RAYZ MANUAL

for Version 2.2

Created April 2002
Silicon Grail Corporation
736 Seward Street
Hollywood, CA 90038 USA

Documentation for Silicon Grail RAYZ 2.2 multi-platform compositing software.

This document was created at Silicon Grail Corporation, 736 Seward Street, Hollywood, CA 90038, USA.

CONTENTS

PART One

GETTING STARTED

Part I provides an introduction to RAYZ. It explains what RAYZ is, and how to install it and perform related setup tasks. Chapter 1 also tells you where to locate RAYZ tutorials.

IN PART I: GETTING STARTED

INTRODUCTION

This manual documents the features and use of RAYZ 2.2, a 2D compositing software application that runs native on IRIX, Linux, and Windows operating systems.

How to Use This Manual

This manual does not presume to be a compositing textbook, and readers are assumed to be experienced digital compositors. However, the backgrounds of compositing professionals in today's effects facilities can vary greatly. For example, your previous experience may be in

- traditional optical composition for film,
- video editing,
- computer graphics, or
- computer programming.

Each of these areas provides a different focus of expertise, as well as a different jargon. (In fact, the terminology relating to a number of 2D and 3D techniques varies from facility to facility.)

Given this situation, this manual attempts to strike a balance between explaining too little and too much about the topics it needs to address. For example, a compositor with a film background may understand the intricacies of log-to-linear conversion for the Cineon film format, while a compositor with a C programming background may have no trouble writing complex expressions.

No matter what your background may be, we recommend that you at least skim the chapters of this manual that cover topics in which you are already experienced. For example, even if you are familiar with Cineon conversion issues, it will be profitable to find out how RAYZ handles this conversion, what options for Cineon conversion are available, and exactly what we mean by the Cineon-related terms we use.

The Fastest Way to Find Information

In addition to using the Index, you should also take a moment now to familiarize yourself with the structure of this manual, so that you can find the exact information you need when you need it.

The chapters in part 2 cover basic concepts relating to building nodes into a network that defines a shot and using the tools common to most operations. They also explain how to import and render images, and how to customize the RAYZ interface and set up preferences to fit the way you work and the type of shot you are building.

This knowledge is assumed in the node reference chapters in part 3, which are designed to be consulted for specific and detailed information about the use of individual node parameters.

The appendixes in part 4 provide technical details about specific topics covered in the main chapters. This information falls into two categories, information that is needed only rarely or information that is common to many node operations and therefore has been gathered in one place for frequent, easy reference.

Descriptor Conventions Used

This manual describes keyboard and mouse actions in terms of keys pressed and mouse buttons clicked. “Click” is always synonymous with “left-click,” that is, with pressing the left mouse button. The shorthand terms “middle-click” and “right-click” are often used to refer to the action of clicking the middle mouse button and right mouse button respectively.

The term “drag” refers to holding down the left mouse button as you move the mouse; similarly, “middle-drag” and “right-drag” refer to dragging with the middle and right mouse buttons. “Hold” refers to holding down the left mouse button, usually to access a menu list, while “middle-hold” and “right-hold” refer to holding down the middle and right mouse buttons.

“Hotkey” is the term used to identify keyboard equivalents, also known as keyboard shortcuts. Hotkeys can be pressed to issue commands in lieu of using the mouse to click buttons or pull down menus. Hotkeys may or may not include modifier keys.

Modifier keys—the Control, Shift, and Alt keys on the keyboard—are held down as another key is pressed or as a mouse button is clicked or held to modify the action. Commands that use modifier keys are often referred to in abbreviated form, such as “Ctrl-click” for holding down the Control key as the left mouse button is clicked.

Terms That Have Specific Meanings in RAYZ

The terms “size” and “resolution” refer to spatial resolution; that is, to width and height, usually expressed in pixel units.

The level of color resolution is referred to as bit depth, and bit depth is always per channel of an image. RAYZ operates in 8-bit (per channel), 16-bit (per channel), and floating point (32-bit per channel) bit depth. The image data referred to is in linear space, unless log is explicitly stated—as in “10-bit log data”—or unless the term “Cineon” is used.

The terms “float” and “floating point” in reference to colorspace indicate the highest level of precision in RAYZ, equivalent to 32-bits per channel. These terms are also often used interchangeably with “fractional” or “decimal” to specify floating point as opposed to integer values. For example, white would be expressed as 65535 in 16-bit and as 1.0 in floating point units.

Other Sources of Information About RAYZ

Online Help System

The entire manual is available in HTML format from the RAYZ Help menu. In addition, if you turn on the Help Tags option in the Help menu, popup help text appears when you mouse over objects in the interface.

Printable Manual

The RAYZ Manual is also available on the RAYZ distribution CD-ROM in Adobe Acrobat pdf format, suitable for printing. The file is called RAYZ2.2_Manual.pdf. You may prefer to print a relevant section of the manual and refer to it as you work in the RAYZ interface, rather than switching back and forth between RAYZ and your HTML browser.

Tutorials

A series of tutorials are available that cover the basics you need to know to start creating shots in RAYZ. The tutorials are in Adobe Acrobat format, suitable for printing, so that you can read the hard copy as you perform each step in RAYZ. Sample project files and images are also provided.

It is highly recommended that you start with these tutorials, which are included in the RAYZ release distribution.

TRAINING

RAYZ training may be provided periodically by Silicon Grail in certain circumstances. In addition, schools that specialize in teaching graphics applications to effects artists also offer RAYZ training. Contact Silicon Grail for more information.

CONTACTING SILICON GRAIL

You can contact Silicon Grail by phone, fax, email, or mail:

Silicon Grail
736 Seward Street
Hollywood, CA 90038

Phone: 323-871-9100
Fax: 323-871-9199

Email: support@sgrail.com

WHAT'S NEW IN VERSION 2.2

Version 2.2 of RAYZ includes new nodes and new interface features, as described below. Please also refer to the changes file (release notes) distributed with the release, which includes the most up-to-date information.

NEW NODES

RAYZ 2.2 includes the a number of new nodes. For more information, refer to the following node descriptions:

"BlurXY Node" (ch. 19, p. 349)

"Channel Split" (ch. 20, p. 399)

"Checker Node" (ch. 14, p. 165)

"Clamp Node" (ch. 16, p. 250)

"Despill Node" (ch. 15, p. 206)

"File Group Node" (ch. 14, p. 173) (see also "Creating File Groups" in chapter 22, p. 434)

"Grid Node" (ch. 14, p. 176)

"Tint Node" (ch. 16, p. 274)

"Switch Node" (ch. 21, p. 419)

Changes to the Interface

In the Worksheet

A Navigator (radar view) has been added to the Worksheet (see "Using the Navigator" in chapter 5, p. 43), along with a Toolbar that puts many commands in a convenient button strip.

The new Toolbar's "Display Menu" (ch. 5, p. 44) provides new display options for the Worksheet: flowlines display channel info, nodes indicate whether they are cloned or animated, and you can display link lines between clones and nodes that reference other node parameters.

It is now easier to manage Custom node menu items, as described in "Managing Custom Node Menu Entries" in chapter 5, p. 59.

In the Image Viewer

Additional options have been added to the Display Conversion parameters, including separate RGB and Other Gamma controls and the ability to import LUT files. See the section on "Display Conversion" in chapter 6, p. 72.

Full screen, or "blackout" mode has also been added to the Image Viewer (just press the n hotkey), along with two new overlays: Field Chart and 1.85 Mask. See "Border Display Overlays" in chapter 6, p. 77.

In the Node Panel

Now you can easily invert any mask channel after it is connected to the node (see "Using Mask Inputs" in chapter 7, p. 102). The "X-Parm" (ch. 7, p. 90) button has been added to every Node Panel so that you can add extra (spare) parameters. And node presets can now be removed (and renamed) more easily, as described in "Managing Presets" (ch. 7, p. 89).

Dynamic parameters, such as layer entries in Multi-comp, no longer have explicit buttons for moving or deleting a layer. The way you rename a layer has also changed. These changes are described in the section on "Dynamic Parameter Groups" in chapter 7, p. 93.

Other Changes and New Features

Command line scripting (see "Command-Line Execution of Scripting Commands" in appendix D, p. 457) and retiming (see "Retiming Sequences from the Command Line" in chapter 12, p. 145) options have been added.

New Photoshop import options are available, as described in "Import File…" in chapter 9, p. 117. A "Revert…" (ch. 9, p. 118) command has also been added to the RAYZ File menu. And you can now save RAYZ files with Bzip2 compression, as described in "Saving Compressed RAYZ Files" (ch. 9, p. 118).

Motion blur controls have been added to the Transform, Match Move, and Stabilize nodes, as described in "Adding Motion Blur to Transformations" (ch. 17, p. 280).

Roto and Morph splines can now be copied and pasted using the new Layer Actions menu. The Layer Actions menu, which applies to all nodes that generate dynamic parameters, is described in "Copying, Pasting, and Deleting Layers" in chapter 7, p. 94.

The "Sequence Node" (ch. 21, p. 415) now reports the new output sequence range for each input, and the "Convolve Node" (ch. 19, p. 352) now has an editable matrix of cells for creating custom kernels, in addition to the library of preset kernels. The "Turbulence Node" (ch. 14, p. 199) now has a Scale parameter.

Group nodes now have edit panels for parameters and inputs/outputs. See *Chapter 22: Creating Group Nodes* (p. 427) for more information.

chapter 2

INSTALLATION

This chapter describes the hardware and operating systems on which the RAYZ software operates and provides instructions for installing and keying RAYZ on your system. To install RAYZ, you will need to

- install the application,
- install the license manager,
- generate your host code and send it to Silicon Grail,
- put the license file received from Silicon Grail in the appropriate location and instruct the license server to check for it.

You can install a copy of the RAYZ *application* on as many machines as you like, but you should only install the *license manager* once, on one machine, which will serve licenses for all machines running RAYZ.

NOTE:

Be sure to read INSTALL.doc and any other readme documents included with the installation files for the most current information.

IN THIS CHAPTER

SYSTEM REQUIREMENTS

The following requirements and recommendations apply to all systems:

MEMORY: RAYZ 2.2 requires a minimum of 64MB of RAM, although 128MB or more is recommended.

DISK SPACE: The RAYZ application files take up approximately 40MB of hard disk space. In addition, hundreds of additional megabytes of temporary disk space could be required while the application is running. See "RAYZ_TMPDIR" on p. 16 for information about specifying the directory RAYZ should use for caching temporary files.

GRAPHICS: A 24-bit or 32-bit color graphics accelerator is required, with a recommended resolution of 1280 x 1024 or higher. (See also "Recommended Graphics Cards" below.)

DISPLAY: A large (19- or 21-inch) monitor is recommended. To take advantage of dual monitors, see "RAYZ_SPLIT_SCREEN" on p. 15.

RECOMMENDED GRAPHICS CARDS

Some recommended cards for Linux or Windows systems include nVidia GeForce–based cards (with nVidia hardware-accelerated drivers if you are running Linux) and Matrox G400 or G450 cards.

RAYZ does *not* require OpenGL-capable graphics cards.

OPERATING SYSTEMS

RAYZ will run on IRIX, Intel and Alpha Linux, and Intel Windows configurations that meet the criteria described in the following paragraphs for each system. RAYZ runs on single or multiple processors.

IRIX

RAYZ will run on any Silicon Graphics workstation with at least an R4000 CPU running the IRIX 6.X operating system.

In addition, a 64-bit version of RAYZ can be installed to take advantage of the increased memory addressing available on machines with an R8000 or faster processor that are running the 64-bit version of IRIX 6.5 (or higher).

LINUX

RAYZ will run on Red Hat 6.2 (or higher) or Debian 2.2 (or higher) distributions of the Linux operating system on workstations with Intel Pentium Pro (or faster) or Alpha processors.

Other distributions of Linux with a glibc version 2.1 or later should work also, but they have not been tested by Silicon Grail and we cannot vouch for them.

Windows

RAYZ will run on Microsoft Windows NT, Windows 2000, and Windows 98 operating systems on workstations with an Intel Pentium Pro (or faster) processor.

IRIX Installation

There are two versions of RAYZ available for IRIX: 32-bit and 64-bit. The 64-bit version of RAYZ is designed to take advantage of the increased memory that can be addressed on 64-bit IRIX systems. (A 32-bit processor can only address about 2GB, which means that you can run out of memory—even virtual memory—when doing large comps.)

An IRIX system is not considered 64-bit unless it has the 64-bit version of the IRIX 6.5 (or newer) operating system *and* an R8000 or faster processor.

The "regular" 32-bit version of RAYZ must be installed on all SGIs that are not 64-bit; the 32-bit or 64-bit version of RAYZ can be installed on 64-bit machines.

Check 64-Bit Status

The following instructions should be used to determine whether your machine is 64-bit. If you already know which version of RAYZ you are going to install, however, you can skip to the installation instructions.

1. Log in to your machine and type `uname` at the prompt. The system will return either "IRIX" or "IRIX64."
2. If the system returns "IRIX," skip steps 3 and 4 and install the 32-bit version. If the system returns "IRIX64," follow step 3.
3. Type `hinv` at the prompt to get a hardware inventory for the machine.
4. If the machine has an R8000 or faster processor, you can install either the 64-bit or 32-bit version; if not, you must install the 32-bit version.

Install the Application

If installing the software from the RAYZ distribution CD, insert the CD into your local CD-ROM drive. If you have downloaded the software from the Silicon Grail website, copy the tar file to /tmp.

1. Log in as root.
2. Type the following commands:

```
mkdir /usr/grail
cd /usr/grail
gzcat /tmp/rayz.irix.tar.gz | tar xvf -
cd rayz2.2
./Install
```

Install the License Manager

Be sure to install the license manager only once, on the machine that will serve licenses for all computers running RAYZ.

1. Log in as root to the machine that will be used as the license server.
2. If the directory **/usr/lib/grail** exists, remove it and its contents.
3. If the file **/etc/init.d/grail** exists, remove it.
4. Type the following commands:

```
cd /usr/lib
tar xvf license_manager.irix.tar
cd grail
./Install
```

Key the Software

Follow these steps to get a key from Silicon Grail.

1. Type the following command to generate a host (license server) code:

   ```
   /usr/lib/grail/hostinfo
   ```

2. Contact Silicon Grail support and give us the resulting host code string, along with name of the license server machine. The string will take this form: AA-BB-CC-DD-EE-FF-2.2. You can reach us via

   phone: (323) 871-9100
   fax: (323) 871-9199
   email: key@sgrail.com

   We will email a file to you called "license.lic."

3. Place the license.lic file in the following directory:

   ```
   /usr/lib/grail
   ```

   You can name the file anything you like, as long as it ends in the ".lic" extension. Multiple .lic files can be placed in this directory; all of them will be searched for a valid license.

4. Type the following command to force the license server to check for license files:

   ```
   /usr/lib/grail/grailadmin -r localhost
   ```

RAYZ should now be ready to run. To verify the procedure, log in as a user and type `rayz` from your home directory to launch the application.

Linux Installation (Intel and Alpha)

If installing the software from the RAYZ distribution CD, insert the CD into your local CD-ROM drive. If you have downloaded the software from the Silicon Grail website, copy the file to /tmp.

Install the Application

1. Log in as root.
2. Type the following command, for either Intel or Alpha, to unpack and install the RAYZ application.

   INTEL:

   ```
   rpm -U rayz.linux.intel.rpm
   ```

   ALPHA:

   ```
   rpm.-U rayz.linux.alpha.rpm
   ```

Install the License Manager

Be sure to install the license manager only once, on the machine that will serve licenses for all computers running RAYZ.

1. Log in as root to the machine that will be used as the license server.
2. If the directory **/usr/lib/grail** exists, remove it and its contents.
3. If the file **/etc/init.d/grail** exists, remove it.
4. Type the following commands, for either Intel or Alpha.

   INTEL:

   ```
   cd /usr/lib
   tar xvf license_manager.linux.intel.tar
   cd grail
   ./Install
   ```

   ALPHA:

   ```
   cd /usr/lib
   tar xvf license_manager.linux.alpha.tar
   cd grail
   ./Install
   ```

Key the Software

Follow these steps to get a key from Silicon Grail.

1. Type the following command to generate a host (license server) code:

   ```
   /usr/lib/grail/hostinfo
   ```

2. Contact Silicon Grail support and give us the resulting host code string, along with name of the license server machine. The string will take this form: AA-BB-CC-DD-EE-FF-2.2. You can reach us via

   phone: (323) 871-9100
   fax: (323) 871-9199
   email: key@sgrail.com

   We will email a file to you called “license.lic.”

3. Place the license.lic file in the following directory:

   `/usr/lib/grail`

   You can name the file anything you like, as long as it ends in the ".lic" extension. Multiple .lic files can be placed in this directory; all of them will be searched for a valid license.

4. Type the following command to force the license server to check for license files:

   `/usr/lib/grail/grailadmin -r localhost`

RAYZ should now be ready to run. To verify the procedure, log in as a user and type `rayz` from your home directory to launch the application.

Windows Installation (NT/2000)

Locate the executable files for the application (rayz.nt.exe) and the license manager (license_manager.windows.exe), either on the RAYZ distribution CD or in the folder to which you downloaded them from the Silicon Grail website. Then follow these instructions to install and key RAYZ.

Install the Software

1. Log in as Administrator or a user with administrator privileges.
2. Double-click on "rayz.nt.exe" to launch the installation program.

Key the Software

Follow these steps to get a key from Silicon Grail.

1. Double-click on the file "license_manager.windows.exe" to launch the license manager.
2. Answer "Yes" to the question about obtaining the host code. The host code will be displayed as a string with the form AA-BB-CC-DD-EE-FF-2.2.
3. Write down the host code string, being sure to double-check it for accuracy.
4. Contact Silicon Grail support and give us the host code string you recorded in the previous step, along with name of the license server machine. You can reach us via

   phone: (323) 871-9100
   fax: (323) 871-9199
   email: key@sgrail.com

   We will email a file to you called "license.lic."

5. Place the license.lic file in the following folder:

   ```
   C:\Program Files\Silicon Grail\Grail LM\Keys
   ```

   You can name the file anything you like, as long as it ends in the ".lic" extension. Multiple .lic files can be placed in this folder; all of them will be searched for a valid license.

6. In the Start menu, run the program Silicon Grail > License Manager > Refresh License Server.

RAYZ should now be ready to run. To verify the procedure, log in as a user and double-click the RAYZ application icon to launch the program.

Other Setup Options

This section covers two RAYZ environment variables. Be sure to review *Chapter 13: Setting Preferences* (p. 147), however, for more information about setting up your work environment. In particular, a number of environment variables are described in the section on "File Paths" (ch. 13, p. 148), including paths to RAYZ layout, preset, plugin, and custom node files, as well as the paths to search for the help browser used by the online manual and fonts used by the Text node.

The first environment variable described here, RAYZ_SPLIT_SCREEN, lets you set up RAYZ to take advantage of dual monitors. It can only be set from the command line (or Properties panel, in the case of Windows).

The other environment variable, RAYZ_TMPDIR, can also be set in the RAYZ Preferences panel, but it is included here because it you will probably want to set as soon as you install RAYZ, if at all.

Setting Environment Variables

To set an environment variable from the command line on **IRIX** and **Linux** systems, type `setenv` followed by the variable name and argument.

On **Windows NT**, right-click on the My Computer icon and select Properties from the menu. In the Properties panel, select the Environment tab and type the name of the variable in the variable field and the argument in the value field.

RAYZ_SPLIT_SCREEN

To take advantage of a workstation that incorporates two (or more) display devices, set the RAYZ_SPLIT_SCREEN environment variable to the number of monitors in your system.

For example, on a Linux box with two display devices, type "2" as the argument:

```
setenv RAYZ_SPLIT_SCREEN 2
```

RAYZ_TMPDIR

RAYZ may require a substantial amount of temporary disk space while running in order to cache image data in temporary files and directories. Each frame of film at 2K resolution, for example, may require 2 to 8 MB, depending on the file format.

RAYZ_TMPDIR tells Rayz where to store temporary files and directories. By default, RAYZ will put temp files in /usr/tmp.

We suggest you set RAYZ_TMPDIR to a file system path that has a substantial amount of disk space available. (You may want to consult with your system administrator for the best way of handling this issue.)

To set this variable on an IRIX machine, for example, type the following, replacing “[pathname]” with the actual file path to use:

```
setenv RAYZ_TMPDIR /[pathname]
```

chapter 3

KEY CONCEPTS

RAYZ is a procedural, node-based compositor in which the completed shot consists of a network of nodes, where the nodes represent image operations and the connections among them represent the flow of the image data through the network.

IN THIS CHAPTER

Creating a Shot in RAYZ

A RAYZ project file starts with one or more source nodes, such as the Image In node. The Image In node points to the location (on a local or networked disk drive or other storage device) of a sequence of image files you will be modifying, such as the background plates for the scene or a foreground element rendered from a 3D modeling program. The source image data is loaded into memory for manipulation in RAYZ.

The source imagery is modified and combined using various nodes, which are categorized by general function: color correction, matte generation, composite operations, and so forth.

The nodes are connected in a flowchart. As you build this flowchart, you are creating a description of the shot. You can think of the nodes as the building blocks, which you can add, delete, and rearrange as needed to produce the appropriate result. The source images begin streams of data that flow through this network of nodes, being modified by each node in turn.

The RAYZ interface is designed to let you focus on the images instead of the flowchart, however, if you choose. To a great extent, the flowchart of nodes builds itself as you work on the images, and it is always available to be reviewed and edited in the Worksheet.

Whatever operations you perform on this imagery, finally you will want to render new image files of the completed shot to disk. You can render any node image by connecting it to an Image Out node, where you specify the name, location, and format of the files and then render them whenever you choose.

The dataflow model also has the advantage of being "self-documenting," which simply means that the structure of the network itself shows you how the shot has been constructed, making it easier to pinpoint areas to be changed or fine-tuned—by the shot's creator or by another artist.

Controlling Data Display: Dynamic Focus

The Worksheet is used not only to build the flowchart of nodes, but also to control which node data appears in other parts of the interface. To see how an image looks in any particular node, and to modify the node parameters, you select the node in the Worksheet. The node image is automatically displayed in the Image Viewer and the node parameters show up in the Node Panel.

This concept is called dynamic focus and it is basic to controlling what image data is displayed where and when in the interface. For a detailed description of how to use it, and how to override it when you wish, refer to "Dynamic Focus" in chapter 4 (p. 27).

How RAYZ Handles Images

RAYZ will use non-adjacent concatenation of operations wherever feasible to render imagery, and employ the fastest possible filtering algorithms that do not compromise image quality.

The resolution of the output is only determined at render time. Whatever output frame size you specify, RAYZ is storing the image data in an infinite *x,y* coordinate space so that data is not lost when it is transformed out of frame. When you render an image, all pixels that fall within the area you specify at that time are rendered.

Modifying Images

Once you import imagery into RAYZ, you can modify it by selecting operations directly in the Image Viewer. (Hold down the Space bar for a menu of all available node operations.) Each time you select a filter, transform, or other node, not only is the image updated but also the Worksheet area, where the corresponding node is inserted into the network.

The parameters that can be set for the various node operations appear automatically in the Node Panel for the currently selected node image. The individual parameters are organized to show the most common, convenient options by default and to hide the rest, but every option can be displayed with a mouse click, as when you need control of individual image channels, advanced filtering options, etc.

