Rights clause at FAR 52.227-19 or the Commercial Computer Software — Licensing clause at NASA FAR Supplement 1852.227-86, as applicable, or in successor provisions, including the limitations set forth in the End-User Software License Agreement included with the software program and this manual. Manufacturer is Discreet Logic Inc., 10 Duke Street, Montreal, Quebec, Canada, H3C 2L7.

fire, flame, flint, inferno, smoke, stone, and wire are registered trademarks of Discreet Logic Inc. and **Discreet, backdraft, combustion, and sparks** are trademarks of Autodesk Inc./Discreet Logic Inc. in the USA and/or other countries. All other brand or product names are trademarks or registered trademarks of their respective owners.

Printed in Canada.

Title:	flame User's Guide
Part No.	900-70250
Publication I.D.:	Release 7.0
Date:	July 2000

toc

Table of Contents

Section 1: Getting Started	1
1 Introduction	3
Summary	3
Welcome to flame	3
flame Documentation	4
What's New in flame 7.0	6
Using This Guide	15
Notation Conventions	16
Getting More Help	16
2 Starting and Exiting flame	17
Summary	17
Starting flame	17
Exiting flame	20
Using the "Kill Process" Command	20
3 Project Management	23
Summary	23
Starting a Work Session	23
Creating a Project	25
Creating a User	27
Saving Space on the UNIX File System	28
Modifying Projects and Users	29
Deleting Projects, Setups, or Clips	30
Deleting Users	31
Loading Setups from Other Projects	32
Loading Preferences from Other Users	32
Switching to a Different User	33

4	Fundamentals	35
	Summary	35
	The Primary Interface	35
	The Desktop	38
	The Menu Area	49
	The File Browser	61
	The Image Window	64
	The Player	77
	The Reference Buffer	81
	System-Wide Hot Keys	83
5	Framestore Setup	85
	Summary	85
	About Framestore Management Systems	86
	Accessing the Framestore Setup Menu	86
	Enabling a New Framestore Setup	95
6	Hot Keys	97
	Summary	97
	About the Hot Key Editor	97
	Hot Key Editor	98
	Keystroke Conventions	99
	Editing Local Hot Keys	100
	Creating New Local Hot Keys	100
	Managing Your Hot Keys	101
	Editing Global and Shared Hot Keys	102
	JLCooper MCS-3800 Interface	102
7	Saving Setups and Preferences	105
	Summary	105
	About Setups and Preferences	105
	Using Directories	106
	Saving, Loading, and Removing Items	107
	Restoring a Backup Copy of a Setup	111
	Setting the Default Preferences	111
	Setup Compatibility between Discreet Products	112
8	System Utilities and Preferences	113
	Summary	113
	The System Menu	113
	The Preferences Menu	118
	Desktop Preferences	118

	Pointer Preferences	124
	Audio Preferences	125
	Using Custom Menus	125
9	Animation	127
	Summary	127
	About Animation	127
	Using the Channel Editor	129
	Editing Animation Curves and Keyframes	134
	Understanding Motion Path Animation	141
	Using Explicit Animation	148
	Using Curve Functions and the Curve Value Field	157
	Changing the Timing of an Animation	163
	Working with Expressions in the Channel Editor	165
10	Monitor Calibration	173
	Summary	173
	About Calibration	174
	Black Level and Picture Level	175
	Colour Temperature	177
	Calibrating Your Monitor	177
	Gamma	179
	Monitor LUTs	182
	Manual Calibration	186
	Saving Your Setup	191
	Customizing the Configuration File	191
	Troubleshooting	192
	Section 2: Managing Clips	193
11	Overview: The Library Menu	195
	Summary	195
	The Library Menu	195
	Accessing The Library Menu	196
12	Clip Libraries	197
	Summary	197
	About Clip Libraries	197
	Framestore Basics	198
	The Clip Library Box	199
	Opening a Clip Library	199
	Saving Items to a Clip Library	201