Node parameter values can be displayed in pixel units, or in floating point units, as you choose. However, RAYZ always stores the values at floating point precision. This means that you can switch back and forth between proxy and full size images, or clone a node and connect the clones to different size images, without having to make any adjustments.

Image Channels

You can view and modify an image or any individual channel in an image. Whenever appropriate, parameters offer a master control that operates on all channels, with the option of accessing the individual channel controls to adjust them separately.

You can carry the opacity data (the matte) with the RGB image as an integrated alpha channel and still control which channel a node affects; you do not have to send a separate alpha channel image through the network unless you choose.

In fact, most node parameters enable you to control five image channels: the RGB channels, the alpha channel, and a fifth channel (labeled “O” for Other), such as that used to carry z-depth or other image information.

Combining Disparate Images

RAYZ lets you combine images with different resolutions and bit depths without having to explicitly specify which image is scaled, translated, promoted or demoted, and so forth.

For images of different sizes, pixels are aligned according to their *x,y* coordinates in relation to a 0,0 point, which means that images are aligned to the lower left corner unless you specify an offset or apply some other type of spatial transform.

For images of different bit depths, the lower is linearly remapped to the higher. RAYZ operates in 8-bit, 16-bit, or floating point (32-bit) color depth per channel, as you choose.

Cineon Conversion

By default, any Cineon 10-bit log images you import are automatically remapped to 16-bit linear, using Kodak specifications. But you always have the option to modify the default conversion values or to use the unconverted log data.

RAYZ will automatically choose an appropriate LUT or other method for display conversion as you view different types of imagery. You can override the default setting for any image and select a display conversion method manually (such as Cineview emulation), and you can add custom LUTs to the list of options.

Customizing the Interface

By design the RAYZ interface is dynamic and flexible. You can add, duplicate, and delete the main components—the Image Viewer, the Worksheet, etc.—as well as resize and reposition them. The components can all be docked together in one window, or individual components can float in separate windows.

The interface can be adjusted on the fly, but you can also save and reuse any configuration, which is called a layout. The individual parameters and tools are just as flexible, and you can set preferences to control colors, default values, and hotkey assignments.

Be sure to read *Chapter 4: Overview of the Interface* (p. 23), for a thorough introduction to the RAYZ interface and how to customize it for your work habits and projects.

PART

TWO

WORKING IN RAYZ

Part II describes the RAYZ interface and how to use the various components. It covers the preferences and project settings in RAYZ, and it explains how to import and render images and the options available for editing image sequences.

IN PART II: WORKING IN RAYZ

chapter 4

OVERVIEW OF THE INTERFACE

The RAYZ interface consists of docked windows, which can be thought of as panes in the RAYZ application window.

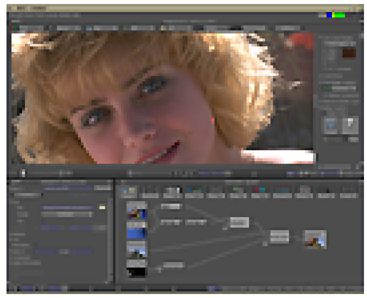
Six types of pane are available, each of which provides a different view of the data and a different set of controls, based on the task for which it is designed: the Worksheet, Image Viewer, Node Panel, Clip Editor, Curve Editor, and Render Control.

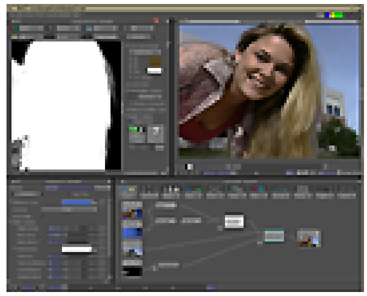
The Worksheet, for example, contains the network of nodes representing the flow of images and operations, and the Node Panel is where you adjust parameter values for each node operation.

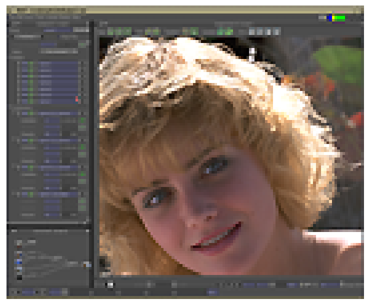
You can create more than one pane, or "data view," of a single type. To display the parameter settings for two separate nodes simultaneously, for example, you would use two Node Panel views. Any view pane can be resized, repositioned, duplicated, or deleted.

The configuration of the RAYZ interface—the way the views and other interface elements are arranged in the application window—is known as a layout, and any layout can be saved for reuse. This enables you to switch instantly from one customized layout to another.

IN THIS CHAPTER







The View Types

The paragraphs below describe the purpose of each type of data view. For more information about using them, refer to the chapter referenced in the following descriptions and to "Working with Views" on p. 27. For information about creating, deleting, and rearranging view panes in the layout, see "Manipulating View Panes" on p. 31.

Worksheet: Where you create and connect nodes to build the dataflow (flowchart of node operations) that describes the shot. See also *Chapter 5: Nodes in the Worksheet* (p. 41).

Image Viewer: Where you see the image as it appears in the currently selected node. See also *Chapter 6: Using the Image Viewer* (p. 63).

Node Panel: Where you access the parameter controls for a node. See also *Chapter 7: Using the Node Panel* (p. 87).

Curve Editor: Where you manipulate curves to adjust and animate parameter values. See also *Chapter 8: Using the Curve Editor* (p. 105).

Clip Editor: Where you re-sequence source imagery interactively using filmstrips arranged in a timeline. See also "Using the Clip Editor" in chapter 12 (p. 136).

Render Control: Where you can initiate rendering of the imagery in any or all Image Out nodes in the RAYZ project file. See also *Chapter 11: Rendering Images* (p. 129).

4.1 The default layout includes an Image Viewer, (top) a Node Panel (lower left), and a Worksheet (lower right).

OTHER TOOLS IN THE LAYOUT

In addition to the data views listed above, the RAYZ interface provides several general purpose tools: the Main Menu strip, Time Scooter, Process strip, and Status Bar.

MAIN MENU STRIP

This strip provides menus for commands used to access and modify RAYZ project files, modify the current layout, render images to disk, and access help files.

4.2 Main Menus.

Each menu is described in detail in *Chapter 9: Main Menus* (p. 115), however, three menus that affect the interface should be noted here:

- **Views,** which can be used to add any type to view pane to the current layout,
- **Tools,** which is used to turn the display of the Time Scooter, Process Strip, and Status Bar on and off in the interface, and
- **Layouts,** which is used to save and access custom layouts as well as to restore the factory default layout.

The Main Menu strip is located by default in the upper left corner of the RAYZ interface, although you can reposition it along any outside edge by dragging the grabber handle located at the left edge of the strip. An outline representing the menu strip will appear under the cursor, and when you release the mouse button the strip will drop into the new location.

TIME SCOOTER

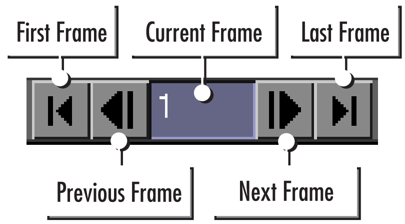
4.3 Time Scooter.

The Time Scooter is used to navigate through time. The current frame specified in the Time Scooter is the frame displayed in the Image Viewer, Node Panel, and Curve Editor. When you change frames, these views will update accordingly. For example, the Image Viewer will display the new frame image.

HOTKEYS:

The frame navigation hotkeys are the **Comma** (,) and **Period** (.) keys. They can be used in any view to go to the previous or next frame.

The Shift modifier key can be used with these hotkeys to jump directly to the first frame (Shift-,) or the last frame (Shift-.)



4.4 Time Scooter frame navigation controls.

To change the current frame, drag the slider bar back and forth in the time scale, click the navigation arrows, use the hotkeys, or enter a specific frame number in the Current Frame field.

Global Time Range

You can change the default frame range (1–100) of the Time Scooter by typing new values in the range fields located at each end of the time scale. See *Fig. 4.3*. Typically, the range is set to match the total length of the shot.

NOTE:

The current range set in the Time Scooter is the global time, which RAYZ uses to set the default range of any source nodes you create (Color, Gradient, etc.) except for the Image In node, which is set to the range of the imported sequence regardless of the Time Scooter settings.

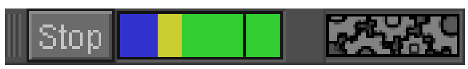
Time Display Style

The Time Scooter uses frames by default, but you can specify a different type of unit in the Project Settings panel, which is accessed from the RAYZ Edit menu. You can select Frames, Seconds, Edge Code, or Time Code from the Time Display Style preference menu. See also "Editing Project Settings" in chapter 13, p. 156.

Process Strip

The Process strip is located by default in the upper right corner of the interface. Use the Stop button in the strip if you want to cancel the current process, or use the equivalent **hotkey: Esc**.

The current memory usage is displayed in the color-coded bar between the Stop button and the gears icon. The gears animate whenever RAYZ is actively processing data.



4.5 The Process strip provides feedback on current memory usage: blue represents memory used by RAYZ; yellow represents memory used by the operating system and any other applications that are running; green represents the remaining memory available. (The black line dividing the green bar separates physical RAM, on the left, from swap space, right.)

Status Bar

The Status Bar displays information about the object or parameter currently selected or pointed to with the cursor. If you position the cursor over an image in a Viewer, for example, the Status Bar tells you which image it is, what size it is, how many channels it has, whether the image is log or linear, and the bit depth.

If you position the cursor over a parameter in the Node Panel, on the other hand, the Status Bar will explain the purpose of that parameter.

4.6 Point to a node, connection, parameter, or image to display information about it in the Status Bar.

Working with Views

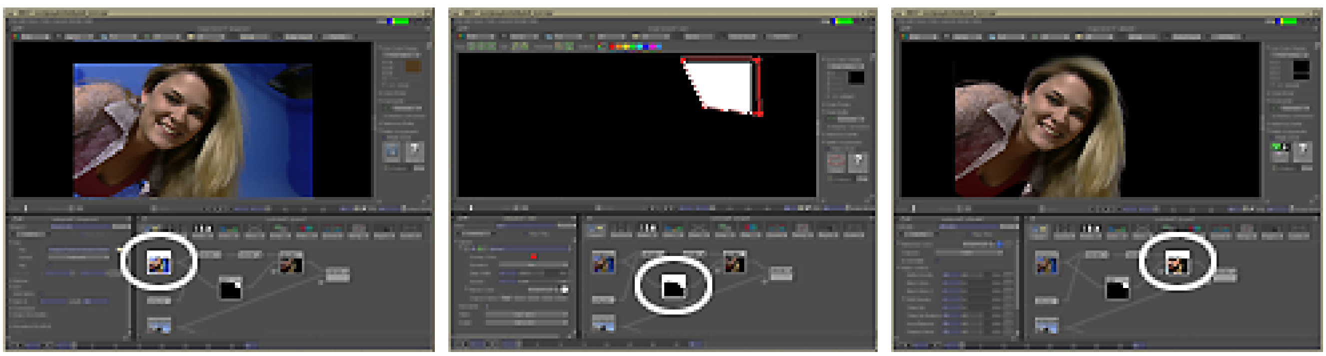
This section covers integral features of data views, such as the mechanisms used to specify the content of a view and how it is displayed.

Dynamic Focus

An important concept to understand when working in RAYZ is that three of the view types, by default, are dynamic—they change their focus as you select different nodes. The views that follow the current selection are:

- Image Viewer
- Node Panel
- Curve Editor

The Worksheet is used to control the focus of these views; that is, to specify what node data is displayed in the view. By default, the Image Viewer, Node Panel, and Curve Editor all display the data that is relevant to the node currently selected in the Worksheet, and when you select a different node, they automatically update their content to show the data relevant to the new node selection.



4.7 The Image Viewer display changes as different nodes are selected in the Worksheet: Image In node (left), Roto node (middle), Ultimatte node (right).

NOTE:

The title bar of every Node Panel, Image Viewer, and Curve Editor always displays the name of the individual node data it is displaying.

PINNING A VIEW TO A NODE

You can override dynamic focus at any time by "pinning" a view to a specific node. When you pin a view to a node, that view will not change focus when you select a different node in the Worksheet.

4.8 Title bar of Node Panel displaying "blur1" node data: it is pinned to the node (top), and free to follow the current Worksheet selection (bottom).

This change is specific to the individual view pane; if you have two Node Panels in your layout, for example, the other Node Panel will continue to follow the node selection in the Worksheet unless you choose to pin it to a node also.

NOTE:

The dynamic focus concept is not relevant to the Worksheet, as it contains the nodes that are selected for display in other views.

It is also irrelevant to the Clip Editor view, which by design contains all of the source nodes in the project file, and to the Render Control view, which lists all Image Out nodes in the file.

There are three ways to pin a node to a view. You can use

- the Pin icon in the view's title bar,
- the drop-link method, or
- the "pin to" commands in the Node Actions menu.

You can use whichever method is most convenient, as explained in the following paragraphs. The only way to unpin a view, however, so that it starts following the current node selection again, is to use the Pin icon. (Of course, if you delete the node to which a view is pinned, this action will also unpin the view.)

PIN ICON

Click the Pin icon in the title bar to toggle the view state.

4.9 Pin Icon in default state (left) and when view is pinned (right).

If the view has been following the current node selection, it will be pinned to the node currently displayed in the view. If it has been pinned to a node, it will update to display the currently selected node.

Drop-Link

Drag the node out of the Worksheet and over the view, and then release the mouse button. The node data is now pinned to the view.

A "ghost node" icon will appear under the cursor as you drag a node over a view to pin it. (The actual node remains in its original position in the Worksheet.)

4.10 Node "blur1" is dropped into Image Viewer (left). The pinned state of the node is indicated by blue stripes (right).

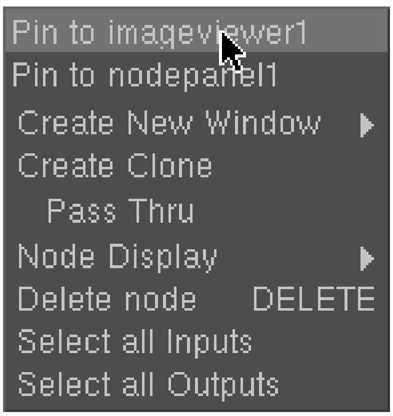
The node will remain pinned until you press the Pin button in the title bar of the view to release it, or until you pin a different node to that view.

NOTE:

You can drop-link between separate windows. Just drag the node out of the Worksheet in one RAYZ window and drop it over a view in another RAYZ window. See also "Creating a View in a Separate Window" (p. 36).

Node Actions Menu

Click and hold the right mouse button on the node you want to pin and the Node Actions menu will pop up. Select a view from the list and release the mouse button. The node data is now pinned to the view.



4.11 "Pin to Image Viewer" option being selected from Node Actions menu.

The node will remain pinned until you press the Pin button in the title bar of the view to release it, or until you pin a different node to that view.

Pinning Nodes to the Curve Editor

The Curve Editor can display data from multiple nodes. (See also "Displaying Parameters from Multiple Nodes" in chapter 8, p. 105.) This means that multiple nodes can be pinned to the Curve Editor too. When you use any method to pin a single node, and all nodes in the Curve Editor will become pinned to the view. To pin an additional node or nodes, repeat the drop-link or menu-selection method.

Click the Pin icon in the title bar of the Curve Editor to restore dynamic focus to the view.

Infinite Viewspace

The way that the Worksheet, Image Viewer, Curve Editor, and Clip Editor are used dictates that these views have an infinite viewspace, which means that the work area within the view is not limited to the current size of the view frame. You can move the viewspace around under the view frame, scale it up or down, and recenter it if necessary.

Moving the Viewspace

You can slide the viewspace around in the view frame by dragging it with the middle mouse button. In the case of the Image Viewer, for example, you can bring any part of a large image into view without resizing the view frame. In the Worksheet, on the other hand, it means that you can bring a specific part of a large node network into the view frame.

Scrolling

As an alternative to using the middle mouse button, these views also provide scrollers for navigation, which are located along the bottom and left edges. You can use the left mouse button to drag back and forth (or up and down) in a scroller to navigate around the viewspace.

NOTE:

The scrollers in the Curve Editor graph have a unique look, but they operate in the same way as the scrollers in other views.

Scaling

You can also use the scrollers to scale the viewspace by holding down the Control key as you drag. Ctrl-dragging left, or down, in a scroller scales the viewspace down and Ctrl-dragging right, or up, scales the view back up to normal size.

In the Image Viewer and Worksheet, you can also zoom in and out by using the **Minus** (zoom out) and **Plus** (zoom in) hotkeys for zooming. And the Curve Editor graph has a unique feature, the Fit button (lower left corner) that scales the graph to fit all active curves into the visible area of the graph.

Centering the Contents

An infinite viewspace also means that the contents of a view can end up out of view. When this happens, click the Center button—located in the lower left corner of the Image Viewer, Worksheet, and Clip Editor—to re-center the contents of the viewspace.

4.12 Center button.

In the Image Viewer, for example, this centers the image frame in the view pane, while in the Clip Editor it centers the timeline around the current frame marker. In the Worksheet, This command centers the node network around the currently selected node.

The hotkey for this command is the **Backspace key**. Be sure the cursor is over the view you want affect.

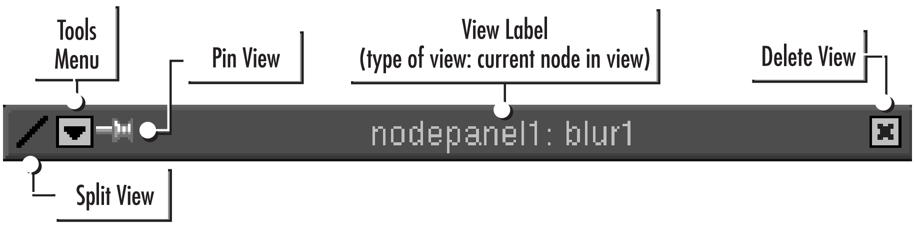
> **TIP:**
> If you have zoomed in or out, you can use the **Home hotkey** to reset the scale to normal (full size).

MANIPULATING VIEW PANES

Although the task-based displays and controls vary among the view types, all data views provide the same interface controls (the same view frame) for manipulating the view pane in the layout.

TITLE BAR

At the top of every view pane is the title bar, which is labeled with the name of the view and the node currently in the view. The title bar also has controls used to split or delete the view, and a Tools menu.



4.13 All view panes feature the same title bar controls.

TOOLS MENU

The specific contents of the Tools menu varies from view type to view type, but it is always used to access view-level control strips and panels. In the Image Viewer, for example, you can use the Tools menu to control display of the Flipbook and Wipe control strips, while in the Worksheet the Tools menu controls display of the Node Menu strip.

RESIZING A VIEW

Position the cursor over the border between two views and the cursor icon will change to up/down or left/right arrows, depending on the orientation of the border. Drag the border to resize.

4.14 The cursor icon changes to double arrows whenever you position it over the border between two views. You can then drag the border to resize the views.

If necessary, any tool strips in the view will adjust to fit the new display area. For example, if you resize an Image Viewer such that all of the items in the main control strip no longer fit in a single row, the strip will change proportion to display its tools in two rows.

Moving a View

4.15 The cursor icon changes to compass arrows when positioned over a title bar.

Position the cursor over the title bar of a view and the cursor icon will change into compass arrows. Start dragging, and an outline representing the view pane will appear under the cursor. Release the mouse button when the outline is positioned in the area you want. The other view panes in the layout will rearrange themselves around the relocated view.

Swapping Views

You can swap the position of two views using the same technique described above for moving a view. To swap two views, however, you have to match the shape of the outline to the view you want to swap.

Drag a view outline over another view until the outline snaps to the border of the other view, matching it exactly, and release the mouse button. The two view frames will swap their contents. If you drag a Node Panel over an Image Viewer, for example, the Image Viewer frame will remain the same size but now contain the Node Panel, and vice versa.

Adding a View

You can add a new view pane to the layout by selecting the type of view you want to add from the Views menu. The new view will appear at the bottom of the layout, although you can then move it as described above.

Duplicating a View (Splitting)

Another way to add a view to the layout is to split an existing view into two independent panes of the same view type, that is, to duplicate the view. The duplicate view initially has the same settings as the original, but it is not tied to the original view in any way and can be manipulated independently just like any other view.

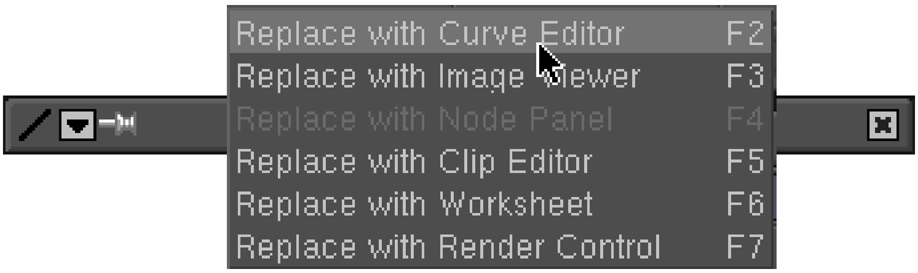
Drag the Split icon in the title bar (the diagonal line on the far left) to split the view in two. Drag to the right to split horizontally (for side-by-side views); drag downward to split vertically (one view on top of the other).

4.16 Drag the Split icon (circled) of the title bar to duplicate the view.

As you drag the Split icon, an outline representing the border that will separate the split views appears under the cursor. When you release the mouse button, the view is split at that position.

Replacing a View

You can replace any view with a different type of view in the same location. Right-click and hold on the title bar of the view pane to access the "replace with" menu and select the type of view you want.



4.17 To replace one view type with another, right-hold on the title bar and select a different view from the popup menu.

"Replace With" Hotkeys

You can also use hotkeys instead of selecting a menu command to replace whichever view the cursor is currently hovering over:

F2	Replace with Curve Editor
F3	Replace with Image Viewer
F4	Replace with Node Panel
F5	Replace with Clip Editor
F6	Replace with Worksheet
F7	Replace with Render Control

Deleting a View

To delete a view, simply click the Delete button in the upper right corner of the title bar. The other views in the layout will resize to fit the new configuration.

Maximizing a View

You can maximize any view pane to fill the interface by using either of the following hotkeys, while the cursor is positioned over the view to be maximized:

- Press the **F11 key**, or
- Tap the **Space bar** on the keyboard (pressing and holding the Space bar displays the popup node menu).

To switch back to the full layout, repeat the command.

Changing the Layout of View Elements

Many views offer control strips that are specific to that type of view. For example, the Worksheet provides a node menu strip and the Image Viewer provides a tool strip with controls for channel display, zoom, etc.

You can choose which tool strips to display in any particular view (not all tool strips are displayed by default) by using the Tools menu in the title bar of the view pane. The Tools menu is the down-arrow icon on the left side of the toolbar. Check any item in the menu to display the corresponding tool strip in the view; uncheck the item to hide it.

Moving a Tool Strip in a View

Just as you can reposition the Status Bar or the Time Scooter around the edges of the RAYZ layout, you can also rearrange the tool strips in an individual view to suit your needs.

Tool strips have grabber handles at the left edge, for horizontally oriented strips, or at the upper edge, for vertically oriented panels such as the Viewer Tools panel in the Image Viewer.

4.18 Examples of grabber handles on a tool strip (vertical stripes) and a tool panel (horizontal stripes).

Tool strips can be moved to the top or bottom edge of the view. If the view provides multiple toolstrips (the two strips at the top of the Node Panel, for example), they can be reordered or moved to opposite edges. Tool panels can be moved to the left or right side of the view frame.

When you mouse over the grabber handle for a tool strip, the cursor icon changes to compass arrows. In the same way that you reposition a view, drag the tool strip and an outline representing the strip appears under the cursor. Release the mouse when you are in position and the tool strip will drop into the new location.

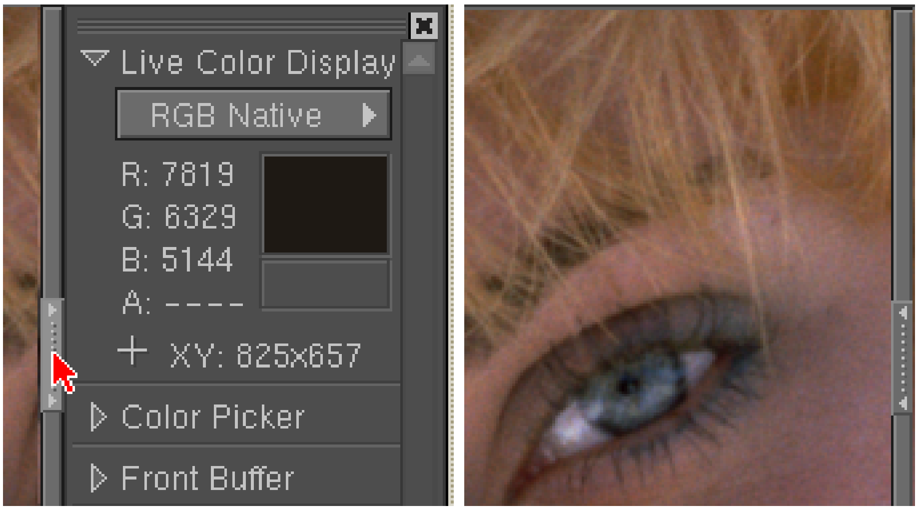
Working with Tool Panels

The most prominent tool panel in RAYZ is the Viewer Tools panel in the Image Viewer, located by default along the right edge of the view pane. The Clip Editor and the Curve Editor also feature tool panels.

Like any other view tool, you can control the display of tool panels in the Tools menu for the view. However, you also have the option of hiding the bulk of the panel while still retaining a thin control bar you can use to access the tools at any time.

This option works like the "windowshade" controls on some operating systems to stow the contents of the panel out of the way while still retaining easy access to its tools.

To shut the tool panel, or to open it if it has already been shut, click the Shutter button.



4.19 When the Shutter is clicked (left), the tool panel is stowed away (right).

You can also adjust the width of any tool panel by dragging the inward-facing edge to the left or right.

Expanding and Collapsing Grouped Controls

Most views and tool panels provide groups of parameters or other controls that are organized outline-style, with outline arrow buttons you can use to expand or collapse the group, to show or hide the controls in the group.

In *Fig. 4.19* above, for example, the Live Color Display, Color Picker, and Front Buffer controls are all groups whose display is controlled by their outline arrows.

Clicking an expanded outline arrow (which points down) will collapse the group, and clicking a collapsed outline arrow (which points to the right) will expand it.

Expand/Collapse All

As a shortcut, you can expand or collapse all groups at the same outline level in a view or panel by pressing the **o hotkey** while the cursor is over one of the arrows. (The Shift-o hotkey expands or collapses only the group under the cursor.)

Creating a View in a Separate Window

In addition to pinning a node to a specific view pane in the layout, as described in the section on "Dynamic Focus" (p. 27), you also have the option of creating a new Node Panel, Curve Editor, or Image Viewer in a separate window so that it can be resized, minimized, and so forth without affecting the current RAYZ layout.

To display the contents of a node in a separate view window:

1. Click and hold the right mouse button on the node to access the Node Actions menu.
2. Drag the mouse through the list to the Create New Window item and a submenu will appear.
3. Select New Image Viewer Window, New Node Panel Window, or New Curve Editor Window from the submenu.

The new window will appear over the main layout with the selected view type (Image Viewer, Node Panel, or Curve Editor) filling the window and pinned to the node from which it was spawned.

HOTKEYS:

You can also use hotkeys to create a separate view window. With the cursor over the node you want to pin, press **1** to create a new Image Viewer window, press **2** to create a new Node Panel window, or press **3** to create a new Curve Editor window.

Using View Windows

You can detach a node from a view in a separate window by clicking the Pin button in the title bar of the view. The view window will then display the contents of whatever node is selected in the Worksheet in the main layout.

Using separate, floating windows is just another way of arranging interface elements in RAYZ. In fact, no matter how many separate windows you create, RAYZ is still a single interface functionally in which you work on the same project file.

For example, if you create a new node while you are working in a separate Image Viewer window, the node will appear in the Worksheet of the main layout.

MULTIPLE VIEW WINDOWS

You can create as many separate view windows as you need. Be aware, however, that each Image Viewer uses additional memory, especially when it has cached a flipbook sequence, and it is easy to forget about view windows that have been minimized or are hidden behind other view windows or the main layout.

DELETING VIEW WINDOWS

To delete a separate window, use the Close Window command in the File menu at the top of the window. Do *not* use the Exit command (or Ctrl-q), which will quit the RAYZ application when invoked from any RAYZ window.

CHANGING THE LAYOUT

You can change the layout of elements in the RAYZ interface whenever you want. You can change the number, type, size, and position of data views. You can also reposition the Main Menu strip and the controls accessed from the Tools menu (Time Scooter, Process strip, and Status Bar), although the options are more limited—these elements must remain along the edges of the interface.

HOTKEY:

To restore the default layout, press the **F10** key, which is equivalent to selecting the Factory Default command from the Layouts menu.

Once you have adjusted the layout to suit a particular task or phase of shot-building, you can save it so that it can be recalled at any time.

CUSTOMIZING LAYOUTS

The default layout is a good all-purpose combination of views, but you can change the default to fit your work habits. In addition, you can create layouts optimized for specific tasks. For example:

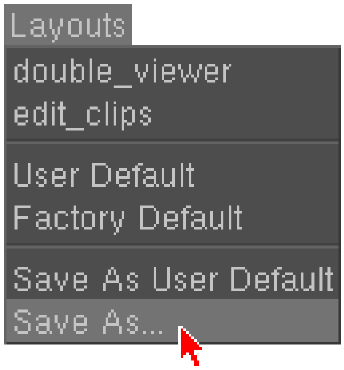
Try setting up a layout with a Worksheet, Node Panel, Clip Editor, and a small Image Viewer to use for importing all your imagery and doing whatever re-sequencing is necessary.

For adjusting mattes, you will probably want to use a layout with two Image Viewers. You can use the primary Viewer to draw a spline around an element and the second Viewer to see the result in a composite node while you continue to tweak the shape or edge characteristics of the roto.

To animate parameters, create a layout that includes a Curve Editor in the size and location that suits you best.

SAVING LAYOUTS

Once you have taken the effort to get the layout just the way you want it, you can save it for reuse.



4.20 Layouts Menu in the RAYZ Main Menu strip.

You can select **Layouts > Save As User Default** from the Main Menu strip to use it as the default layout for new project files instead of the factory default layout.

Or you can use **Layouts > Save As** to save any current configuration under whatever name seems appropriate. The new layout will subsequently appear in the Layouts menu.

LOADING LAYOUTS

To change the current layout to a saved layout, just select it from the Layouts menu and the interface will update accordingly.

DELETING LAYOUTS

To delete a layout, remove the corresponding file from the Layouts directory, which is located by default in the .rayz/2.2 application directory.

NOTE:

Layouts are saved in the directory path specified in Edit > Preferences > File Paths > Layout Search Path. For more information about using the Preferences panel, refer to *Chapter 13: Setting Preferences* (p. 147).

KEYBOARD AND MOUSE USAGE IN RAYZ

The conventions adhered to for mouse usage and hotkey assignments are described in this section.

MOUSE BUTTON USAGE

In RAYZ, each mouse button is used for a different type of activity:

- **left mouse:** general action/selection
- **middle mouse:** scrolling in the viewspace
- **right mouse:** local popup menu access (for commands specific to selected item)

CURSOR USAGE

The cursor icon in RAYZ is dynamic, indicating the potential usage of the cursor at the current position. For example, the cursor icon changes to an I-bar over data entry fields to indicate that you can type in the field. It becomes a double-headed arrow when positioned between view frames to indicate that you can click and drag the border to move it. And when using the Color Picker, the cursor changes to an eyedropper to indicate you can click the color under the cursor to sample it.

Cursor position also determines how hotkey commands are applied, as explained next.

USING HOTKEYS IN RAYZ

Many common commands have keyboard equivalents, or hotkeys. When using hotkeys in RAYZ, it is important to remember that the position of the cursor determines which view in the layout the command will be applied to. For example, the F11 hotkey, which maximizes a view, applies to whichever view the cursor is over when the key is pressed.

Hotkeys for individual commands are noted in the section of the documentation that covers the corresponding task, and a complete list of the factory default hotkeys can be found in "Default Hotkeys in RAYZ" in chapter 13 (p. 153).

You can reassign the hotkey currently used for a command, and you can add a hotkey for a command that does not currently have one. This process is described in "Hotkeys" in chapter 13 (p. 152).

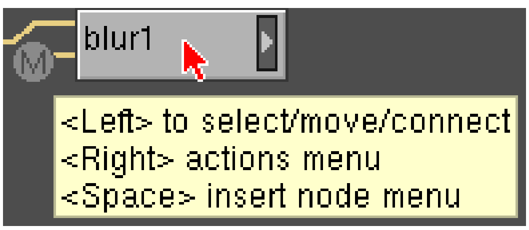
GETTING HELP

This manual is accessible from with RAYZ and, in addition to the "Status Bar" (p. 26) readouts, you can also get contextual feedback by using Help Tags.

HELP TAGS

Help Tags are context-sensitive popup help boxes that appear when you let the cursor hover over individual objects in the interface. The Help Tag will tell you what the item is for or how to use it (by noting a mouse button action or hotkey for it).

To enable this feature, check the Help Tags option in the RAYZ Help menu.



4.21 Help Tag (yellow box).

The hotkey or mouse button to use, if applicable, may be indicated in <angle brackets> and followed by a description of the type of action that can be taken or will result.

Online Manual

You can access this manual in HTML format from within the RAYZ application by selecting RAYZ Manual from the Help menu or by using the **F1 hotkey**.

RAYZ searches for the HTML manual files in the directory specified in Edit > Preferences > File Paths > Help Search Path.

By default, the HTML manual is opened in Netscape; however, you can override this internal setting by specifying another browser in Edit > Preferences > Settings > Help Browser. For more information about using the Preferences panel, refer to *Chapter 13: Setting Preferences* (p. 147).

chapter 5

NODES IN THE WORKSHEET

The Worksheet view is where you create and edit the flowchart of nodes that describes a shot, and it is also where you choose which node data to display and modify in the Image Viewer, Node Panel, and Curve Editor.

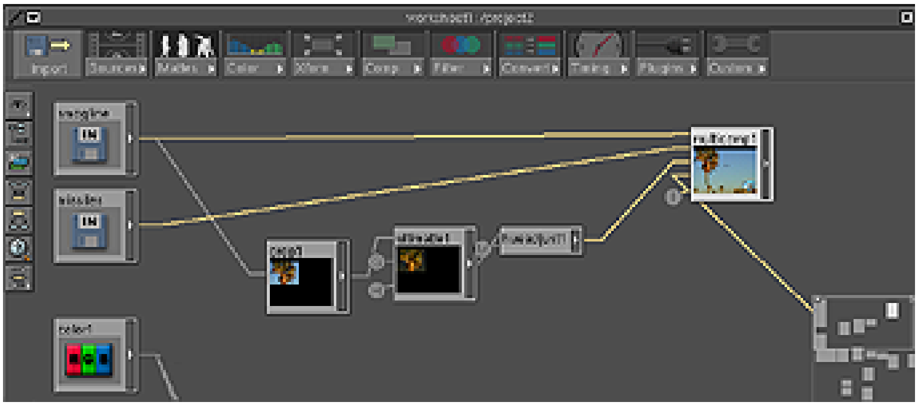
Building the Shot

The Worksheet includes a node menu strip that you can use to create any node you need, starting with one or more source nodes. Then you connect other nodes that represent the operations you wish to perform on the source imagery.

You can think of the image data as flowing downstream from the source nodes through the network, being modified along the way, until the modified imagery is ready to be rendered (written to disk as a series of image files). This means, for example, that when you composite two or more images, the separate flows of data merge at the composite node.

Changing the Shot

The node network is essentially a description of the image operations to perform, and the order in which to perform them. You can edit this description at any time and at any point in the node network. You can insert and delete nodes and change the order in which operations occur by changing the connections to and from the nodes.



5.1 Node network in Worksheet. Also note the Node Menu strip at the top, the Toolbar on the left, and the Navigator in the lower-right corner.

Controlling the Contents of Other Views

In addition to editing the flowchart of nodes, the Worksheet is also used to specify the particular node you want to edit in other views, so that parameters for a node operation can be set in the Node Panel and the effect can be evaluated in the Image Viewer.

Click once on a node and release the mouse button to select it. The node will highlight and its data will be displayed in any Image Viewer, Node Panel, or Curve Editor in the current layout that has not been pinned. (A view that is pinned does not follow the current Worksheet selection. See "Dynamic Focus" in chapter 4, p. 27 for more information.)

Getting Around in the Worksheet

The Worksheet offers an infinite workspace. The current size of the view frame may not be large enough to view the entire node network at once; in fact, a very large network may not fit in the visible workspace even when the Worksheet is maximized.

To make it easier to view and manipulate the current node network, you can drag the viewspace around under the view frame to bring a specific section into view, and you can zoom in and out.

Moving

There are several ways to move the workspace around in the view frame:

- Hold down the middle mouse button and drag the workspace in any direction—do not click on a node, or you will drag it around instead.
- Use the **nudge hotkeys**—the arrow keys on your keyboard—to move the viewspace in even increments in the direction of the arrow. You can also hold down the Shift key as you press an arrow key to move in larger increments.

ZOOMING

You can scale down the contents of the Worksheet to see more of the node network at once:

- Hold down the Ctrl key while dragging the workspace. Drag left to zoom out (scale the nodes down); drag right to zoom back in to normal scale.
- You can also use the **zoom hotkeys**. Press the minus key (-) to zoom out and the plus key (+) to zoom back in to normal.

CENTERING

The Center button in the lower left corner of the Worksheet is used to recenter the node network in the view frame. Click the button, or press the **Backspace hotkey**, and the network will snap to center.

TIP:

When you have zoomed out on the viewspace, press the **Home hotkey** to simultaneously recenter the contents and reset the scale to normal.

USING THE NAVIGATOR

Another way to get around in the network is to use the Navigator panel, which is located by default in the lower-right corner of the Worksheet.

The Navigator provides a "birds-eye" view of the network with an outline over it indicating the view frame of the Worksheet. Click or drag to move the view frame indicator over a different part of the network.

You can turn off display of the Navigator if you wish by unchecking it in the Tools menu (in the Worksheet title bar). However, you can also shrink the Navigator to a button when you need more room in the view frame; then you just click the button to reveal the full Navigator panel again.

5.2 Navigator: Click button (highlighted, left) to stow (middle); drag button to resize (right).

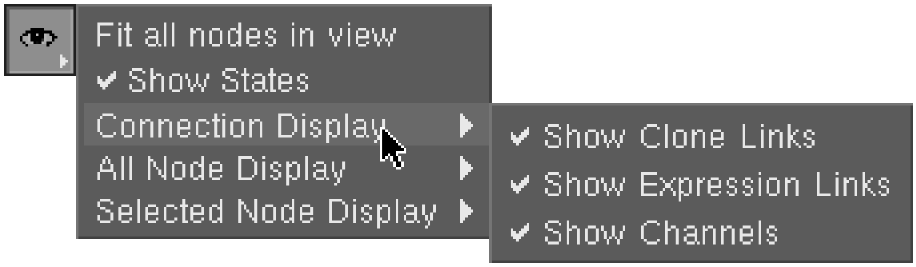
To "stow" the Navigator, click the button in the upper-right corner. You can also resize the Navigator by dragging the same button.

ABOUT THE TOOLBAR

The Toolbar is displayed by default in the Worksheet as a vertical strip running down the right side of the view frame. Like the other tool strips in the Worksheet, you can control its display in the Tools menu located in the title bar of the Worksheet.

The Worksheet Toolbar contains menu and command buttons for displaying and modifying node networks, which are described below.

Display Menu



5.3 Contents of the Display menu, which is located at the top of the Worksheet Toolbar.

Fit All Nodes in View

This command scales and repositions the network to fit into the view frame.

Show States

When Show States is checked, as it is by default, nodes that are animated or cloned display an color-coded bar on them to indicate their state. See “Node States” on p. 56 for a description and illustration.

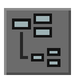
Connection Display

This submenu lets you check and uncheck Show Clone Links, Show Expression Links, and Show Channels. Clone and Expression Links are described in the section on “Link Indicators” on p. 56; Channel information display is described in the section on “Flowlines” on p. 50.

All Node Display & Selected Node Display

These two submenus have the same commands as the Node Display submenu in the Node Actions menu for controlling the display of thumbnail images or icons on nodes.

The difference is that All Node Display affects all nodes in the Worksheet, and Selected Node Display affects all currently selected nodes in the Worksheet. See also “Node Actions Menu” on p. 51.

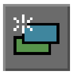


Hide/Show Navigator

Click this button to toggle display of the Navigator on and off in Worksheet. See also “Using the Navigator” (p. 43).

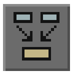
Find Nodes

Click this button to bring up the Find Nodes panel, which is described in the section on “Finding Nodes” on p. 56. (This button is equivalent to pressing Ctrl-F.)



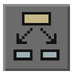
CREATE UNDERLAY

You can use this button as an alternative to Ctrl-clicking in the Worksheet to create an underlay. (For more information about underlays, see the section on "Adding Underlays to the Worksheet" on p. 60.) Click the button and move the cursor over the Worksheet. The cursor icon changes to indicate that your next click will create a new underlay.



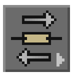
GROUP SELECTED NODES

You can click this button as an alternative to using the Worksheet Actions menu command for grouping nodes, as described in the following section on the Worksheet Actions menu.



UNGROUP SELECTED GROUP NODE

You can click this button as an alternative to using the Node Actions menu command to ungroup a Group node. For more information, see "About Group Nodes" in the following section.



CREATING AND EDITING FILE GROUP PROJECTS

The final button in the Worksheet Toolbar is used *only* when you want to turn the current project file into a customized File Group file, which will be imported into other compositors' projects using the File Group node. Do *not* use it to edit regular Group nodes. For more information, see "Creating File Groups" in chapter 22, p. 434.

WORKSHEET ACTIONS MENU

Click and hold down the right mouse button in the Worksheet to access the Worksheet Actions menu, which is used to cut, copy, paste, and delete nodes. Refer to "Cut, Copy, and Paste" on p. 53 and "Deleting Nodes" on p. 53 for more information about these commands.

NOTE:

To clone a node, use the Create Clone command in the Node Actions menu of that node. Clones are described in "Creating Clones" on p. 54.

The Worksheet Actions menu is also used to access the node finder, as described in "Finding Nodes" on p. 56. The Custom commands in the Worksheet Actions menu are described in "Adding Nodes to the Custom Menu" on p. 59.

ABOUT GROUP NODES

The final command available in the Worksheet Actions menu, Create Group, can be used to place all selected nodes in a container node, called a Group node. This is a good way of organizing a large network into manageable sub-networks.

In addition, however, the Node Panel of a Group container can be customized to display relevant parameters from various nodes within the Group. (The Group node can even be saved to the Custom node menu for reuse.) Group nodes merit their own chapter in this manual: *Chapter 22: Creating Group Nodes* (p. 427).

To ungroup it, select Ungroup from the Node Actions menu of the Group node.

Creating and Connecting Nodes

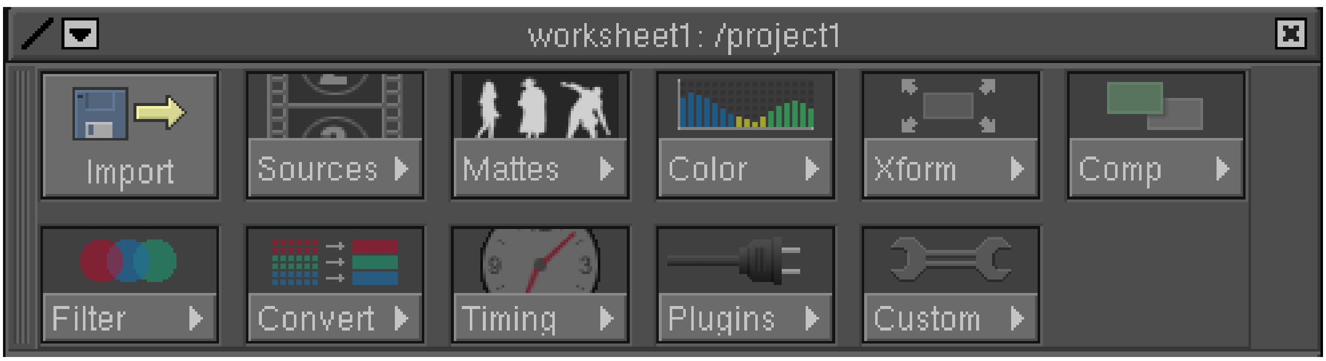
You create a new node by selecting it from a menu in which all node operations are listed. The nodes are organized in categories according to general purpose: matte creation and editing, color correction, compositing, and so on. When you select a node from the menu list, a new node of that type is created and placed in the Worksheet.

The node menus are available in a menu strip in the Worksheet and in a popup menu. The popup menu can be accessed from multiple views and offers more control over how the new node is inserted into the network, as explained in the section on "The Popup Node Menu" on p. 47. In fact, once you become familiar with the popup node menu, you may use it exclusively.

TIP:

You can even assign a hotkey to create any type of node you use frequently. For more information about assigning hotkeys to commands, see "Hotkeys" in chapter 13, p. 152. For more shortcuts to node creation, see "The Popup Node Menu" on p. 47.

The Node Menu Strip



5.4 The Node Menu strip includes the categorical node menus and the Import button (top left).

Click and hold one of the node menus in the strip at the top of the Worksheet to access a list of nodes in that category. For example, the Source node menu contains the Image In node, which is used to import image files into RAYZ, and other nodes, such as Color Bars and Gradient, that are used to generate images from scratch.

NOTE:

> The node menu strip also includes the Import button, which is equivalent to using the Import Footage command (Ctrl-i) in the File menu. Import Footage is described in *Chapter 10: Importing Images* (p. 123).

If a node is selected in the Worksheet when you create the new node, the new node's input is connected to the output of the previously selected node. If no node is selected, the new node is placed in the Worksheet unattached.