	Clip Library Structure	203
	Displaying Entries in Clip Libraries.	205
	Selecting Entries in a Clip Library	212
	Loading Entries from a Clip Library	213
	Transferring Clips Between Partitions	214
	Organizing Clips.	217
	Clip Compatibility Between Products.	223
	Using the Network Menu	226
	Hot Keys.	228
13	Clip Input and Output	231
	Summary	231
	About Clip Input and Output.	231
	Inputting Clips from a VTR	232
	Outputting Clips to a VTR	245
	Inputting Clips from a VCR, Camera, or Camcorder	250
	Outputting Clips to a VCR	251
	DDR Input and Output	252
	Clip I/O with the Sony HDCAM Codec	256
	Data Tape Input and Output	263
	Troubleshooting	265
14	EDLs	269
	Summary	269
	About EDLs.	269
	The EDL Menu	270
	Loading an EDL	272
	Editing an EDL	274
	Preferences For EDLs.	285
	Auto-Capturing an EDL	287
	Assembling an EDL	292
	Searching EDLs.	294
	Generating an EDL.	296
	Exporting an EDL.	299
	Importing EDLs through Your Network	302
	Hot Keys.	302
15	24p Mastering	305
	Summary	305
	About 24p Mastering	305
	Creating a 24p Project	306
	Clip Input and Output for 24p Projects	306

	Using EDLs with 24p Projects	310
	Playing 24 fps Clips	317
	PAL Mastering	319
16	Image Import and Export	321
	Summary	321
	About Image Import and Export	321
	File Formats and File Extensions	322
	Importing Images	322
	Using a Lookup Table	330
	Lookup Table Editor	332
	Exporting Images	338
	Removing Images	343
17	OMF Files	345
	Summary	345
	About OMF Files	345
	Exporting a Clip as an OMF File	345
	Importing a Clip from an OMF File	349
18	Archiving	351
	Summary	351
	About Archiving	352
	Archive Compatibility	352
	Accessing the Archive Menu	353
	Archive Structure	354
	Archive Table of Contents	354
	Auto Testing	355
	Estimating Archive Size	355
	Creating an Archive	356
	Opening an Archive	362
	Saving to an Archive	364
	Loading Entries from an Archive	369
	Removing Entries from an Archive	371
	Closing an Archive	372
	Recovering an Archive	372
	Using an ASCII TOC to View an Archive	374
	Deleting Tables of Contents	374
	Archiving Setups	375

Section 3: Editing	377
19 Overview: The Editing Menu	379
Summary	379
Terminology	379
Editing Options.....	380
The Editing Menu.....	380
Gestural Editing	382
Editing Video and Audio Simultaneously	383
The EditReel	386
Soft Edits	389
Editing Timecodes	390
Consolidating and Committing Soft Clips.....	391
The Force Render Command.....	392
20 Basic Editing	395
Summary	395
About Basic Editing	396
Editing Collapsed Clips	396
Cutting Clips	396
Splicing Clips.....	400
Inserting Frames and Clips	403
Replacing Frames and Clips	406
Gestural Splice, Insert and Replace Edits	408
Matching a Soft Edit with Its Source Clips	408
Repeating Frames in a Clip	410
Reversing Frame Order in a Clip	411
Swapping Shots.....	412
Consolidating Clips	414
Deleting Clips	414
Naming and Renaming Clips	416
Moving Clips	416
Copying Clips	418
21 Dissolves	421
Summary	421
About Dissolves	421
The Dissolve Command	422
Creating and Modifying Dissolves in the EditReel	432

22	Trimming Shots	435
	Summary	435
	About Trimming	435
	Trimming, Sliding, and Slipping Shots	438
	Sliding Transitions	439
23	Timewarps	441
	Summary	441
	About Timewarps	441
	Timewarping Audio	441
	Creating Basic Timewarps	442
	Modifying Timewarps in the Timewarp Editor	444
	Modifying Timewarps in the EditReel	447
24	Using Audio	451
	Summary	451
	About Audio	451
	Starting flame with Audio	452
	Using SGI Audio	452
	Sync Issues	461
	Using Discreet Audio	464
	Audio Timewarps	481
	Audio Sparks	483
	Section 4: Transforming Clips	487
25	The Processing Menu	489
	Summary	489
	About Processing	489
	Processing Commands	491
26	Colour Corrector	515
	Summary	515
	Accessing the Colour Corrector	515
	Previewing and Processing the Changes	517
	Colour Sampling	520
	The Basics Menu	522
	Adjusting the Colour Balance	526
	Using the Histogram	528
	Remapping the Colour Values	530
	Animating the Setup Values	532