If the node menu strip is not visible, select it from the Tools menu in the title bar of the Worksheet. You can use this menu to turn the node menu strip on or off at any time. You may want to turn it off to save room in the Worksheet and use the popup menu instead.

Text-Only Node Menu Display

You can display a text-only node menu in the Worksheet to save space, in lieu of using the iconic node menu strip. The type of node menu display is a preference. To switch from icon to text display, select Edit > Preferences > Worksheet > Icon Node Menu, and change the value from On to Off.

5.5 Node menu display can be set to text only in the Preferences panel.

For more information about using the Preferences panel, see *Chapter 13: Setting Preferences* (p. 147).

The Popup Node Menu

You can create new nodes without using the node menu strip: hold down the Space bar and a popup version of the node menu will appear. The popup menu is context sensitive—the new node will be connected based on the position of the cursor (that is, based on the object being targeted by the cursor) when you access the menu.

Creating Multiple Nodes

You can create as many new nodes as you want while the popup node menu is visible. Just continue to hold down the Space bar after selecting the first node and then select as many additional nodes as you need. Each new node is placed in the Worksheet as soon as you select it from the menu and connected to the previous node automatically.

Guidelines for Adding Nodes to a Network

You can access the popup node menu with the cursor over a node, over a specific input connector, or over a connection line, and the new node will be inserted into the network accordingly:

- With the cursor positioned over a node, the new node is connected to the output connector of the target node.

- With the cursor positioned over an input connector, the new node's output is connected to the target input connector.
- With the cursor positioned over a connection line, the new node is inserted between the nodes at either end of the target line.

Inserting a New Node into an Existing Dataflow

If the output connector of a target node is already connected to another node, the new node is inserted into the network between the target node and the next node downstream. If the target node has multiple connection lines branching from its output, the new node is inserted into the top branch.

TIP:

To insert a new node into a specific branch, target the appropriate connection line branching from the node instead of the node itself. The line will brighten when the cursor is over it to indicate that it is targeted.

Creating a New Branch

To start a new branch from the target node, hold down the Control key when you access the popup node menu. The **Ctrl-Space bar** option accesses the *Branch* menu, which is green to distinguish it from the regular, or Insert, menu.

When you select a node from the Branch node popup menu, it is connected to the output of the target node in a new branch.

Replacing a Node

To replace the target node with a new node of a different type, use the Shift modifier with the Space bar. Holding down **Shift-Space bar** accesses the *Replace* menu, which is blue to distinguish it from the Insert and Branch menus.

Any node you select from the Replace menu will replace the target node, with any connections to other nodes intact. For this reason, the Replace menu only includes nodes with the same number of connectors as the target.

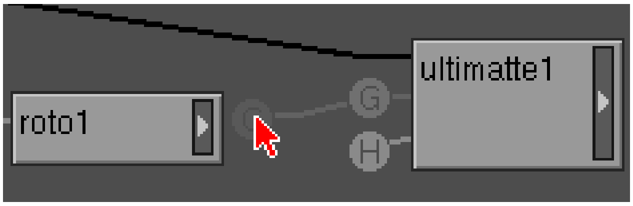
TIP:

You can hold down the Space bar to access the popup node menu from the Image Viewer and Node Panel, too. In that case, the new node is appended to the node currently in the view. If you use the Control-key modifier, the new node starts a new branch, and if you use the Shift-key modifier the new node replaces the node in the view.

Connecting Nodes

As discussed in the previous section on "The Popup Node Menu" (p. 47), node connections are often made automatically when you create a node. In any case, it is easy to make, change, and delete connections.

To **connect** one node to another, you drag one of the “Input Connectors” (p. 49) of a node to the “Output Connector” (p. 50) of another node and release the mouse button. (You can also drag the output connector of a node to an input connector of another node.)



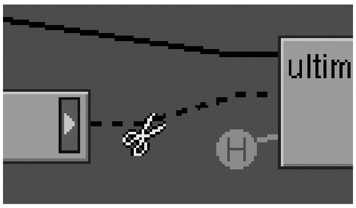
5.6 A “ghost connector” appears under the cursor as you drag.

If you change your mind, just release the mouse button while the cursor is over the Worksheet instead of over a node.

NOTE:

To avoid infinite loops, RAYZ will not allow you to connect the output of a downstream node to an upstream input.

To **delete a connection**, Ctrl-click the connection line.



5.7 When you hold down the Control key and move the cursor over a line, the cursor changes to a scissors icon to indicate that clicking on the line will delete the connection.

To **change a connection**, drag the connection line from the input connector of one node to the output of another and release the mouse button.

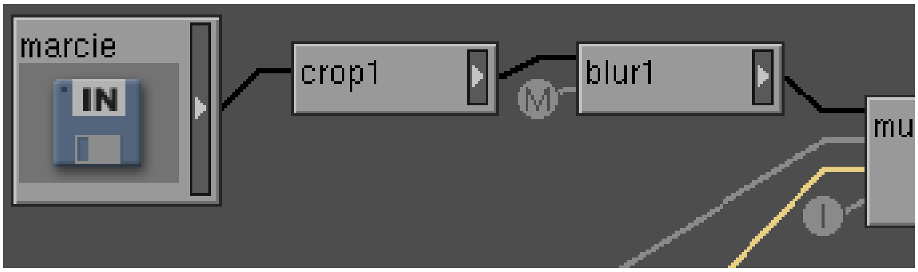
TIP:

The RAYZ Status bar displays relevant information about any node or connection line you target with the cursor. Point the cursor at each connection line to a multi-input node such as Multi-comp to quickly sort out which line flows from which input node.

INPUT CONNECTORS

When you create a new, unconnected node, you will see one or more input connectors hanging off the left side of it. The number of input connectors depends on the type of node.

The Image In node has no input connectors, because it is used to start a flow of image data. The Brightness node, on the other hand, has two inputs, one for the image to be brightened and an optional input to use for a mask. And a node like Multi-comp, which can accept an unlimited number of input images, generates a new input connector each time you use the last one.



5.8 Image In node has no input connector; Resize has an optional Reference input connector (R); Blur has an optional Mask input connector (M); and Multi-comp creates new Image input connectors (I) on the fly.

Input Types

In addition to the primary input image—the image being modified by the node operation—many nodes give you the option of connecting a mask input, which is used to control which pixels in the primary input are modified, and how much. (See also "Using Mask Inputs" in chapter 7, p. 102.)

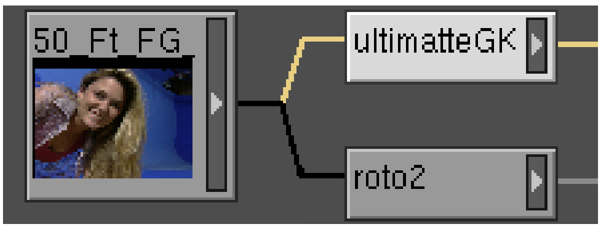
Some nodes accept both a mask input and another control image, in addition to the primary input. For example, the Ultimatte node enables you to input separate garbage and holdout mattes, and the Convolve node lets you input a mask image and a kernel image.

The node input connectors for these optional inputs are easy to distinguish because they are labeled differently: rather than being labeled "I," mask inputs are labeled "M," the Convolve kernel input connector is labeled "K," and so forth.

Position the cursor over any input connector to find out more about it; its purpose will be explained in the Status bar.

Output Connector

Regardless of the number of input connectors, all nodes have a single output connector, because the data can flow out of a node in as many directions (that is, create as many branches) as you need.

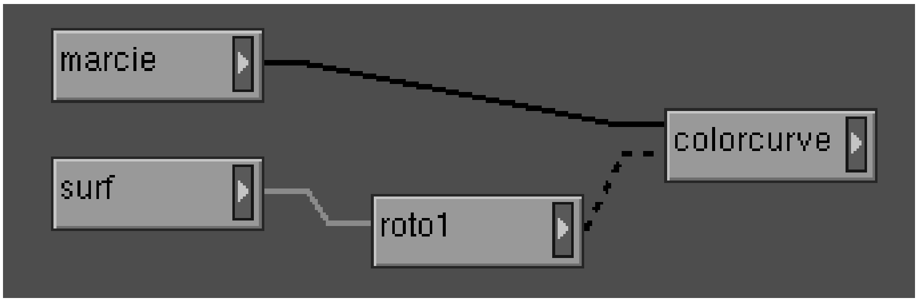


5.9 Example of node image data branching out into separate flows.

The exception is the Channel Split node, which is specifically designed to branch the image data into separate RGB and Alpha outputs. And a Group node may also have multiple outputs, where each output represents a different node within the group. For more information, see *Chapter 22: Creating Group Nodes* (p. 427).

Flowlines

By default, the flowlines connecting the nodes are coded to indicate the type of image data flowing through them. RGBA images are indicated by gray flowlines, RGB images by black, and Alpha (single channel) images by dotted black lines. Image data with five or more channels is indicated by white flowlines.



5.10 Black flowline indicates RGB image data; gray indicates RGBA; and dotted black represents Alpha only.

You can turn off channel-coded display of the flowlines by unchecking Connection Display > Channels in the "Display Menu" (p. 44) of the Worksheet Toolbar.

Working with Nodes

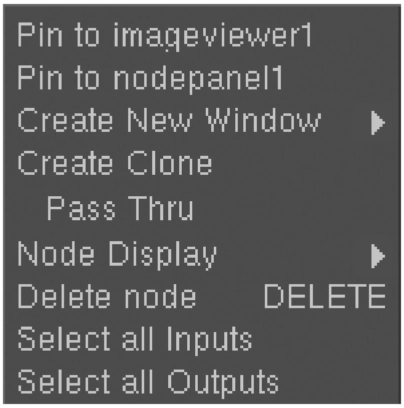
You can reposition any node in the Worksheet by dragging it. You can cut, copy, paste, clone, or delete a node by selecting the appropriate command from the Node Actions or Worksheet Actions popup menu or by using the hotkey equivalent. In the same way, you can display a thumbnail image of the node contents or turn a node off temporarily so that the image data passes through it unchanged. All of these options are described in the following paragraphs.

Node Actions Menu

Every node features a Node Actions popup menu. Commands in this menu affect only the individual node from which the menu was accessed. You can use the Node Actions menu to

- pin the node to a view
- create a new view in a separate window
- create a clone of the node
- "declone" a cloned node
- "ungroup" a Group node
- turn "Pass Thru" mode on and off
- turn thumbnail image display on and off
- turn node icon display on and off
- delete the node
- select all inputs (upstream nodes) or outputs (downstream nodes)

To access the Node Actions menu, click and hold down the right mouse button on the node. (The node does not actually have to be selected to use the menu, as explained below in "Manipulating a Node Without Selecting It.")



5.11 Typical example of items available in the Node Actions menu.

Commands accessed from the Worksheet Actions menu, on the other hand, affect all selected nodes. (The Worksheet Actions menu is accessed by right-holding in the Worksheet instead of on a node.)

Selecting Multiple Nodes

You can select multiple nodes to manipulate them as a unit. Shift-click to select multiple nodes, or drag a bounding box around them.

To deselect a node or nodes, you can click on another node to select it instead, or you can click anywhere in the background of the Worksheet so that no nodes are selected.

Selecting All Node Inputs or Outputs

For any node, you can select all nodes located upstream in the dataflow by choosing “Select all inputs” from the Node Actions menu. All nodes that contribute data to the node will be selected in the Worksheet.

To select all nodes downstream from a particular node, choose “Select all outputs” from the Node Actions menu. All nodes into which the current node data flows will be selected in the Worksheet.

Manipulating a Node Without Selecting It

You can reposition a node, make node connections, and invoke any command in the Node Actions menu without actually selecting the node. This means that you can arrange and edit a node network in a number of ways without changing the current display in the Image Viewer and Node Panel, given the fact that node selection is the method used to control which node data is displayed in other views.

NOTE:

Whenever you want to apply the same action to multiple nodes at once, however, you will have to select the nodes.

To move a node without selecting it, simply drag it to a new location and release the mouse button. The drag action, where you click and hold down the mouse button as you move the mouse, is interpreted differently from a click. Of course, you can also drag a selected node to reposition it.

Just as you can move a node without selecting it, you can also invoke commands from the Node Actions menu without selecting the node: Right-click and hold down the mouse button to access the popup menu, drag through the menu until the command you want is highlighted, and then release the mouse button.

To use the hotkey for a command in the Node Actions menu, you need to indicate which node the command should apply to by targeting the node; that is, by positioning the cursor over the node. For example, if you press the p key you will toggle "Pass Thru" (p. 54) on or off for the targeted node.

Cut, Copy, and Paste

To cut or copy a single node or multiple nodes, select the node or nodes and then use the appropriate command in the Worksheet Actions menu (right-hold in the Worksheet to access).

You can also use the hotkeys: **Ctrl-x** to cut and **Ctrl-c** to copy. As long as the cursor is anywhere in the worksheet, these hotkeys apply to any and all selected nodes. (You cannot target an unselected node and cut or copy it.)

To paste a node or nodes, select the Paste command from the Worksheet Actions menu, or use the hotkey equivalent, which is **Ctrl-v.**

Deleting Nodes

Single Node To delete a single node, target it in the Worksheet and press the Delete key. This is the equivalent of selecting the Delete Node command from the Node Actions menu. Be sure that the cursor is *not* over an unselected node at the time or the targeted node will be deleted instead.

Multiple Nodes To delete multiple nodes simultaneously, select them and then use the Delete Nodes command in the Worksheet Actions menu (right-hold in the Worksheet to access). If the cursor is targeting one of the selected nodes and you use the Delete key, only the target node will be deleted; the other selected nodes will remain in the Worksheet.

Pinning a Node to a View

The Node Actions menu includes an "pin to" choice for every Node Panel, Image Viewer, and Curve Editor in the current layout. For example, you would select "pin to imageviewer1" to pin the node image to the Viewer named "imageviewer1" in the layout. This concept is explained fully in the section on "Dynamic Focus" in chapter 4, p. 27.

Pinning a Node to a New View Window

In addition to pinning the node to an existing view in the current layout, you can use the Node Actions menu to create a new view window that displays the contents of the node. This is described in detail in the section on "Creating a View in a Separate Window" in chapter 4, p. 36.

Creating Clones

You can create a clone of any node (using the Node Actions menu command described above). When you clone a node, the clone and the original node become indistinguishable. Not only is the clone an exact copy of the original node in all of its parameter values and other characteristics, but any change you make to either node is reflected in the other. You can create as many clones of the same node as you wish.

Clones are a great way to apply the same change to a number of different images because you can adjust the values in one of the clones and all of the others change in the same way. Clones are always in sync. This means that you can go back and tweak the effect at any time by changing one node, not all of them.

Declining

To break the sync between a node and its clone (or clones), select "Declone node" from the Node Actions menu. Once the node has been decloned, you can make changes to its parameters independently.

Pass Thru

You can temporarily turn off a node, so that the image data flowing into the node from upstream passes through the node unchanged. This is more convenient than having to actually reroute the connections around the node and then change them back again later.

Select Pass Thru from the Node Actions menu, or press the **p key** on the keyboard while the cursor is over a node to toggle Pass Thru on and off. (Pass Thru mode can also be controlled by using the checkbox in the strip at the top of the Node Panel.)

5.12 When Pass Thru mode is in effect, a red "x" drawn over the node indicates this fact.

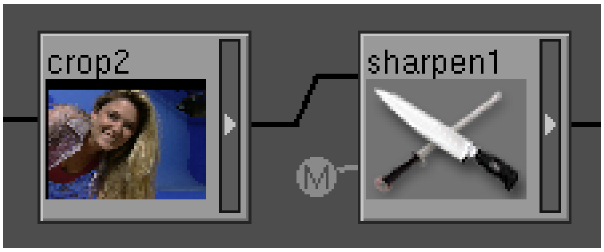
Pass Thru is disabled in the Source nodes (nodes which do not have an input connector), as they start a dataflow and therefore have no upstream data to pass through, and in the Image Out node, which terminates a flow.

Node Thumbnail Display

You can display icons or thumbnail images on any node by selecting an option from the Node Display submenu of the Node Actions menu:

- node icon (for type of node: Image In, Blur, Multi-comp, etc.)
- node image (image data as output by the node)
- node only (no image; the most compact type of node display)

This is a great way to make large node networks more "legible" and to quickly scan networks to locate specific nodes.



5.13 Crop node (left) displays thumbnail of image data; Sharpen node (right) displays icon of node type.

For example, you can display thumbnails on all Image In nodes to easily distinguish various source imagery, and you can use thumbnails to tag nodes at significant points in a shot. In fact, all source nodes, including the Image Out node, appear by default with the node thumbnail image displayed.

TIP:

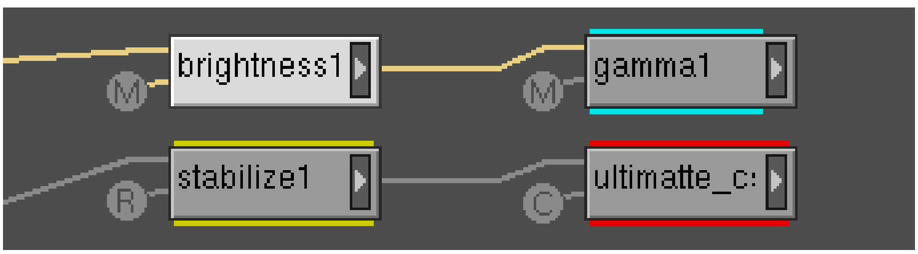
You can use the Display menu in the Worksheet Toolbar to turn thumbnail or icon display on or off in all nodes, or all selected nodes, at once by selecting an option from the "All Node Display" or "Selected Node Display" submenus.

For another way to make a large network easier to scan, see "Adding Underlays to the Worksheet" on p. 60.

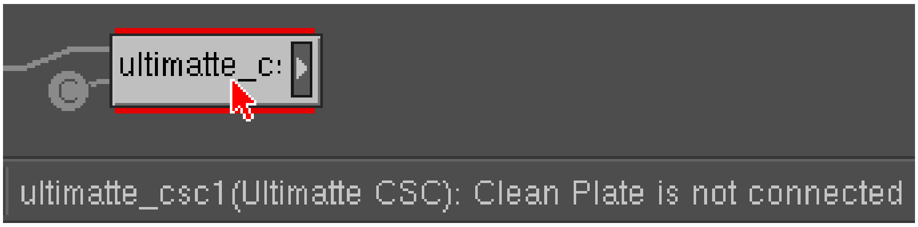
Interpreting Node Feedback

Nodes in the Worksheet change their appearance to indicate various conditions:

- When you position the cursor over a node it brightens subtly to indicate that it is targeted, and if you click on the node to select it, it highlights distinctly.
- When the network is processing image data, each node becomes blue in turn as its data is processed.
- If you pin a node to a view, aqua stripes appear across the top and bottom.
- Yellow stripes appear to alert you to a potential problem.
- If the node is in an error state, red stripes appear.



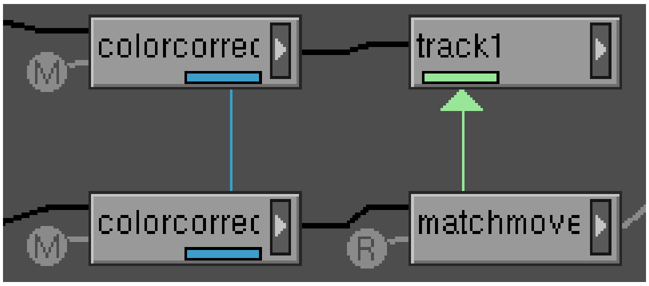
5.14 Node feedback, clockwise from top left: Brightness is selected; Gamma is pinned to a view; Stabilize is in alert mode; Ultimatte CSC is in error mode (because it requires the other input to be connected).



5.15 Mouse over a node to read the error or alert message in the Status Bar.

Node States

Nodes also change their appearance to indicate whether the node is animated (green bar on node) or cloned (blue bar on node).



5.16 Animated nodes have a green bar on the bottom-left edge; cloned nodes have a blue bar on the bottom-right edge. Also note the clone and expression link lines.

You can turn off state display if you prefer, and the nodes will take up less space in the Worksheet. Node state display is controlled in the "Display Menu" (p. 44) in the Worksheet Toolbar.

Link Indicators

Color coded lines called link lines are sometimes drawn between nodes to indicate that a certain relationship exists between them, rather than indicating the flow of image data. *Fig. 5.16* above illustrates this.

Clone Links: Blue link lines are drawn between cloned nodes in the Worksheet to indicate which nodes are clones of each other.

Expression Links: When a node parameter uses an expression that references another node, a green link line is drawn from the node with the expression to the node being referenced, with an arrow pointing to the referenced node.

You can turn off display of expression and clone link lines by unchecking corresponding item in the Connection Display submenu of the Display menu in the Worksheet Toolbar.

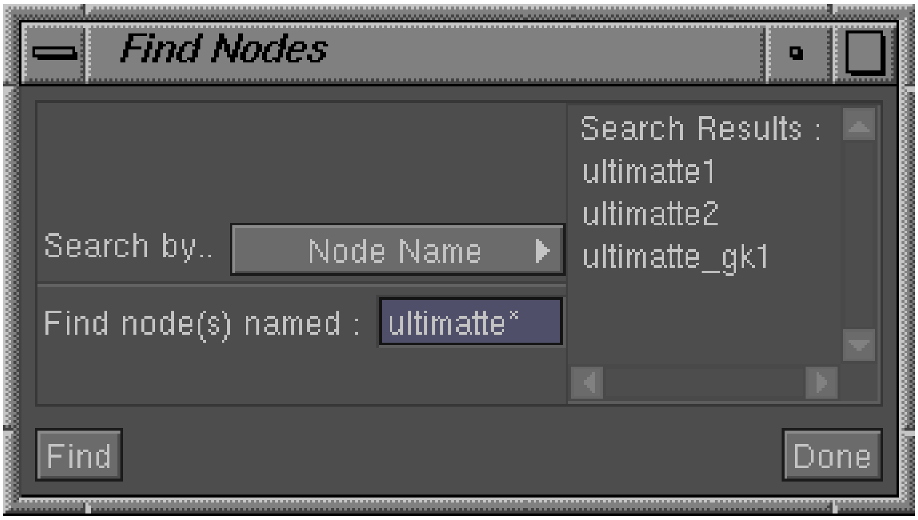
Finding Nodes

You can search for any node in the Worksheet using the following criteria:

- name
- type
- category
- state

To access the Find Nodes panel, do any of the following:

- Click the Find Nodes button in the Worksheet Toolbar.
- Right-hold in the Worksheet and select “Find Nodes…” from the Actions menu.
- Press the **Ctrl-f hotkey** while the cursor is over the Worksheet.



5.17 Find Nodes panel shows results of search for all nodes with names that start with “ultimatte.”

SELECTING SEARCH TYPE

Select the type of search you want to perform from the Search By menu and then enter the search string, if searching by name, or choose a search criterion from a menu of options for the selected search type.

SEARCH BY NODE NAME

When Node Name (the default) is selected, a text field is active in which you type the search string, as shown in *Fig. 5.17* above.