27	Colour Warper	533
	Summary	533
	About the Colour Warper	533
	Accessing the Colour Warper	534
	Fundamentals	534
	Using the 2D Vectorscope and RGB Viewer	537
	Viewing Clips in the Reference Buffer	541
	Colour Matching	541
	Basics Menu	542
	Warp Gam Menu	545
	Selective Menu	547
	Subsetups	552
	Hot Keys	553
28	Filters	555
	Summary	555
	About Filters	555
	Accessing the Filter Menu	555
	Applying a Filter to a Clip	556
	The Filter Library	557
	Custom Filters	558
29	Batch Processing	563
	Summary	563
	About Batch	563
	Assembling a Process Tree	565
	Manipulating Nodes	572
	Previewing Results	580
	Processing Batch Jobs	586
	Setting Batch Options	589
30	Batch: Node Reference	593
	About Batch Nodes	593
	Action Node	594
	Auto Matte Node	595
	Colour Correct Node	595
	Coloured Frame Node	596
	Colour Warper Node	596
	Degrain and Regrain Nodes	597
	DeInterlace and Interlace Nodes	598
	Export Node	599
	Field Merge Node	600

Filter Node.....	600
Film Compress and Film Expand Nodes	601
Flip Node	601
GMask Node	602
Import Node	602
Keyer Node	603
Library Node	604
Logic Op Node	604
LUT Editor Node	605
Modular Keyer Node.....	609
Monochrome Node	609
Negative Node.....	610
Optics Node.....	610
Output Node.....	611
Posterize Node	611
Quick Composite Node.....	612
Reels Node.....	612
Spark Node	613
31 Formatting Clips	615
Summary	615
About Clip Formats	615
Changing Timecode.....	617
Combining RGB Channels in a Clip	618
Compressing Clips from 30 to 24 fps	618
Dealing Clips	619
Deinterlacing Fields	620
Expanding Clips from 24 to 30 fps.....	621
The Field Dominance Box	622
Interlacing Fields	623
Interleaving Two Clips.....	623
Merging Fields	624
Reversing Field Dominance	625
Separating RGB Channels in a Clip.....	625
Stamping Timecode or Frame Number	626
Section 5: Applying Effects	627
32 Overview: The Effects Menu	629
Summary	629
The Effects Menu	629
Accessing the Effects Menu.....	630

33	The Keyer	631
	Summary	631
	About the Keyer	631
	Accessing the Keyer	632
	Selecting a Keying Technique	633
	Creating a Key by Extracting a Single Colour	633
	Creating a Key by Extracting a Range of Colours	635
	Creating a Key by Setting the Luminance	640
	Adjusting the Luminance of the Key	641
	Adjusting the Key	646
	Removing Background Spill and Colour Correcting Your Clips	649
	Animating Your Key	652
	Processing Your Clips	652
	Keyer Hot Keys	654
34	The Modular Keyer	655
	Summary	655
	About the Modular Keyer	655
	Accessing the Modular Keyer	656
	Using Nodes	676
	Blend Nodes	690
	Animating the Shape Channel	698
	Nodes Placement Table	699
	Hot Keys	700
35	The 3D Keyer	701
	Summary	701
	About the 3D Keyer	701
	Basic Keying Technique	702
	Mastering the RGB Viewer	717
	Animating Key Elements	733
	More Keying Techniques	735
	Setting 3D Keyer Preferences	738
	Hot Keys	738
36	Garbage Masks and the Tracer	741
	Summary	741
	About Garbage Masks	741
	Creating Garbage Masks	742
	Using Garbage Masks from the Modular Keyer and Batch	756
	Using Region of Interest (ROI)	757
	Pulling Keys with the Tracer	758
	Garbage Mask Hot Keys	770

37	Compositing and Optics	773
	Summary	773
	The Compositor	773
	The Quick Composite Module	786
	The Optics Module	787
38	Tracking and Stabilizing	789
	Summary	789
	About Tracking and Stabilizing	789
	How the Stabilizer Works	790
	Working with Trackers	793
	Tracking	798
	Stabilizing	806
	Smoothing Out Motion	811
	Correcting Errors during Analysis	812
	Strategies for Tracking Difficult Shots	813
	Setup Options	819
	Importing and Exporting Data	823
	Copying Shift Channels	823
39	The Warper	825
	Summary	825
	Loading the Clips	825
	Setting Up the Clips	827
	Creating Meshes	828
	Modifying a Mesh	829
	Modifying the Meshes for Live-Action Clips	836
	Warping	840
	Morphing	842
	Setup Options	848
	Warper Hot Keys	851
40	Text	853
	Summary	853
	About Text	853
	Adding Text to Clips	855
	Formatting Text	860
	Creating Text Rolls and Text Crawls	868
	Animating Text	872
	Defining Styles	873
	Working with Fonts	876
	Setting Text Options	879