If you do not remember the exact name of the node, or if you want to find a group of similarly named nodes, you can type wildcards (metacharacters) in the search string:

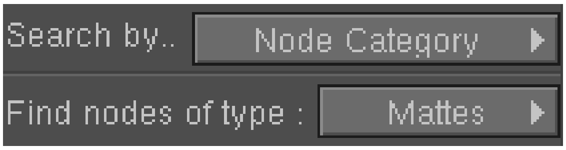
CHAR.	RESULT	EXAMPLE
*	matches any string of characters, including none	`ultimatte*` would find *ultimatte1* and *ultimatte_gk1*
?	matches any single character	`roto?` would find *roto1* and *roto2*
[ ]	matches any single character you type within the brackets	`*[1]` would find *roto1* and *ultimatte1* and *image101*

Search by Node Type

5.18 Node Type Search.

When you search by Node Type, the text field is replaced by a menu from which you can select any type of node, from Add to Z-comp. The search will return all nodes of that type, listed by their unique name (“add1” or “my_zcomp,” for example).

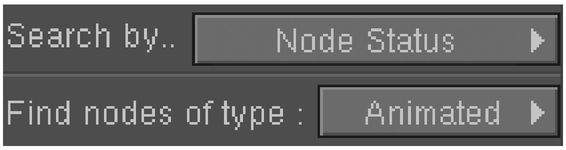
Search by Node Category



5.19 Node Category Search.

Search by Node Category when you want to find all node types in a category, such as Source, Matte, or Composite nodes. The results will be listed by name. For example, if you select Matte Nodes as the category, the search will return any Roto or Ultimatte nodes in the network, listed as “roto1,” “roto2,” “ultimatte1,” and so forth, depending on how you named them.

Search by Node Status



5.20 Node Status Search.

You can search for nodes by status to find all nodes that are currently

- Animated
- Pinned
- Cloned
- In Error State
- In Warning State

Search Results

Press the Find button to display the search results. All nodes matching the search criteria are listed by name (“blur1,” e.g.) in the pane on the right.

Double-click on any name in the list to select the corresponding node and reorient the Worksheet to bring the node to the center.

You can perform as many searches as you need without closing the panel. When you are finished, press the Done button to close it.

Adding Nodes to the Custom Menu

You often use several nodes together in a specific way to create a particular effect. If you want to reuse such a configuration, you can save it to the Custom menu, where it can be selected in one step rather than starting from scratch each time. This is analogous to recording a macro.

1. Select all nodes you want to save to the Custom menu.
2. Right-hold in the Worksheet and select "Save Selected As Custom" from the Worksheet Actions popup menu.
3. Type a descriptive name in the dialog box that will appear and press Enter when you are done.

The name will subsequently appear in the Custom menu, just like any other node. When you select it, the exact configuration of nodes you saved, with any connections intact, will appear in the Worksheet.

Managing Custom Node Menu Entries

Once you have created a custom node, you can use the "Manage Custom" command in the Worksheet Actions menu to rename or delete any custom node, as described next.

Renaming a Custom Node

To rename a custom node:

1. Select "Manage Custom" from the Worksheet Actions menu.

   A panel will appear that lists all custom nodes.
2. Double-click the node in the list to activate an editable text field.
3. Type the new name in the field in place of the old one.
4. Click the OK button at the bottom of the panel to actuate the change and dismiss the panel.

Deleting Custom Nodes

To delete a Custom node:

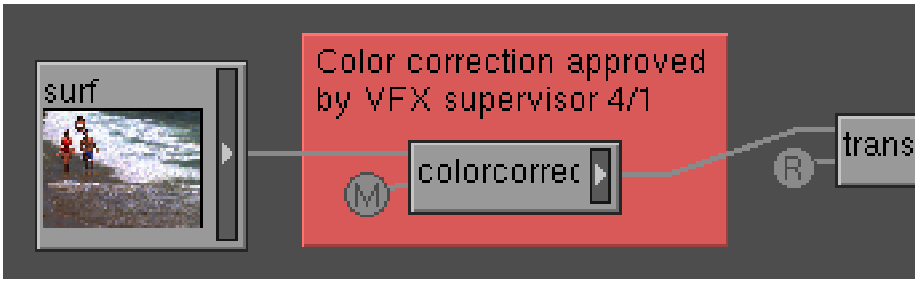
1. Select "Manage Custom" from the Worksheet Actions menu.

   A panel will appear that lists all custom nodes.
2. Click once on the node in the list you want to remove and then press the Delete key.
3. Click the OK button at the bottom of the panel to actuate the change and dismiss the panel.

Adding Underlays to the Worksheet

Underlays are resizable, color-coded markers that make it easy to find specific nodes or groups of nodes. This Worksheet feature can be especially useful in large networks. Underlays include an editable text field so that they can be labeled or used for notes and comments. Underlays never cover up the nodes (hence the name).

5.21 Example of an underlay used to comment on the approval status of a color correction operation.



To draw an underlay, Ctrl-drag a bounding box in the Worksheet. The underlay will be drawn in the bounding box area. Alternatively, you can click the Create Underlay button in the Worksheet Toolbar instead of holding down the Ctrl key. The cursor icon will change to indicate that an underlay will be created by your next click or drag in the Worksheet.

Adding Comments

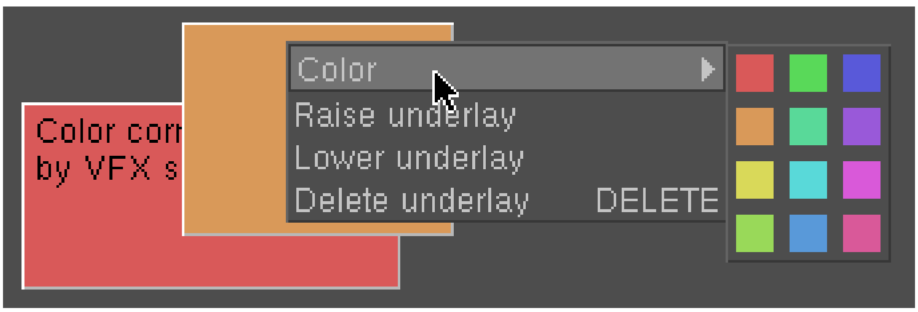
To add text, click the underlay to select it and a box with a dashed outline will appear in the upper left corner. Then you can click in the text box and start typing. Press Enter when you are done.

To type a line break and continue typing on the next line, press Shift-Enter. To edit existing text, double-click it.

Underlay Actions Menu

To change the color or stacking order of an underlay, right-click and hold on the underlay to access the Underlay Actions popup menu.

5.22 Underlay Actions Menu: right-hold on underlay to access.



Underlay Color

You can change the color of any underlay by selecting a different swatch from the Color submenu of the Actions menu. (To edit the swatch colors, refer to the section below on the Underlay Colors Tool Strip.)

Underlay Stacking Order

Underlays will overlap if you position them to do so. To change the order of an underlay in a stack of underlays, select Raise or Lower from the Actions menu.

Underlay Colors Tool Strip

A different color is assigned to each underlay you create in turn, as defined in a palette of underlay colors. To display this palette in a tool strip, check the Underlay Colors item in the Tools menu (located in the title bar of the Worksheet).

To change the color of any swatch in the palette, right-hold on the swatch to access the popup color spectrum. Drag the cursor across the spectrum bar to select the color you want and release the mouse button.

NOTE:

To change the default color palette, edit the color values in the Project Settings panel, which is accessed from the RAYZ Edit menu. Expand the Current Colors group and select Node Underlay Color Palette.

chapter 6

USING THE IMAGE VIEWER

The Image Viewer is used for every image display task in RAYZ, whether you want to view a single frame or play back a sequence.

You can evaluate an image at any frame of any node in the shot and adjust the various parameters in the Node Panel accordingly, modifying the effect until you are satisfied. Display of the image frame or flipbook will update automatically with every modification (unless you choose to turn off this option and update manually).

The Image Viewer has two separate buffers for comparing imagery and provides overlays and inspection tools you can use to evaluate the current image. Some overlays are interactive tools that can be used in lieu of Node Panel parameters to modify imagery.

IN THIS CHAPTER

The spatial resolution of the image displayed is independent of the size of the view frame, which can be resized or maximized, and the viewspace under the view frame is effectively infinite.

By default, the background color of the viewspace, which is visible surrounding the image, is black. You can specify that the Image Viewer use a different color by changing the Outside Image Color preference setting in Edit > Preferences > Settings > Image Viewer. For more information, see "Image Viewer Settings" in chapter 13, p. 150.

Active Area

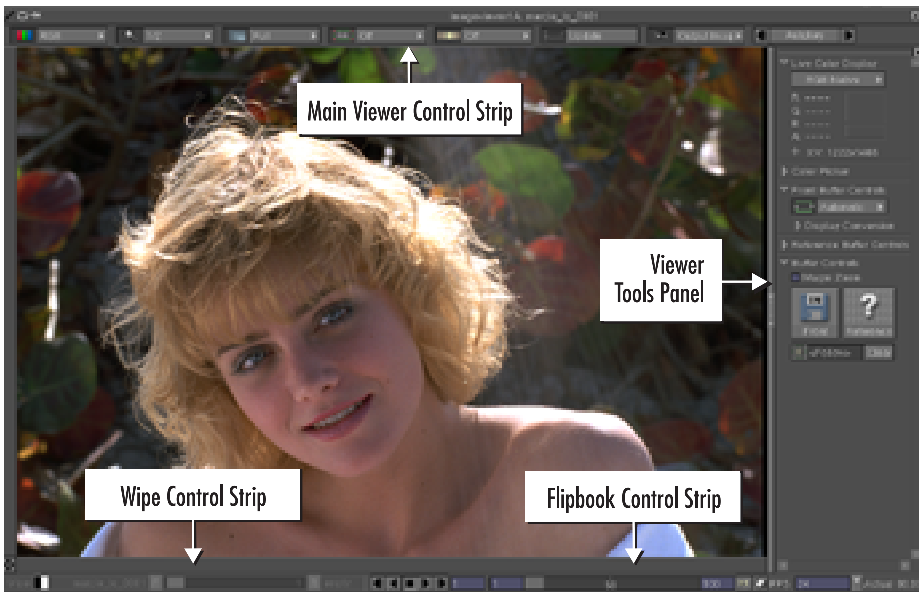
When image pixels are transformed out of frame they are not discarded, which means that they can be transformed back into frame later, and that the frame size itself can be redefined later.

The area that encompasses all pixel data at a point in time is called the *active area* to distinguish it from the output *frame area*. "Viewer Overlays" on p. 76 explains how to turn on visible borders representing the active area and frame area, and *Fig. 6.15* (p. 77) illustrates the concept.

Viewer Tools

The Image Viewer provides several control strips, each with its own set of tools for adjusting how an image is displayed or for getting information about specific pixels in the image:

- The "Main Viewer Controls" (p. 67) are used to select the image channel, size, zoom level, and so forth.
- The "Viewer Tools Panel" (p. 70) includes pixel inspection tools and the image buffer controls.
- The Wipe controls are used for "Comparing Images" (p. 80).
- The Flipbook controls are used to play back image sequences, as described in "Running a Flipbook" (p. 81).



6.1 Tool strips available in the Image Viewer.

Any tool strip can be accessed from the Tools menu in the title bar of the Viewer.

6.2 If an item in the Tools menu of the title bar is checked, the corresponding tool strip will appear in the Viewer.

As with other types of view, you can choose whether or not to display a tool strip and you can reposition a tool strip by dragging it to a different edge of the view frame. If you need more information about this, refer to “Changing the Layout of View Elements” in chapter 4 (p. 34).

The shutter control on the Viewer Tool Panel is especially useful for stowing the panel temporarily when you need more Viewer real estate to display a large image. The shutter control is described and illustrated in “Working with Tool Panels” (ch. 4, p. 35).

Displaying Images

By default, the front buffer (the main buffer) of the Image Viewer displays the RGB data for the node that is currently selected in the Worksheet, at the frame currently specified in the Time Scooter. However, you have a number of options for the display of images in the Viewer.

You can view any channel in the image, or a channel based on luminance values, by selecting it from the “Channel Menu” (p. 67).

When working with large images, you can display them at a lower resolution by selecting one from the “Size Menu” (p. 68). And “ROI” (p. 68) can be used to confine processing to a limited area of a large image.

You can use the middle mouse button to drag the image around when it is larger than the current view frame. In addition, you can resize or maximize the Image Viewer to display more of a large image. You can also “Zoom” (p. 68) in and out.

Full Screen View Mode: To fill the screen with the contents of the viewspace; that is, to maximize the Viewer and black out the view frame so that only the image is visible, switch to Full Screen mode by pressing the **n key**. This hotkey is a toggle; press n again to switch back to the full layout.

TIP:

Press the (numeral) **1** key while the cursor is hovering over a node to launch a new Image Viewer window with that node image displayed in it.

By default, RAYZ will update the image displayed in the Viewer after every parameter modification, but you can specify Manual, Automatic, or Continuous Update mode for each buffer as described in “Front Buffer” (p. 72) and “Reference Buffer” (p. 74).

RAYZ selects the display LUT to use for an image automatically, based on the type of image data it is. However, you can override the automatic setting or add your own custom lookup tables, as described in "Display Conversion" (p. 72).

Displaying Node Input or Output

Depending on the type of node, you also have the option of viewing either the image that will be output by the node or any of the node's input images. This enables you to view the image before and after modification.

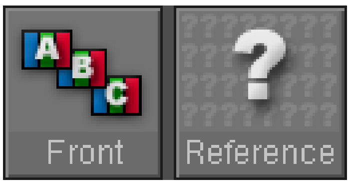
Some nodes actually display an input image by default. In the Ultimatte nodes, for example, you start by using an eyedropper tool to sample an area in the bluescreen backing of the primary input.

Use the "Source Menu" (p. 69), which lists all available options for the current node, to select an input or the output for viewing.

About the Image Buffers

The Image Viewer has two image buffers, the front buffer and the reference buffer. The front buffer is the main buffer and all Viewer functions are available to use with it.

The reference buffer, on the other hand, has a more limited purpose—to provide an image for comparison with the image in the front buffer. In particular, the reference buffer can never follow the current node selection in the Worksheet.



6.3 Front and Reference Buffer Buttons: the reference buffer remains empty (as indicated by the question-mark icon) until you drag and drop a node onto the Reference Buffer button.

The **F8 hotkey** toggles the display back and forth between the front and reference buffer images. Use the Wipe controls to wipe between the two buffers, as described in "Comparing Images" on p. 80.

Pinning an Image to the Front Buffer

You can pin the front buffer to the current node image, if you wish, so that the Viewer no longer follows the node selections in the Worksheet but continues to display the node image to which it is pinned.

To pin the image currently in the buffer, click the Pin icon in the Viewer title bar. To pin a different node image, drag the node over the Image Viewer and drop it (release the mouse button).

To enable the buffer to follow the current node selection again, click the Pin icon in the Viewer title bar. For more information, see also "Dynamic Focus" in chapter 4 (p. 27).

FILLING THE REFERENCE BUFFER

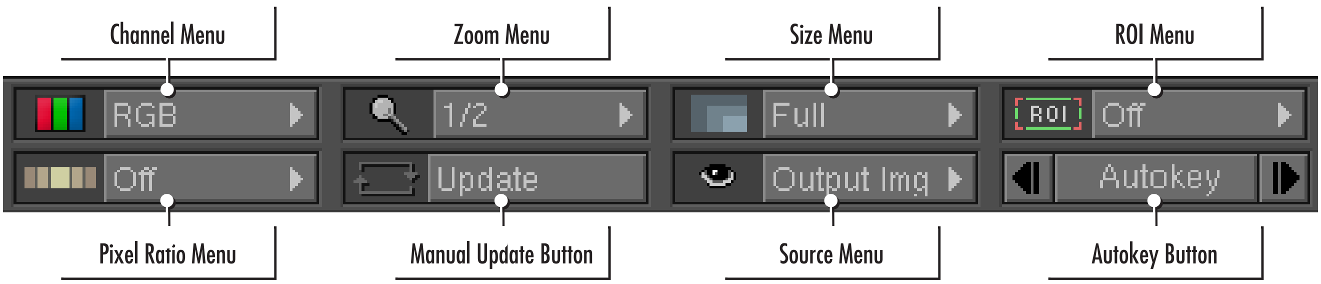
The reference buffer, which is used to hold a separate image that can be compared with the image in the front buffer, is empty by default. Unlike the front buffer, it does not follow the current node selection.

To add an image to the reference buffer, drag a node over the Reference Buffer button in the Viewer Tools panel and release the mouse button. The image will remain in the reference buffer until you clear it by selecting "Clear Buffer" from the Reference Buffer menu.

See also "Buffer Controls" on p. 72.

MAIN VIEWER CONTROLS

The Main Viewer control strip provides core image display tools to specify the channel to display, at what resolution, zoom level, and even pixel ratio when relevant. Most of these tools can also be controlled using hotkeys.



6.4 Controls in the Main Viewer control strip.

FRONT AND REFERENCE BUFFERS To accommodate wipes between the image buffers, the Zoom, Size, and Pixel Ratio settings apply to both buffers.

FRONT BUFFER ONLY The Channel, Source, Manual Update, and ROI controls, on the other hand, can only be used on the front buffer image. The "Reference Buffer" (p. 74) in the Viewer Tools panel provides separate channel selection and update controls for the reference image.

CHANNEL MENU

Use the Channel menu to select which channel or channels of the image to display. The default is RGB, however you can select any individual color channel, luminance, or the alpha channel, if there is one.

HOTKEYS:

With the cursor over the Viewer, press the **c** key for RGB display (think "color"), **a** to view the alpha channel, or **r**, **g**, or **b** for the red, green, or blue channel respectively. For luminance, press the letter **l**. If the image has a fifth channel, as for z-depth data, press the **o** key (for "other").

Zoom

The zoom menu enables you to zoom in on or out of the image by selecting a zoom factor from the menu or using the hotkeys.

HOTKEYS:

Use the Plus (+) and Minus (-) keys to scale the image in discrete increments. Press the Minus key to zoom out of the image and the Plus key to zoom in. (Technically, you use the "equal sign" key to zoom in; that is, you do not have to hold down the Shift key as you press the Minus key.)

Alternatively, you can hold down the Ctrl key while dragging with the middle mouse button. Ctrl-middle-drag up or right to scale up; down or left to scale down.

Size Menu

Use the Size menu to select the display resolution for the image: Full, Medium, Low, or Fit. Full displays the image at full spatial resolution, while Medium and Low scale the image down to specific fractions of full size. Fit scales the image to fit the current size of the Viewer.

By default, Medium is half and Low is a quarter of full resolution. You can change the scale factors in the General Preferences panel, which is described in the section on "Settings" in chapter 13 (p. 149).

If you have proxy image files specified in the Image In node, the Viewer will use them for Medium and Low display. Otherwise the full size imagery will be scaled down. For more information, refer to "Using Proxies in RAYZ" in chapter 10 (p. 126).

ROI

This menu controls ROI, which is used to define a "region of interest." Only pixels inside the region will be processed and updated in the Viewer. Refer to "Defining a Region of Interest" on p. 84 for more information.

Pixel Ratio

The Pixel Ratio menu becomes relevant when you are displaying anamorphic film footage or certain video imagery that appears squeezed in the Viewer. (Refer to "Image Description Parameters" in chapter 14, p. 186 for more information about images with non-square pixel spacing ratios.)

The menu has three choices:

- Off (the default)
- Cheap (unsqueeze using approximate ratio)
- Accurate (unsqueeze using exact pixel ratio specified in Image In node).

Both Cheap and Accurate unsqueeze the image display, and if your anamorphic footage, for example, has a 2:1 pixel ratio they are equivalent choices. If it has a pixel ratio other than 2:1, however, the choice does make a difference. Cheap will be faster but will give you an approximation, while Accurate will take longer to calculate but will display the exact ratio of your imagery.

Manual Update Button

Whenever manual update mode is in effect for the front buffer, the Manual Update button in the Main Viewer control strip is activated. In manual mode, the image is not updated to reflect changes to node parameters until you press the Update button.

HOTKEY:

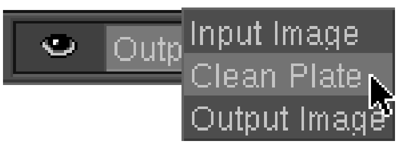
Press the **y key** to update the front buffer image when in Manual Update mode.

To change the update mode, select Manual, Automatic (the default), or Continuous from the Update mode menu located in the "Front Buffer" (p. 72) of the Viewer Tools panel.

Source Menu

The Source menu is used to select whether the Viewer should display the input image or the node output image. For most nodes, the default selection is Output, which shows how the image will look after processing by the current node.

If the node has multiple input images, this menu will list them also. For example, you can switch from viewing the node output to viewing the primary input image or the mask input image.



6.5 Source Menu: This image illustrates the options available for the Ultimatte CSC node, which uses a primary input image and clean plate input image.

HOTKEY:

Press the **s key** to change the current Source menu selection. If the Source menu contains more than two options, you can press the key repeatedly to cycle through all the choices.

Autokey

The Autokey controls are identical to, and linked to, the Autokey function in the Node Panel. They are duplicated in the Viewer for convenience, and they become active whenever a node image that uses overlays is displayed in the Viewer, or whenever the "Node Overlay Buffer" (p. 75) is used. The overlay buffer takes precedence, which means that the Autokey controls apply to the node in the overlay buffer, if any, rather than the node displayed in the front image buffer.

You turn on Autokey mode to tell RAYZ that you are animating parameter values across time. The Autokey arrow buttons navigate directly to the next or previous keyframe. For a complete description, refer to "Using Autokey Mode" in chapter 7 (p. 99).

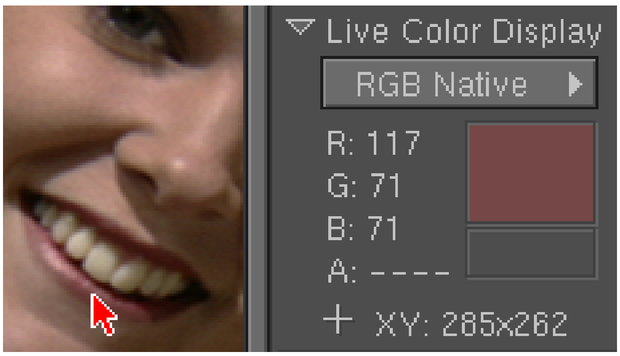
Viewer Tools Panel

The Viewer Tools panel provides interactive image inspection tools such as a pixel value reader and a color picker, which can be used on the front buffer image only. The panel also provides a display conversion toolset and update options for both image buffers. The Viewer Tools menu is also where buffer assignments are controlled.

Live Color Display

This group of pixel inspection tools displays the color value and position of the pixel currently under the cursor in the *front buffer* image. As you move the cursor over the image, the readout updates accordingly:

- Color channel values, including the alpha channel, if any, are shown in the units you specify.
- Position data is shown in *x,y* coordinates. (The lower left corner of the full size frame area is the 0,0 point.)



6.6 Live Color Display shows the RGBA values and *x,y* position of the pixel under the cursor.

By default the values are displayed at native bit depth as RGB data (this is the "RGB Native" option in the menu). However, you can change the colorspace units to use by choosing another option.

Color Picker

The Color Picker group includes an eyedropper tool for interactively picking a color in the image currently being displayed in the *front buffer*. (See also "Buffer Assignments" on p. 75.)

6.7 Color Picker: Use the eyedropper to pick a color from the image using the sample method selected in the associated menu. The color is stored in the selected color swatch. The RGB values are displayed at the bottom in the units selected from the Readout menu.

The menu next to the eyedropper is used to specify how and what to sample. The sampled value is stored in the selected color swatch, and there are 8 swatches in which you can store color values.

EYEDROPPER TOOL

Use the eyedropper tool to pick a pixel:

1. Select a sample method from the menu (see below).
2. Click the eyedropper once to select it. (The cursor will change to an eyedropper icon.)
3. Move the eyedropper cursor over the pixel(s) you want to pick and click, scrub, or drag, depending on your selection in step 1.

The color you sampled will appear in the selected color swatch.

SAMPLE METHOD MENU

Use this menu to select the sample method to use. There are two sample methods: Click/Scrub and Drag Box. And for each sample method you can choose whether to get the average value, the maximum value, or the minimum value of the pixels you sample:

- The **Click/Scrub** method lets you click an individual pixel or hold down the mouse as you scrub over an area. Choose Scrub/Avg (the default), Scrub/Min, or Scrub/Max.
- The **Drag Box** method lets you drag a bounding box around the pixels to be sampled. Choose Drag/Avg, Drag/Min, or Drag/Max.

COLOR SWATCHES

The Color Picker also includes a group of color swatches, which work like all other color swatch palettes in RAYZ. When you use the eyedropper tool to sample the image, the color is stored in whichever swatch is currently selected, replacing the previous color. Click any swatch to display its color values in the readout.

You can also change the value stored in a swatch by right-holding on the swatch to access the popup spectrum bar. Drag the cursor across the spectrum to select a color and release the mouse button.

TIP:

You can change the default colors used for a project, as when it would be useful to store project-specific reference values in some of the swatches. Choose Project Settings from the Edit menu, expand the Current Colors group, and select Image Sample Color Palette to display a list of swatches. Then select any swatch to edit it.

READOUT

Use the Readout menu to select the colorspace units in which the selected color values should be displayed. The default is RGB Native, which displays the RGB values in the native bit depth of the image, but you can also choose RGB Float or RGB Log. Additionally, you can choose to display the values in their equivalent units in HSL, HSB, HWB, YIQ, or YUV.

BUFFER CONTROLS

The Viewer Tools panel includes display controls for the front and reference image buffers, as well as the buffer assignment tools.

FRONT BUFFER

6.8 Front Buffer Controls.

This group of controls is used to set the image update mode and type of display conversion to use for the front image buffer.

IMAGE UPDATE MODE

By default, the image in the front buffer updates automatically; however, you can use the menu to select Continuous updates or Manual updates instead:

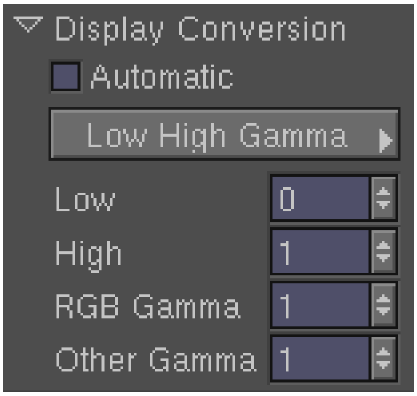
- **Automatic** will update the image after each modification.
- **Continuous** will update the image continuously; that is, in real time as a change is made (for example, as a slider is being dragged).
- **Manual** specifies that the image is never updated unless you press the Manual Update button in the Main Viewer tool strip (or press the manual update **hotkey: y**).

HOTKEYS:

To switch the front buffer to Auto Update mode, press **Shift-a**; to switch to Continuous Update, press **Shift-c**.

DISPLAY CONVERSION

These parameters control how the image data in the associated buffer is interpreted for display in the Image Viewer. It is never strictly necessary to adjust these parameters, because RAYZ selects the appropriate type of display translation automatically, based on the data type (linear or log) and any custom gamma information you have specified in the Image In node. (For more information on these Image In node settings, refer to the section on "Conversion Parameters" in chapter 14, p. 181.)



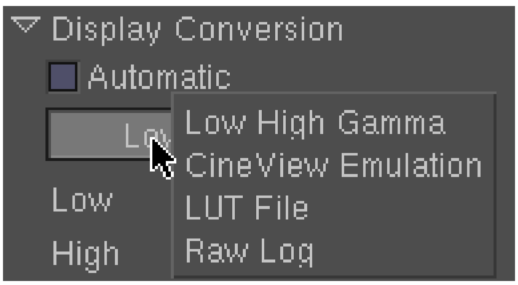
6.9 Display Conversion parameters set to manual mode, with Low High Gamma parameters available for modification.

Cineon Log Display

By default, the Image Viewer displays log image data using **Raw Log**. The Raw Log display LUT simply shifts the log values up in the linear colorspace to make them visible; it does not attempt to remap the nonlinear distribution curve. Log images displayed this way tend to look washed out in the Viewer.

You can also display a log image using **Cineview Emulation**, which will give you a better idea of what the image "should" look like, by turning off Automatic display conversion and selecting the Cineview option from the Display Conversion menu. See *Fig. 6.10* below.

Cineon imagery that has been converted to a linear format is displayed by default using the Low High Gamma settings, with the RGB Gamma set to the corresponding value that was specified when the image was converted (which is 1.7 by default).



6.10 The Display Conversion menu includes Cineview and Raw Log display emulation.

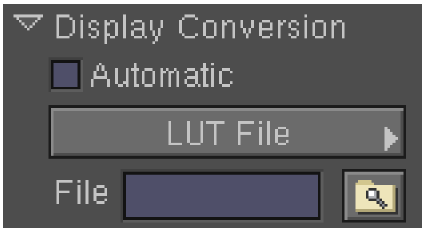
Automatic checkbox

You can override the current display settings by turning off the Automatic checkbox. This activates parameters you can use to select a specific LUT (lookup table) or other type of display emulation, or to manually adjust the Low, High, RGB Gamma, and Other Gamma display values.

RGB and Other Gamma

The RGB Gamma setting affects the display of the RGB channels only, while the Other Gamma setting affects the gamma value used to display the Alpha or other channel, if the image has one.

LUTs



6.11 The File parameter appears when LUT File is selected in the Display Conversion menu.

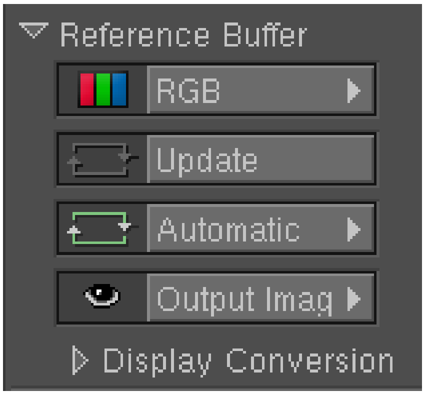
Select LUT File from the Display Conversion menu (uncheck the Automatic box to access) to use an existing LUT file that is not in the menu list. The LUT File parameter will become active, in which you can specify the pathname of the LUT file you want to use.

LUT files should be plain text files that follow the guidelines in "Appendix F: LUTS" (p. 471).

CUSTOM LUTS: You can also create custom LUTs for use in RAYZ by writing a plugin of the appropriate type, which will then appear in the Display Conversion menu. For an example, refer to the file CPI_LUTProvider.h, which is located in the following directory:

/usr/grail/rayz2.2/support/plugIns/include

Reference Buffer



6.12 Reference Buffer Controls.

This group of controls is used to select the source image and channel to display, and to set the image update mode and display conversion for the reference image buffer.

The **Channel menu** for the reference buffer works just like the one in the Main Viewer Control strip that is used for the front buffer image. The default is to display the RGB image, however, you can select any channel in the image for display.

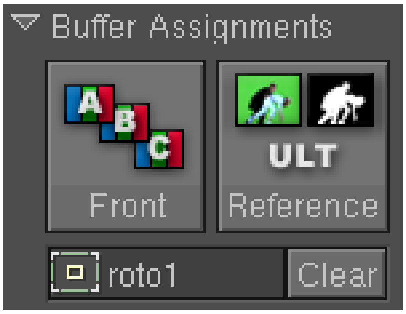
The **Source menu** for the reference buffer (eye icon) also works like the one use for the front buffer image (see "Source Menu" on p. 69). The output image is displayed by default.

The **Image Update mode menu** is also identical to the one used for the front image buffer (see "Image Update Mode" on p. 72) to select Automatic, Continuous, or Manual update modes.

If you select Manual update mode for the reference image, however, use the **Update button** in the Reference Buffer controls to update the reference image manually. The Update button in the Main Viewer control strip is for the front buffer only. Also, be aware that the hotkeys for updating apply only to image in the Front buffer.

The Reference Buffer control group also includes the same tools for "Display Conversion" (p. 72) that are included in the Front Buffer controls.

Buffer Assignments



6.13 Buffer Assignment Controls: This example shows a Multi-comp node image in the front buffer, an Ultimatte image in the reference buffer, and the Roto node overlays in the overlay buffer.

The Buffer Assignment buttons show which node image or overlay, if any, is currently in the front image buffer, reference image buffer, and overlay buffer. The Reference Buffer and Node Overlay Buffer buttons also act as targets onto which you drag and drop the node whose image or overlay you want to add to that buffer.

Front Image Buffer

The Front Buffer button indicates which node image is currently displayed in the front image buffer, and it updates automatically as you select different nodes in the Worksheet (unless you have pinned the image, as described in "Pinning an Image to the Front Buffer" on p. 66).

Reference Image Buffer

To add an image to the reference buffer, drag and drop a node over the Reference Buffer button. To assign a different image to the reference buffer, drag and drop a new node into it. To empty the reference buffer, select Clear Buffer from the popup menu (hold down the buffer button to access).

The reference buffer can hold a different node image than the one in the Front buffer, or it can hold the same node image, so that you can wipe between

- two different node images,
- the input and output images of the same node, or
- the same node image before and after changing parameter values.

See "Comparing Images" on p. 80 for a description of the Wipe controls and how to perform the types of wipe listed above.

Node Overlay Buffer

Many nodes feature overlays and other controls that appear in the Viewer whenever the node is selected. (See also "Node-Specific Controls and Overlays" on p. 79.) In some cases, however, you may want to manipulate a node overlay while viewing an image from another node. For example, you might want to adjust a spline created in a Roto node to fix a foreground matte while viewing the image as it appears composited over the background in a Multi-comp node.

The node overlay buffer is designed for just that purpose. You can drag and drop any node that uses overlays onto the Node Overlay Buffer button to make the overlay available for use in the Viewer when the node image is not currently being displayed.

The overlay buffer is empty by default. When you drag-and-drop a node into it, the node name appears in the buffer. To assign a different node overlay to the buffer, drag and drop the new node onto the Node Overlay Buffer. To clear the overlay buffer, press the Clear button on the right.

NOTE:

When a node that features overlays or other node-specific Image Viewer controls is displayed in the front buffer, its overlay controls appear in the Viewer automatically— it is not necessary to assign the node to the overlay buffer; in fact, the node overlay buffer remains empty in this case.

Viewer Overlays

You can turn various image overlays on and off in the Image Viewer. An overlay is an object that is drawn over the image, such as borders that delineate the title safe areas of the frame or the output frame area itself (see "Border Display Overlays" on p. 77).

Some overlays, such as the "Image Profile Overlay" (p. 78) and "Magnizoom Overlay" (p. 78), update dynamically as you move the cursor.

Two overlays are used to edit boundaries—one defines ROI (see "Defining a Region of Interest" on p. 84) and the other defines the area used for a "Region Wipe" (p. 80).

6.14 The Viewer Actions popup menu is used to turn the display of any overlay on or off.

The overlay display controls are in the Viewer Actions menu, which is accessed by holding down the right mouse button anywhere in the viewspace.

The Node Controls item at the top of the menu controls the display of node-specific overlays, which vary depending on the node image being viewed, as explained in "Node-Specific Controls and Overlays" (p. 79).

As an alternative to using the Viewer Actions menu, you can define a hotkey to control the display of any overlay by selecting the overlay in the Hotkeys list in Edit > Preferences. For more information, refer to "Hotkeys" in chapter 13, p. 152.

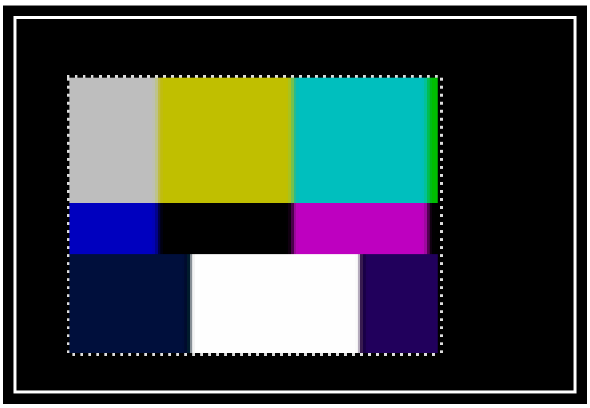
TIP:

You can change the default colors used to draw the overlays if necessary to make them stand out better over your imagery.

Select Project Settings from the RAYZ Edit menu and expand the Current Colors group in the left pane to select the Overlay Gadget Color Palette preferences. Then you can edit the default color values for any of the overlay components listed in the right pane.

Border Display Overlays

Some overlays delineate the borders of the frame or other areas of the image. You can display a border around the **Full Size** area (the frame output) and the **Active Area** (a rectangle encompassing all pixels with nonzero values). This concept is explained in "Active Area" on p. 64.



6.15 The Full Size overlay (solid white line) delineates the output frame area, while the Active Area overlay (dotted line) delineates the area that includes pixels subject to modification.

Video Safe

The Video Safe overlay outlines the title safe and action safe areas of a video frame.

Field Chart

The Field Chart overlay can be used to conform titles to film formats with a 4:3 aspect ratio by referencing the 12 standard field sizes demarcated on the overlay grid. In addition, several other aspect ratios are also outlined on the chart.

1.85 Mask

The **1**.85 Mask overlay masks out the top and bottom of an Academy aperture image so that you can see what will be lost in so-called flat widescreen format, which has a 1.85 aspect ratio.

Magni-zoom Overlay

6.16 Magni-zoom Overlay.

In effect, Magni-zoom turns the cursor into a magnifying glass that provides a close-up view of a rectangular area centered under the cursor.

The Magni-zoom overlay moves with the cursor and enables you to examine image areas in detail without zooming the entire image.

HOTKEY:

You can use the **m hotkey** to toggle the Magni-zoom overlay on and off.

Magni-zoom will magnify whatever pixels are under the magnification box, whether the image displayed is in the front or reference buffer.

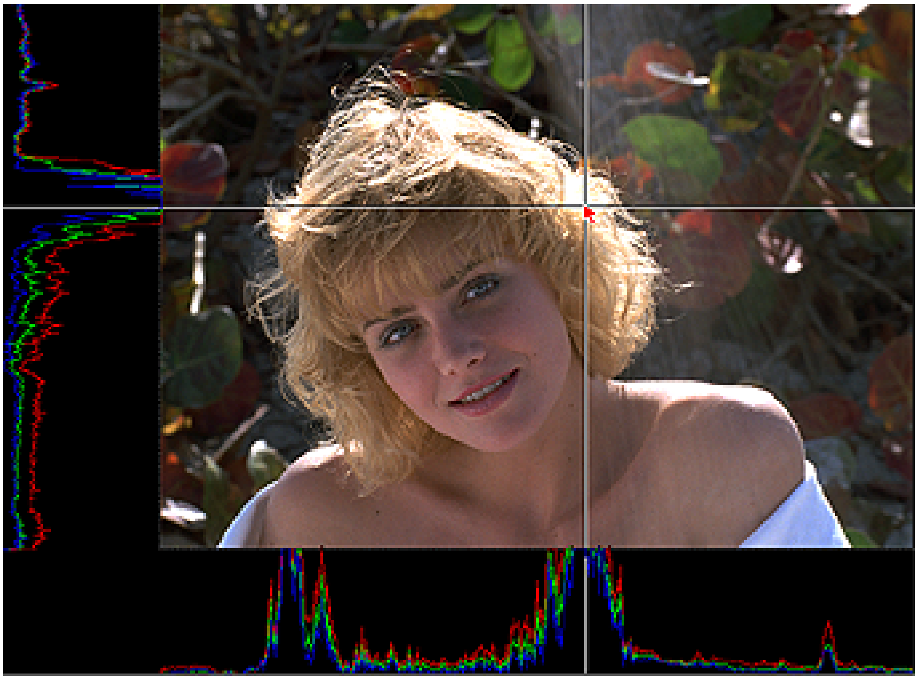
TIP:

Magni-zoom works with interactive node overlays, which means that you can turn this feature on as you manipulate a roto spline handle or transform widget, if necessary, to make it easier to see and adjust.

You can change the level of magnification and the size of the magni-zoom area in Edit > Preferences > Settings > Image Viewer. (For general information about changing preference settings, see also "Editing General Preferences" in chapter 13, p. 148.)

Image Profile Overlay

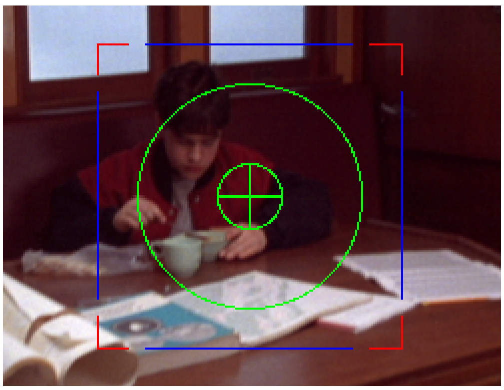
The Image Profile overlay is a graphical representation of image values that updates dynamically as you move the cursor over the image. This overlay graphs the values each channel of each pixel in a horizontal and vertical "slice" along the axis currently under the cursor.



6.17 The Image Profile overlay generates a dynamic graph of the pixel values. Note the hot spot identified in the highlights on the model's hair.

Node-Specific Controls and Overlays

Some overlays are node-specific; that is, they are only applicable if a particular node image is being displayed. Most node overlays are interactive tools. They may have handles that can be manipulated by dragging or they may be accompanied by a control strip.



6.18 The Transform widget is a typical type of node overlay.

For example, the interactive transform overlay is available in nodes that perform spatial transformations.

And the Roto control strip, which contains the mode buttons used to draw and edit roto shapes, is available when a Roto node image is being displayed. The roto splines themselves are node overlays.

The use of each specific node control strip or overlay is explained in the individual description of that node in *Part III: Node Reference* (p. 159).

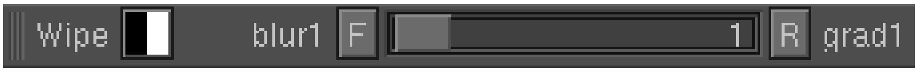
Comparing Images

In addition to the main image buffer (the front buffer), the Image Viewer provides an additional reference buffer to use for comparing two different images. (See also “About the Image Buffers” on p. 66.)

Whenever you have an image in both buffers, you can toggle back and forth between them using the F8 function key. The Wipe control strip also becomes active, however, so that you can perform various wipes between the two buffers.

Wipe Controls

To wipe between the front and reference buffer image, just drag the slider. The farther left you drag the slider, the more of the front image you see; the farther right you drag it, the more you see of the reference image.



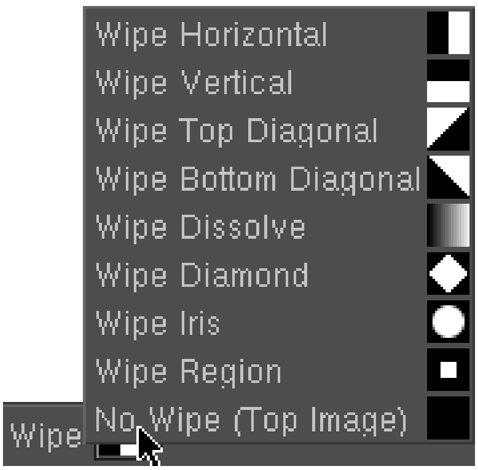
6.19 Wipe Controls: The image in the front buffer (“blur1”) is labeled on the left side of the slider; the reference buffer image (“grad1”) is labeled on the right side.

Wipe Buffer Buttons: F and R

When you have a wipe position set, you can temporarily display the entire image in the front or reference buffer without resetting the wipe point: Hold down the F button to see the front buffer image; hold down the R button to see the reference buffer image. When you release the mouse button, the wipe image will reappear.

Wipe Types

A horizontal wipe is selected by default, but you can also select vertical, top diagonal, bottom diagonal, dissolve, diamond, iris, and region, or no wipe at all.



6.20 Hold down the Wipe button to access the menu and select the type of wipe to use.

Region Wipe

This type of wipe enables you to drag a rectangular “hole” over the top image through which you can see the bottom image.

When you select Region Wipe from the Wipe menu, a crop box overlay appears. Drag the sides or corners of the box to resize it to suit your current need.

To wipe, just click anywhere inside the region and drag.

You can hide the overlay box without turning off the region wipe. Right-hold in the viewspace to access the Viewer Actions menu and deselect Edit Wipe Region. If you need to change the region area, just reselect it.

Wiping Between a Node Input and Output

You can wipe between a node's output image and its input image:

1. Put the same node that is in the front buffer into the reference buffer (as described in "Filling the Reference Buffer" on p. 67).
2. Use the "Source Menu" (p. 69) to switch the front buffer display to the input image.

Then you can use any of the wipe methods described in "Wipe Controls" (p. 80) to wipe between them.

History Wipes

You can wipe between the same node image before and after modifying a node parameter:

1. Put the same node that is in the front buffer into the reference buffer (as described in "Filling the Reference Buffer" on p. 67).
2. Switch the Update mode for the reference buffer to Manual so that it will not update unless you force it to do so. The Update menu for the reference buffer is located in the Viewer Tools panel, as illustrated in *Fig. 6.12* (p. 74).

Then you can make further changes to parameter values and see the result in the front buffer image, while the reference buffer image retains the "before" image.

Running a Flipbook

In addition to displaying individual image frames, you can also play entire sequences (flipbooks) of node imagery in any Image Viewer.

Playing a sequence is as simple as pressing the Play button. All channels are cached, which means that you can still switch from, say, RGB to alpha channel display as the flipbook is running.

If the reference buffer also contains an image sequence, it will be played also so that you can wipe between the sequences as they play, assuming you have enough RAM. If you do not, you can always clear the reference buffer before pressing Play or accept slower playback.

About Caching and Playback

Depending on the size and number of frames, and the system on which RAYZ is running, you may have to wait until all of the frame images are cached into RAM the first time before the sequence plays back at full speed. Frames cached in RAM are indicated by blue markers running along the top of the Flipbook slider, as illustrated in *Fig. 6.24* (p. 83).

You can change the zoom level without affecting the cache, but changing the setting in the Size menu will cause the sequence to be recalculated.

NOTE:

Unless the Image Viewer is pinned to the cached sequence, the cache will be cleared when you select another node in the Worksheet and you will have to re-cache the sequence to play it again.

To avoid this, you can pin the Viewer containing the flipbook and create another Image Viewer to display other nodes.