41	Using Sparks	885
	Summary	885
	About Sparks	885
	Loading a Spark	886
	Using a Spark	886
	Developing Sparks	888
	 Section 6: Power Compositing	 889
42	Action: Overview and Setup Options	891
	Summary	891
	Action Chapters in This Guide	891
	About Action	892
	Action Setup and Processing Options	896
 43	 Action: Layers and Surfaces	 907
	Summary	907
	About Layers and Surfaces	907
	Working with Layers	908
	Using Layers on Surfaces	917
	Working with Surfaces	921
	Reordering Overlapping Surfaces	928
	Using Source Nodes	931
 44	 Action: Camera and the Scene	 939
	Summary	939
	About the Camera and the Scene	939
	Working in the Scene	939
	Using the Camera	945
	Using the Schematic	955
	Animating the Scene Using the Channel Editor	965
 45	 Action: Light and Shadows	 967
	Summary	967
	About Light Sources	967
	The Light Menu	969
	Selective Lighting	971
	Working with Surface Lighting	974
	Adding Shadows	978

46	Action: 3D Geometry	983
	Summary	983
	About 3D Geometry	983
	Importing 3D Models	983
	Creating 3D Text	985
	Working with 3D Geometry	989
	Working with Textures	992
	Displacement Mapping	999
	Extended Bicubics	1002
	3D Deformations	1011
 47	 Action: The Particle System	 1015
	Summary	1015
	About Action's Particle System	1015
	Generating Particles	1017
	Customizing the Particle Stream	1021
	Working with Particle Manipulators	1029
	Working with Particle Bouncers	1047
	Exploding Objects	1051
	Particles on Particles	1053
	Sample Particle Setups	1056
 	 Section 7: Retouching Clips	 1057
48	Paint: Overview	1059
	Summary	1059
	About Paint	1060
	Accessing the Paint Menu	1060
	Paint Basics	1062
	Selecting Colours	1063
	The Flipbook	1065
 49	 Paint: Painting on the Canvas	 1069
	Summary	1069
	About Canvas Mode	1069
	Selecting a Brush	1070
	Rolling the Image	1071
	Painting Straight Lines	1072
	Painting with Geometry	1072

50	Paint: Setting Brush Attributes and Modes	1075
	Summary	1075
	About Brush Attributes and Modes	1075
	Brush Attributes	1076
	Brush Modes	1080
	Preferences Affecting Brushes	1082
51	Paint: Using Filters and Special Effects Media	1083
	Summary	1083
	About Filters	1084
	Using Special Effects Media	1085
52	Paint: Using Wipe, Fill, and AutoPaint	1095
	Summary	1095
	The Wipe Command	1095
	The Fill Command	1097
	AutoPaint	1101
53	Paint: Using Mattes	1109
	Summary	1109
	About the Matte	1109
	Viewing the Matte	1110
	Creating or Modifying a Matte	1111
	Using the Auto Matte Command	1113
54	Paint: Using Graphics	1115
	Summary	1115
	Accessing the Graphics Menu	1116
	Creating Graphics in Paint	1116
	Using the Object Tools	1117
	Selecting Objects	1126
	Editing Objects	1127
	Setting the Object Resolution and Display	1128
	Setting the Object Attribute	1131
	Changing the Object's Appearance	1132
	Changing the Size or Position of an Object	1134
	Changing the Shape of an Object	1136
	Converting Graphics to Autopaint Strokes	1138
	Animating Graphics	1139
	Saving and Restoring Objects	1143
	Tacking Down Objects	1144

55	Paint: Cut and Paste	1147
	Summary	1147
	Accessing the Cut and Paste Menu	1147
	Creating a Cutout	1148
	Using the Cutout Commands	1149
	Increasing Editing Speed	1151
	Using the Paste Mode Options	1152
	Colour Correcting Cutouts	1156
	Animating Cutouts	1156
	Loading and Saving Cutouts	1156
56	Paint: Creating, Loading, and Saving Setups	1157
	Summary	1157
	About Paint Setups	1157
	About Saving Setups	1158
	Loading Setups	1159
	Removing Setups	1160
	Creating and Saving Brush Setups	1160
	Creating a Custom Brush	1161
	Using Colour Correction Setups	1166
57	Utilities	1167
	Summary	1167
	About Utilities	1167
	Opening an IRIX shell	1168
	Requirements	1168
	Using the Utilities	1169
	Parameters	1169
	list_framestore	1171
	to_framestore and from_framestore	1173
	Restrictions	1174
	imcopy	1175
A	Glossary	1181
	Index	1197