Disk Caching

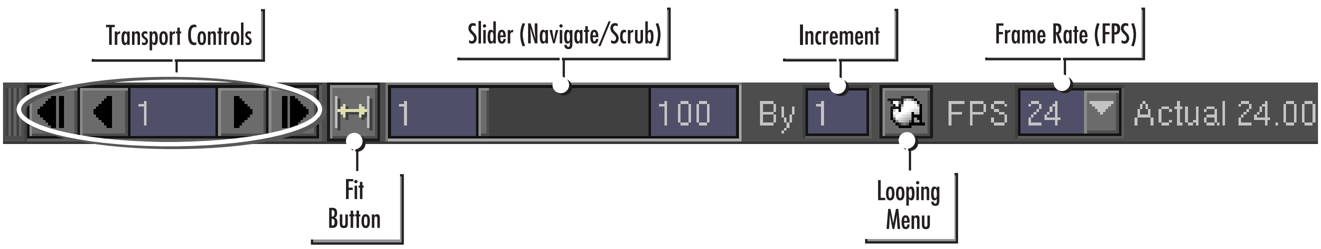
If RAYZ has to resort to disk caching, you may not achieve real-time playback. The actual playback rate is indicated in a readout to the right of the FPS parameter (where you specify the desired playback rate).

Continuous Updates

In continuous update mode, you can modify node parameters while the flipbook is playing and the sequence will update as it plays. It will attempt to update at the frame rate specified in the FPS parameter, but the actual speed will depend on the imagery, the operation, and your hardware.

Flipbook Control Strip

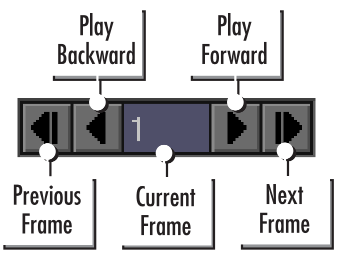
You can specify which frames to play at what increment and frame rate, and you can play the sequence once or continuously, forward or backward, or scrub using the controls available in the Flipbook control strip.



6.21 Flipbook Control Strip.

Transport Controls

The Transport control buttons are used to start and stop playback.



6.22 Transport Control Buttons.

The Play Forward and Play Backward buttons play the entire sequence. Press the same Play button again to stop playback. (The Play button turns into a red Stop icon while a sequence is playing.)

The sequence plays once or continuously based on the type of looping specified, as well as the increment and FPS settings.

HOTKEYS:

You can use the **j** and **k hotkeys** to control playback. The k key is equivalent to Play Forward and the j key, to Play Backward. These keys are toggles, so to stop playback simply press the hotkey again.

You can also use the Flipbook Transport controls in lieu of the frame navigation buttons in the Time Scooter. The Next Frame and Previous Frame buttons to step forward or back a frame at a time. To go directly to any specific frame, type the frame number into the Current Frame field.

HOTKEYS:

Use the **,** and **. hotkeys** to go to the next (period key) or previous (comma key) frame. **Shift-,** takes you to the first frame; **Shift-.** takes you to the last frame.

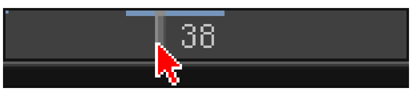
Slider Controls

You can drag the slider bar across the Flipbook slider to navigate through a sequence of frames. You can also simply move the mouse over the slider: As you move, the number of the frame that would be selected if you clicked that location appears next to the cursor; when the frame number you want appears, click on the slider to go to that frame.

6.23 Flipbook Slider.

Using the Slider to Scrub

Drag back and forth in the Flipbook slider to scrub in the sequence. Once the image data for all the frames you are scrubbing has been cached into RAM, you will get a smooth effect.



6.24 Frames that have been cached into RAM are indicated by blue tick marks at the top of the slider.

Frame Range

The First Frame and Last Frame fields, located at either end of the slider, are used to specify the range of frames to play. They default to the global time defined in the Time Scooter, but you can change the range by typing new values into the Start and End fields or by using the Fit Range button.

6.25 Fit Range button (far left) can be used to set the Start and End field values.

Fit Button Press the Fit button to set the frame range to match the length of the node sequence currently displayed in the Viewer.

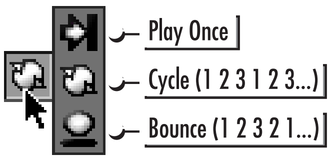
Increment

The Increment value is set to 1 by default, which includes each frame in the playback. If you set the increment to 2, every other frame will play, and so on.

The hotkeys used to go to the next (Period key) or previous (Comma key) frame follow the current increment setting. That is, pressing the "go to next frame hotkey" will take you to the next frame available at the current increment (assuming the cursor is over the Image Viewer).

Loops

This menu is used to specify whether the sequence should play once or continuously when you press a Play button.



6.26 Options available in the Loops menu include Play Once (no looping), Cycle loops, and Bounce loops.

There are two styles of continuous playback, or loop, to choose from:

- **Cycle:** play the first frame to the last, then start over at the first frame and play the sequence again, repeating the cycle until the Stop button is pressed. Cycle is the default playback setting.
- **Bounce:** play forward and back continuously from the first frame to the last, then last to first, then first to last, and so on until the Stop button is pressed.

Frame Rate (FPS)

This data entry field is used to specify playback speed in frames per second. The default is 24fps, but you can type a different value (such as 30fps for video) in the field or use the FPS menu to select 12, 24, 30 or 60.

You can change the default FPS value to match your project as described in "Editing Project Settings" in chapter 13, p. 156.

Actual Rate

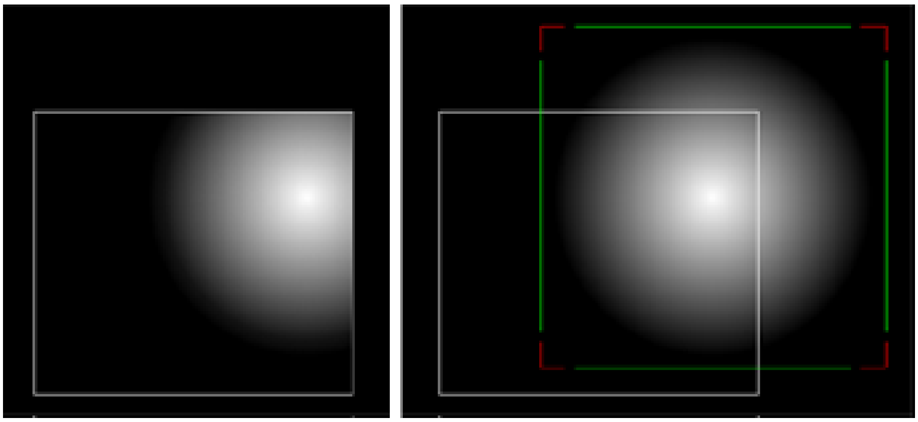
The first time you play a sequence, playback may be slower than the specified rate as RAYZ computes the frames and caches the data in RAM. Subsequent loops will play at full speed, assuming you have enough RAM to hold all of the frames. If not, RAYZ will resort to disk caching, and you may not achieve realtime playback.

The "Actual" readout to the right of the FPS field shows the actual playback rate achieved.

Defining a Region of Interest

ROI, or region of interest, is used to define an area of an image for display and updating in the Image Viewer. Using ROI can save time when working with large format imagery because RAYZ will not update the entire image display, just the region you need to evaluate:

- ROI controls the image display in both buffers (see also "About the Image Buffers" on p. 66).
- ROI does not affect rendering of the final imagery to disk, only the display in the Viewer.
- The region of interest can be any region within the active area, even areas that fall outside of the frame area. For example, if you are transforming an image outside of the frame boundary, you can use ROI to view the pixels that have already left the frame, as shown in *Fig. 6.27*.



6.27 Sphere is being transformed out of frame, as indicated by the white box outlining the frame border (left). ROI is used (right) to define entire active area as region of interest, so that even those pixels outside the frame border are displayed.

Image areas outside the region of interest will still be visible when you switch to ROI mode, but they will not update as node parameters are modified. This enables you to use the entire image as a reference when defining and updating a region.

USING THE ROI CONTROLS

1. Select Edit from the ROI menu to display the overlay used to define the region of interest.
2. Drag the borders to resize the box; drag anywhere inside the borders to reposition it over the image.
3. Then, optionally, you can hide the overlay while remaining in ROI mode by selecting Active from the ROI menu.
4. To return to normal display mode, Select Off from the ROI menu.

EDIT MODE

When the ROI menu is set to Edit mode, a bounding box overlay appears on the image. (You can also use the Viewer Actions menu to turn the ROI Edit overlay on and off without turning off ROI mode.)

6.28 The ROI menu is located in the Main Viewer Control strip.

To define the region of interest, drag the overlay edges or corners to resize the bounding box and drag anywhere inside the box to reposition it over the image.

Active Mode

If you find the overlay distracting, you can switch the ROI menu to Active mode, which hides the overlay box while remaining in ROI mode. If you need to adjust the region of interest, you can always switch back to Edit mode.

chapter 7

USING THE NODE PANEL

Node Panels provide access to the various parameters and other controls available to modify the operation of a node. The specific functions and uses of the individual nodes are described in *Part III: Node Reference* (p. 159); this chapter provides an overview of the Node Panel view and the types of controls common to most nodes.

In addition to setting node-specific parameters, you can use the Node Panel to

- rename a node (see "Name Field" on p. 88).
- create reusable presets for any node type (see "Presets Menu" on p. 88).
- disable a node temporarily (see "Pass Thru" on p. 91).
- add spare parameters (see "X-Parm" on p. 90).
- turn on auto-keyframing mode (see "Using Autokey Mode" on p. 99).

By default, the Node Panel view displays the data for the currently selected node in the Worksheet, at the frame currently specified in the Time Scooter. However, you can pin a Node Panel to a specific node, if you wish, so that the Node Panel no longer follows the current selection but continues to display the parameters for that node. For more information, see "Dynamic Focus" in chapter 4 (p. 27).

TIP:

Press the (numeral) **2** key while the cursor is positioned over a node to launch a new Node Panel window with that node data displayed in it.

Node Panel Control Strips

The top portion of the Node Panel includes two control strips: the Name strip and the Controls strip. Both strips are displayed in the Node Panel by default, however, you can turn their display on and off in the Tools menu of the Node Panel title bar.

7.1 The Name strip and Controls strip available at the top of every Node Panel.

Name Strip

The Name strip is labeled with the node type (Image In, Blur, etc.) and includes a text field with the name that identifies an individual node of that type. It is also home to the Presets menu.

Name Field

RAYZ assigns each new node a name by default ("blur1," "blur2," etc.). However, you can rename a node at any time by typing the new name into the field and pressing the Enter key. It is best to keep node names short but descriptive.

NOTE:

It is advisable for users on all systems to conform to standard UNIX nomenclature when naming nodes. Do not use spaces or special characters such as the semicolon (;), backslash (\), ampersand (&), exclamation point (!), asterisk (*), or pipe (|).

Presets Menu

7.2 Presets menu.

The node Presets menu enables you to save specific parameter settings to a Presets file and reload the settings into other nodes of the same type. This can save you from having to remember and reassign a complex set of parameter values that you want to reuse.

Presets can be used in different sessions and different RAYZ files. For example, if you save a Blur node preset you can load it into any Blur node you create in any RAYZ file.

Another way to duplicate all the parameter settings for a node is to clone the node itself, as described in "Creating Clones" in chapter 5 (p. 54). Presets are used instead of cloning when you want to start with the same settings but still be able to make further adjustments to parameter values separately from the original node.

Save as User Default

The Presets menu also contains a Save As User Default option, which saves the current parameter settings as the new User Default for the current node type. Use this when you want each new node of the same type to start off with these parameter settings.

To restore the RAYZ defaults for new nodes, select Project Defaults from the Presets menu and then select Save As User Default again.

Saving Presets

To save a node preset:

1. Select Save As… from the Presets menu.
2. Enter an appropriate name into the dialog box that appears and click the OK button.

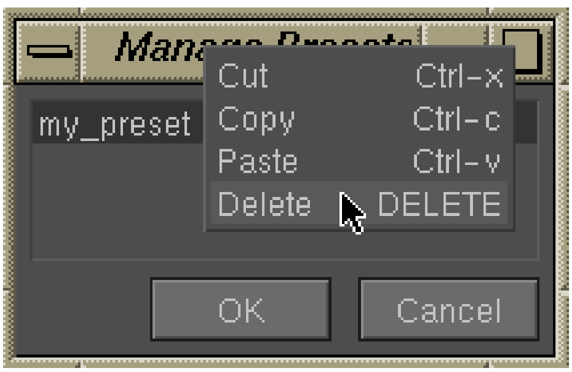
The new preset will appear in the Presets menu list the next time you pull down the Presets menu for any node of the same type.

Loading Presets

To load a node preset, select it from the Presets menu.

Managing Presets

Once you have created a node preset, a new item is added to the Presets menu: Manage Custom.



7.3 Click once on a preset in the list to select it: drag to reorder (if multiple presets exist); right-hold to access Cut/Copy/Paste/Delete commands.

Select Manage Custom from the Presets menu to access a panel in which you can **rename**, **delete**, or **reorder** any preset in the list.

Controls Strip

The Controls strip has the Autokey, X-Parm, and Pass Thru mode controls. X-Parm and Pass Thru are described next; Autokey is described in the section on "Animating Parameter Values" (p. 99).

X-Parm

Click the X-Parm button, which stands for "Extra Parameter," to bring up the parameter editor, in which you can add spare parameters to the Node Panel. Extra parameters can be used to create a master, or gang, control for other parameters by using an expression to reference the spare parameter in the other parameters.

For example, if you had several Brightness nodes that performed the same operation on a different input, you could name the spare parameter "Master_Brightness" and reference it in the Brightness node parameters. Then when you adjusted the Master_Brightness value, all of the Brightness parameters would update accordingly.

You could use clones of a Brightness node for this purpose, but the other Brightness node parameters could not be set separately the way that they can when a spare parameter is used.

Another advantage of using a spare parameter is that, unlike using clones, you can still modify each Brightness parameter individually if necessary: one Brightness expression could reference the spare parameter value as is, another could multiply it by 0.8, and another by 1.2. All the Brightness nodes would change proportionally when you adjusted the Master_Brightness parameter, although each would have a different numerical value.

Using the X-Parm Parameter Editor

To create an extra parameter, press the Add New Parameters button in the parameter editor and a spare parameter entry will appear at the top of the panel.

7.4 The parameter editor, with Type menu displayed.

You can edit the entry to specify its characteristics: what type of parameter it will be, what the default value will be, and how it will be labeled in the Node Panel:

- Select Float or Integer from the Type menu to create a numeric parameter field with accompanying slider bar control. You can also specify the default value in the field and the range of the slider.
- Select String to create a text field, for which you can also specify the default string.
- To rename a spare parameter, double-click on the default name and type the new name into the text field.

Pass Thru

Check this box to put the node into Pass Thru mode, or use the **p hotkey** equivalent. This turns off the node so that the incoming image data flows through it without being modified. This is convenient when you want to disable the node temporarily without actually disconnecting it from the data flow and rerouting the connections around it.

The hotkey for Pass Thru mode is the **p key**. (You can also use the hotkey to invoke Pass Thru in the Worksheet; just be sure to position the cursor over the node you want to affect before pressing it.)

NOTE:

Pass Thru is disabled in nodes without input connectors such as the Source nodes, which start a dataflow and thus have no upstream data to pass through.

Types of Parameter Controls

The workspace of the Node Panel displays parameters that are specific to each node operation, but the controls used to set these parameter values are common to most nodes: menus, checkboxes, buttons, data entry fields, and slider bars. See “Checkboxes and Toggle Buttons” (p. 95), and “Data Entry Parameters” (p. 96).

Related parameters may be grouped together in outline format, in which case outline arrows are used to expand and collapse each group. See “Parameter Groups” (p. 92).

Many nodes feature a common group of parameters used to specify color values by entering numerical values, selecting a color interactively using a spectrum bar, or by sampling image pixels with an eyedropper. For a detailed description, see “Using the Color Parameters” in chapter 14, p. 168.

Some parameter groups are dynamic; that is, they are created as necessary based on user input. For example, a new set of parameters is added to the

Node Panel every time you add another input layer to the Multi-comp node. See “Dynamic Parameter Groups” (p. 93).

An individual parameter or group may not be available in all circumstances. For example, an alpha channel control will be deactivated when the image being manipulated does not have an alpha channel. Whatever the reason, a parameter control will appear grayed out whenever it is unavailable.

TIP:

Whatever type of control is used, you can always undo and redo multiple changes to parameter values, as explained in the section on the “Edit Menu” in chapter 9, p. 118.

You also have the option of changing default values for many node parameters in Edit > Project Settings > Node Defaults. For more information about using the Project Settings panel, refer to *Chapter 13: Setting Preferences* (p. 147).

Parameter Groups

Related sets of parameters are often grouped together in the Node Panel using an outline type of display format. Groups of parameters are collapsed under a descriptive heading, or under a master-control parameter for the group.

A parameter group can be identified by the outline arrow on the left side of the parameter label, which is used to expand and collapse the group. Click this arrow to expand a collapsed group and to collapse an expanded group.

Parameter groups enable you to hide rarely used or highly complex parameters and still access them easily if you need them. They reduce visual clutter and enable more parameters to be visible in a Node Panel view without scrolling.

7.5 Brightness parameter group has been expanded to access the individual channel controls.

CHANNEL GROUPS

Any parameter in RAYZ that affects RGB values can be controlled individually or collectively using channel group parameters. You can adjust, say, brightness with a single master control that affects the RGB channels equally.

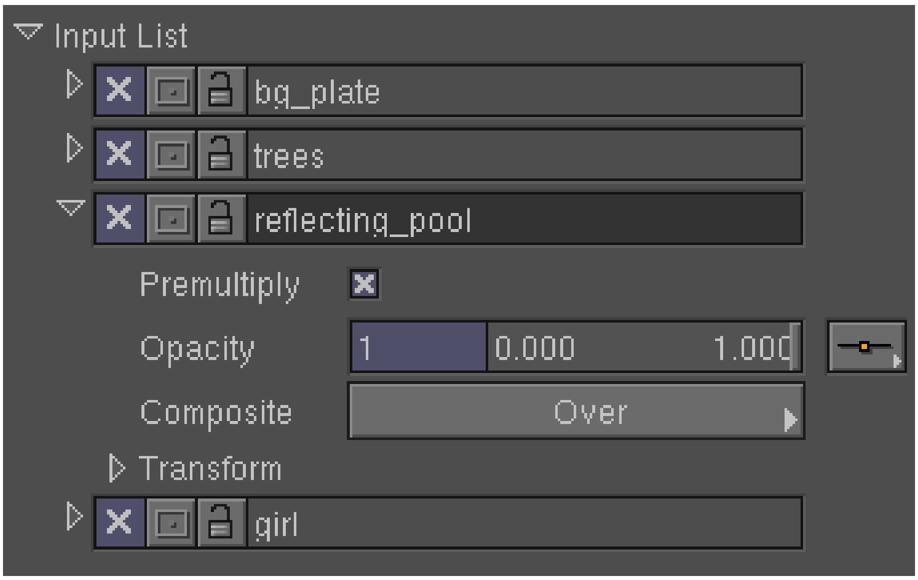
But you can also use the outline arrow to expand the group to access separate brightness controls for each individual color channel, as well as the alpha channel and a fifth image channel, labeled Other, if the image contains these channels.

NOTE:

When you adjust an individual color channel, you are making an incremental change relative to the master value. The channel parameter values in a channel group are always stored as deltas of the master parameter value.

DYNAMIC PARAMETER GROUPS

Some parameters are generated by RAYZ only in response to user action. In the Roto Node Panel, for example, a parameter group is created for each roto shape you draw in the Image Viewer. In the Multi-comp node, a new group is created for each input you connect. And in the Track node, a new group is created for each track point you add.

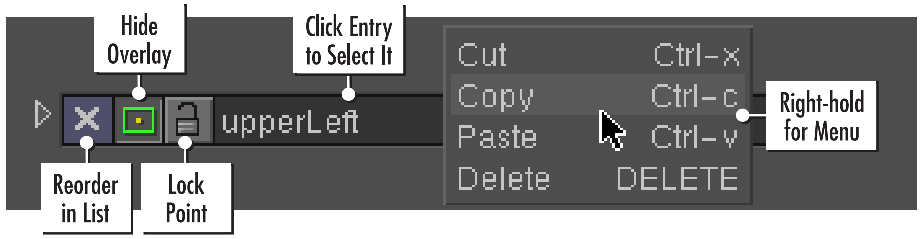


7.6 List of input layers to a Multi-comp node. Each layer entry is generated dynamically when the corresponding input image is connected to the Multi-comp node. The "reflecting pool" entry has been expanded to access additional controls.

These dynamic parameter groups are displayed in the Node Panel in a list. Each group can be expanded to access parameters for that object or input layer. In Multi-comp, for example, you would expand a layer group to specify, among other things, the opacity of that layer of the composite. In the Track Node Panel, on the other hand, you could adjust the size of the tracking area or the tracking method to use.

At their top level, dynamic parameter groups feature a set of controls that affect the object or input layer as a whole.

7.7 Top-level controls available for a track point entry in a Track node.



Changing Layer Order

You may want to change the order of the layers. In Multi-Comp, for example, this changes the order of the image layers in the composite, while in the Roto node it changes how overlapping shapes are stacked.

To reorder a layer, click the entry once to select it (a selected layer is shaded darker than a non-selected layer), and then drag it up or down. Release the mouse button when the white indicator line is in the desired position in the layer list and the layer will be inserted there.

Hiding/Disabling Layers and Overlays

Layers are on by default, but you can disable a layer temporarily (hide it) by clicking the checkbox, which is called out with the label “Disable” in the illustration above (*Fig. 7.7*).

Layer overlays may or may not be on by default, depending on the node. They are off by default in the Multi-comp node, for example, and on in the Roto node. Click the Hide/Show Overlay button (see *Fig. 7.7*) to toggle the overlay for that layer to the opposite state.

Editing the Name Field

Double-click on the layer label to change it into an editable text field in which you can type a new name for the layer.

Copying, Pasting, and Deleting Layers

You can copy, paste, and delete *all selected* layers using the Layer Actions menu (see *Fig. 7.7*):

1. Click once on the layer label to select the layer (the label will invert to indicate that it is selected).
2. Then right-hold on the selected layer to access the Layer Actions menu and select the appropriate command.

SELECTING MULTIPLE LAYERS

Hold down the Shift key as you click to select multiple layers in the Node Panel. Remember that all selected layers are affected by the command you select in the Layer Actions menu.

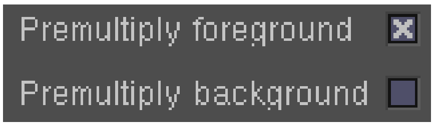
DELETING INPUT LAYERS

When a parameter group represents an input to the node—Multi-comp (see *Fig. 7.6*) and Sequence are two examples—there are two ways to delete it, each of which produces a different result:

- Select the layer and then select Delete from the Layer Actions menu, as described above.
- Ctrl-click the input connection line to cut the connection while leaving the input connector in place. This second method empties the parameter group without deleting it from the Node Panel so that you can replace one input with another, with the new node image inserted at the same layer level.

CHECKBOXES AND TOGGLE BUTTONS

If a box is checked with a checkmark, it indicates that the labeled option is active, or "on." Checkboxes work like toggle switches: click a box that is checked to turn it off; click a box that is not checked to turn it on.



7.8 Over Node Parameters: the foreground will be premultiplied; the background will not.

Some parameters use toggle switches in the form of a button that you depress. They work just like checkboxes, but the button itself is labeled with the item it controls, either as text or an icon. The Overlay and Lock toggle buttons in the Track node (see *Fig. 7.7*) are typical examples that use icons. An example of toggle buttons labeled with text are the Channel Select parameter buttons, which are described next.

CHANNEL SELECT

Most nodes provide a Channel Select parameter to control which channels of the input image are affected by the node operation. The Channel Select parameter consists of toggle buttons for the Red, Green, Blue, Alpha, and Other channels.

7.9 Channel Select Parameters: In this example, the input image is RGBA, but the Alpha channel will not be processed by the node.

All buttons corresponding to channels that exist in the input image will be active. Each channel button is used separately to control processing of the corresponding channel.

The channel buttons that appear to be depressed, or "pushed in," are on, and the image channel that corresponds to that button will be processed by the node. Any image channel that is not selected will not be processed—the input data will be sent out of the node unchanged.

DATA ENTRY PARAMETERS

A data entry field is a parameter control that accepts the keystrokes you type. Most fields are for entering numeric values only and are usually associated with a slider bar control and an animation menu. A typical example would be the Brightness parameter, which represents the brightness value to use.

Some fields do accept text entries, however, such as the File field in the Image In Node Panel, in which you can enter the directory path and file name of image files.

> **TIP:**
> When editing multiple data entry fields in a Node Panel, press the Tab key to jump to the next field. The Shift-Tab key combination jumps the cursor to the previous field.

FIELDS AND SLIDERS

Numeric fields are adjusted by typing directly into the field, or by using the associated slider bar to increase or decrease the field value.

The field shows the current value specified for the parameter, while the slider bar displays the value range (or suggested range, for parameters that are not constrained).

7.10 Data Entry Parameter: The combination of data entry field, slider, and animation menu (on the right end) is a typical configuration for a data entry parameter.

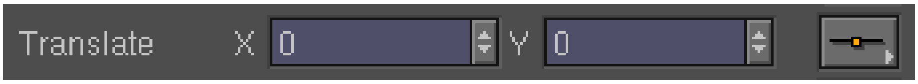
To use the slider associated with a field, click anywhere in the slider bar and drag to the left to decrease the value and to the right to increase it. The number in the field will update as you drag.

> **TIP:**
> To fine-tune values using the slider, hold down the Shift key as you drag back and forth in the slider. This will constrain the increment to a much finer scale.

FIELD PAIRS

Data entry fields often come in pairs and are used to specify *x,y* coordinate values. Field pairs provide a scrolling arrow button for each field, which is more compact than the slider bar.

To use the scroll arrow, click and hold on the button and then drag up to increase the field value or down to decrease it. You can drag the cursor beyond the boundary of the button; you only have to start the drag motion on the button.

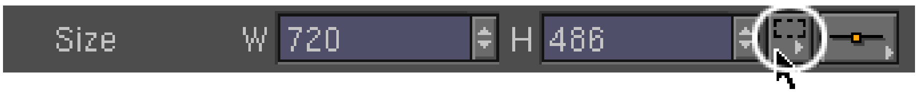


7.11 Data entry parameter with a pair of fields that use scroll arrows (vertical sliders).

For finer control, you can also drag right or left to change the value more gradually. (This is the equivalent of holding down the Shift key as you drag in the slider bar control.)

SIZE MENU

Field pairs for specifying the total width and height of the image are usually accompanied by a Size menu, which lists many film and video resolutions that can be selected as an alternative to actually typing width and height values into the fields.



7.12 Hold down the Size menu button (circled) to access a list of common resolutions.

The Size menu includes the following preset resolutions:

4K 4:3 4096x3072
2K 4:3 2048x1536
1K 4;3 1024x768
512 4:3 512x318

HDTV 16:9 1920x1080
NTSC (CCIR 601) 720x486
PAL (CCIR 709) 720x576
NTSC 640x486
PAL 704x576

IMAX (Full) 4096x3172
Super35 (Half) 2048x871
8-perf 70mm (Full) 4096x2028
VistaVision (Full) 4096x6114
CinemaScope (Full) 3656x3112
CinemaScope (Half) 1828x1556
Full Aperture (Full) 4096x3112
Full Aperture (Half) 2048x1556
Academy (Full) 3656x2664
Academy (Half) 1828x1332
Academy (Quarter) 914x666

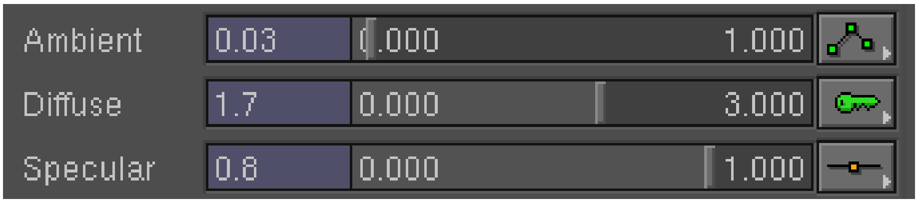
ANIMATION MENU

Any parameter that has an Animation menu can be animated. You can use the commands in the Animation menu to create keyframes and specify the type of interpolation to use between them. Just as important, the Animation menu also makes it easy to delete a keyframe, or all keyframes.

"Animating Parameter Values" on p. 99 explains how to animate Node Panel parameters using the Animation menu commands and Autokey mode. The following paragraphs cover the Animation menu status indicators as well as the Float Display item in the menu.

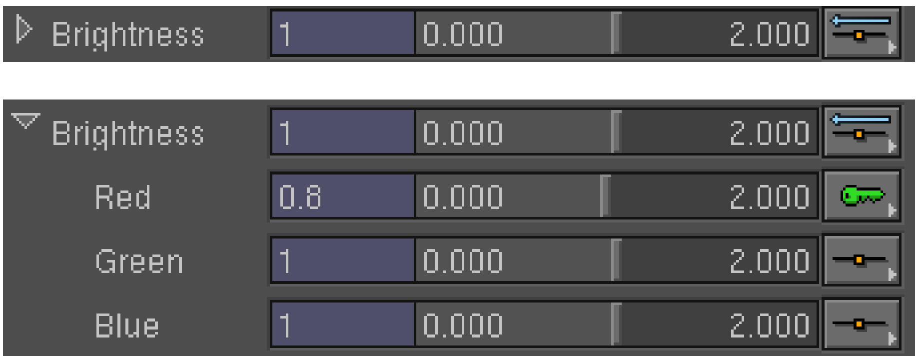
As shown in *Fig. 7.13* and *Fig. 7.14*, the Animation menu indicates the current status of the parameter—whether it is animated, and if so, whether it is currently at a keyframe.

7.13 Ambient parameter is animated; Diffuse is animated and currently at a keyframe; Specular is not animated.



In the case of parameter groups, it is possible to animate one or more parameters within the group without animating the master control. If so, the Animation menu of the master control parameter will indicate this by displaying a blue bar across the top. When the group is collapsed, this serves as a visual cue that a parameter within the group is animated—just expand the group to find out more.

7.14 Brightness parameter (top) is not animated, but the blue bar in the Animation menu indicates that a parameter within the group is. Expanding the group (bottom) reveals that the Red channel is animated.



FLOAT DISPLAY UNITS

The parameter Animation menu also includes the Float Display option. Depending on the type of operation, parameter values may be expressed in different units of measure. Certain color correction parameters use color depth units (a range of 0 to 65535, for example), and parameters that specify size or position use pixels.

Regardless of the default unit display, however, you also have the option of working in floating point (i.e., fractional) units instead by checking the Float Display box in the Animation menu list.

For example, scale parameter values are expressed by default as the number of pixels in X and Y, such as 2048 x 1556. If you prefer, however, you can switch to Float Display and enter a scale factor (such as 0.5, to specify 50 percent of full size) into the fields to scale an image rather than calculating the pixel values.

NOTE:

However you choose to display parameter values, RAYZ always calculates and stores them at floating point precision to facilitate the use of proxies and clones. This means that you can mix and match the display of value units to suit your needs.

ANIMATING PARAMETER VALUES

Virtually all parameters in RAYZ can be animated; that is, their values can vary over time. The default state for most parameters is not animated, which means that the value you set for the parameter applies to all frames of the input sequence.

There are several ways you can animate parameter values in RAYZ, depending on your preference and the specific task at hand:

- You can turn on Autokey mode.
- You can use the Animation menu associated with the parameter.
- You can use the Curve Editor.

This section describes how to use Autokey mode and the relevant commands in the parameter Animation menu. For more information about editing parameter values as curves in a graph, see "Animating Curves" in chapter 8, p. 108.

USING AUTOKEY MODE

The easiest way to animate node parameters may be to turn on Autokey mode, which affects all animatable parameters in a node.

7.15 The Autokey button in the strip at the top of the Node Panel is used to turn the auto-keyframing mode on and off. When the button is depressed it turns green to indicate that Autokey mode is active. The arrow buttons on each side are used to navigate directly to the next or previous keyframe.

When Autokey is on, it changes the way RAYZ interprets any modifications you make to parameter values:

Autokey On

With Autokey on, whenever you go to a new frame and change a parameter value, a keyframe is created automatically and the values of the in-between frames are calculated using linear interpolation.

To create a keyframe without modifying the parameter value, select Add Key from the Animation menu, which is described below in "Animation Menu Options" (p. 100).

Autokey Off

When auto-keyframing mode is off, any change you make to a parameter value at any frame is applied to every frame.

Turning off Autokey mode does not change a parameter that has already been animated. If a parameter is animated and you turn Autokey off, the parameter field will only be active at keyframes (although you can still use the parameter Animation menu to add keyframes).

To delete a keyframe, or to change the default interpolation used between keyframes, use the Animation menu of a parameter, as described below in "Animation Menu Options" (p. 100).

NOTE:

The Image Viewer provides a corresponding Autokey button which you can use instead when you are working directly on the image using a node overlay (such as the Transform widget). Whenever an Image Viewer and Node Panel are displaying the same node, the two Autokey controls are tied; a change to one button is automatically reflected in the other.

Key-to-Key Navigation

The arrow buttons on each side of the Autokey button take you to the previous or next keyframe in the sequence. They are a quick way to navigate directly from keyframe to keyframe without entering frame numbers into the Time Scooter.

The arrow buttons always work as long as any of the node parameters are animated; Autokey mode does not have to be on to use them.

7.16 In the Time Scooter, green tick-marks appear at every keyframe.

Animation Menu Options

You can always animate any individual parameter in a node without turning on Autokey mode by using the appropriate option in the parameter Animation menu.

You can add or delete a keyframe at the current frame, delete all keys in the parameter, or reset a parameter to its project default value (which may or may not be animated). The current parameter status is indicated by the Animation menu icon, as shown in *Fig. 7.13* (p. 98).

Add Key

To add a keyframe, use the Time Scooter to navigate to the frame and select Add Key from the Animation menu.

Delete Key

To delete a keyframe, navigate to an existing keyframe and select Delete Key from the menu.

Delete All Keys

To "de-animate" a parameter, select Delete All Keys. The new parameter value, which applies to all frames, will depend on where you are when you select the command:

- If you are at a keyframe, that keyframe value will be used.
- If you are at an in-between frame, the value of the preceding keyframe will be used.

Reset to Project Defaults

To return the parameter to its default value (the value it had when you first created the node), select Reset to Project Defaults from the Animation menu. The project default value may or may not be animated, depending on the parameter.

Change Interpolation

You can also change the interpolation function used between keyframes. Navigate to an in-between frame and select an option (Linear, Ease, Bezier, etc.) from the Change Interpolation submenu. The new interpolation curve will be used for all the in-between frames in the current segment. You can apply a different type of interpolation to each segment.

The Change Interpolation submenu is equivalent to the Curve Actions menu in the Curve Editor. For information about each interpolation function, see "Controlling Interpolation of Curve Values" in chapter 8, p. 109.

Edit Expression...

To use an expression to control a parameter, select Edit Expression from the Animation menu (or use the **hotkey: Shift-e**). This opens a panel in which you can create and edit an expression to control the parameter value. The Expression Editor is described in "How to Enter an Expression" in appendix C, p. 446.

If the parameter is already animated, choose “Change interpolation to expression” from the Change Interpolation submenu of the Animation menu. Then you can use the Edit Expression command to apply an expression to the current segment.

NOTE:

The Edit Expression command is not available for a parameter with multiple values, which in most cases means a parameter that controls both *x* and *y* coordinate values.

However, you can still use the Curve Editor to enter an expression to control each value separately as described in “Entering Expressions in the Curve Editor” in appendix C, p. 447.

USING MASK INPUTS

Many nodes provide a mask input connector (it is labeled with an “M”). This optional input is used to connect a mask image, which controls which pixels in the primary input are modified by the node.

HOW MASKS CONTROL NODE PROCESSING

Mask inputs can be used in numerous ways to control image processing in a node. To simulate atmospheric haze in a CG image, for example, you could use a channel of z-depth data as the mask input to a color correction node to desaturate the image slightly and reduce its luminance as the distance from the camera increases. In a Blur node, the z-depth data could be used to selectively blur pixels based on depth.

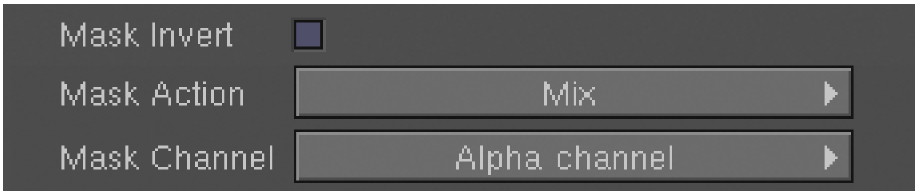
Another example would be to use the output of an edge detection node as a mask input to filter only the edges of an image. Or a matte of a specific object in an image could be used to limit the effect of a filter to that object, or to everything but that object.

NOTE:

In addition to or instead of mask inputs, some nodes feature other types of optional inputs that affect node processing in various ways. Refer to the individual node descriptions for more information about using such inputs.

MASK INPUT PARAMETERS

Whenever you connect a mask input to a node, corresponding mask input parameters in the Node Panel are activated. These parameters enable you to specify which channel of the mask input to use and how to interpret the mask channel data.



7.17 The Mask parameters appear in the Node Panel for every node type that accepts a mask input.

MASK INVERT CHECKBOX

Check this box to invert the pixel values of the mask when you want the opposite areas of the primary input image to be filtered by the node.

MASK ACTION MENU

The Mask Action menu lets you specify how the information in the selected mask channel will be used by choosing On/Off or Mix.

In either case, the value of each pixel in the mask channel governs how the corresponding pixel in the primary input image will be processed. (The "corresponding pixel" refers to the pixel with the same *x,y* coordinates.)

The difference is that the On/Off method, as the name implies, simply turns each pixel fully on or fully off, while the Mix method controls the extent to which each pixel is affected by the node operation.

ON/OFF

The On/Off method specifies whether a pixel will be modified by the node:

- For any mask channel pixel with a value of 0, the corresponding pixel of the primary image will not be filtered; that is, the original value of the pixel will be output unchanged.
- For any mask channel pixel with a nonzero value, the corresponding pixel in the primary image will be filtered by the node, just as it would if the mask image were not connected.

MIX

The Mix method, which is the default, uses the value of each mask channel pixel to determine the extent to which the corresponding pixel in the primary input will be affected by the node operation.

Each output pixel will be a mixture of the filtered and unfiltered value of the primary input pixel, with the ratio being determined by the value of the mask channel pixel. (RAYZ uses the floating point value of the pixel; that is, the pixel value expressed as a percentage of the maximum value for the color depth.)

For example, a mask channel pixel with a float (fractional) value of 0.32 will turn on the node operation 32 percent, so that the corresponding output pixel will be a mixture of 32 percent of the filtered input pixel value and 68 percent of the original input pixel value.

Mask Channel Menu

The Mask Channel menu enables you to specify which channel in the mask image will be used to govern the mask action, when the mask input has more than one channel. This menu defaults to the alpha channel, if there is one, however you can choose any channel.

chapter 8

USING THE CURVE EDITOR

The Curve Editor is used to animate parameter values across time. Parameter data is displayed as a waveform, or "curve," in the Curve Editor graph, which plots the parameter's value across a range of frames.

You can control which curves are displayed in the graph, add and delete keyframes and adjust their value, and apply different distribution functions to curves to control the interpolation of values between keyframes.

IN THIS CHAPTER

Displaying Parameters from Multiple Nodes

Unlike the Node Panel, the Curve Editor view is able to display parameter data from more than one node. This enables you to compare and copy curves from one node to another.

8.1 All nodes selected in the Worksheet (top) are added to the Curve Browser list (lower left).

By default, the Curve Editor displays parameter data for the node or nodes currently selected in the Worksheet.

To remove a node from the Curve Browser list, press the Delete button ("X" icon) at the right side of the node entry.

You can also pin a node to a Curve Editor, if you wish, so that the Curve Editor will no longer follow the current selection in the Worksheet. Pinning any node to the Curve Editor will actually pin all nodes displayed in the Curve Editor at that time. For more information, see "Dynamic Focus" in chapter 4 (p. 27).

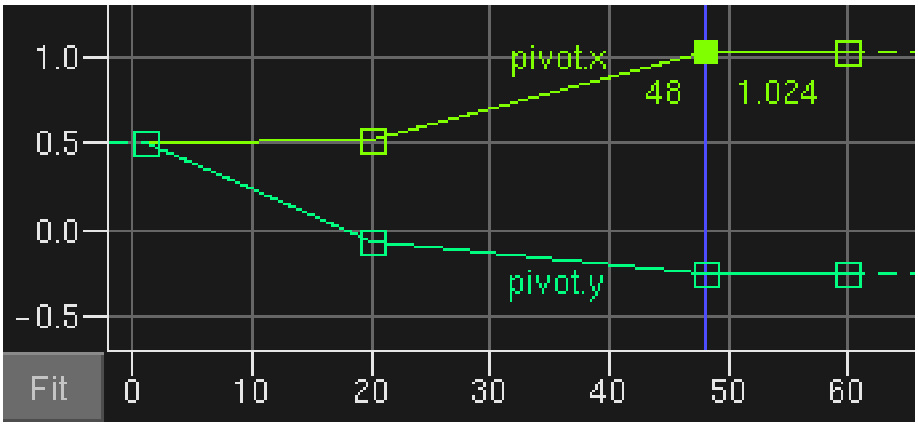
TIP:

Press the **3** key while the cursor is hovering over a node to launch a new Curve Editor window with that node data displayed in it.

The Graph

The Curve Editor graph displays a curve for all node parameters specified in the "Curve Browser" (p. 107).

8.2 Curve Editor Graph: horizontal axis represents time; vertical axis, value.



In the graph, the **horizontal axis** represents time, expressed in equal increments of frame numbers, and the **vertical axis** represents the range of parameter values being plotted. The current frame specified in the Time Scooter is indicated by a vertical blue line.

Adjusting the Graph Display

You can reposition the graph in the view frame and scale the graph up or down to get the best view of your curve data. In most cases the Fit button is the best option, however, you can also manipulate the graph directly.

Fit Button

The Fit button is located in the bottom right corner of the Curve Editor graph. When you click it, the graph is scaled to fit all of the curves into the currently visible graph area.

Moving the Graph in the View Frame

You can move the graph around in the view frame by middle-dragging in the background. Alternatively, the horizontal time scale at the bottom of the graph and the vertical value scale at the left side of the graph can be used as scrollers to constrain movement to one axis. Drag the time scale to move back and forth in time, or drag the value scale to move up and down in the value range.

Scaling the Graph

You can scale in both axes, or constrain scaling to either axis:

- To scale the graph display up or down in both axes simultaneously, hold down the Control key while middle-dragging in the graph.
- To scale the time axis only, Ctrl-drag the time scale at the bottom of the graph.
- To scale the value axis only, Ctrl-drag the value scale at the left side of the graph.

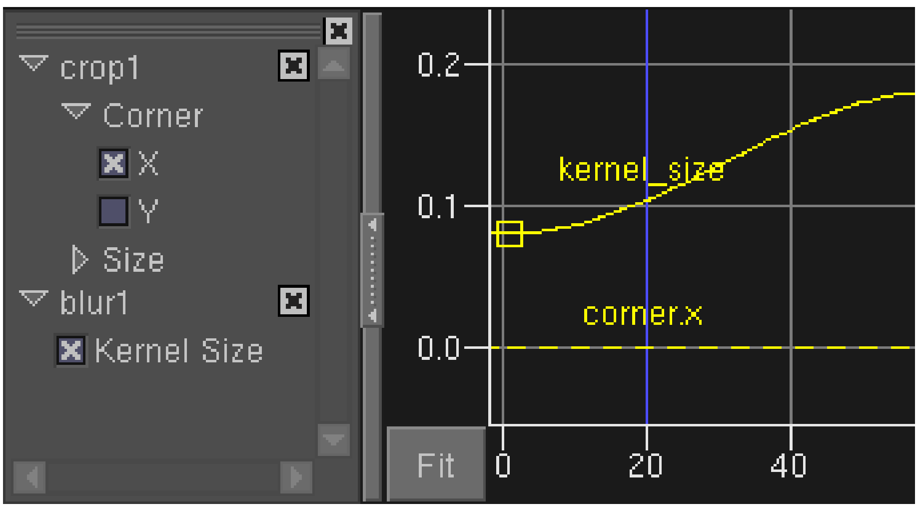
Curve Editor Tools

The Tools menu in the title bar of the Curve Editor is used to control the display of the Curve Browser panel, Keypoint Viewer panel, and Curve Colors palette.

Curve Browser

The Curve Browser It lists all animatable parameters for a node. The checkbox next to each parameter is used to control display of the corresponding curve in the graph.

As in the Node Panel, some parameters in the Curve Browser list are grouped under a heading in outline format, which can be expanded to access parameters in the group or collapsed to make more room in the list.



8.3 A corresponding curve is displayed in the graph (right) for each parameter you check in the Curve Browser list (left).

Keypoint Viewer

The Keypoint Viewer is used to edit keypoints (the keyframe markers on a curve) numerically rather than by dragging them around in the graph. See “Editing Keypoints Numerically” on p. 111, which also explains how to enter and edit expressions in this panel.

Stowing the Panels

The Curve Browser and Keypoint Viewer are displayed in the Curve Editor by default. However you can use the Shutter button (located on the inner edge of each panel) to stow either of them out of the way temporarily if you need more room for the graph display.

Curve Colors

The Curve Colors palette is used to redefine the colors used to display curves in the graph. A color is assigned to a curve in the order in which the corresponding parameter appears in the Curve Browser list. The first swatch color in the palette is assigned to the first parameter in the list, the second color to the second parameter, and so on.

To change a color, check the Curve Colors item in the Tools menu to access the palette (it is not displayed by default). Right-hold on a swatch in the palette and a color spectrum bar will pop up. Drag the cursor across the spectrum to the color you want and release the mouse button. Any curve that uses that swatch will update to match the new color.

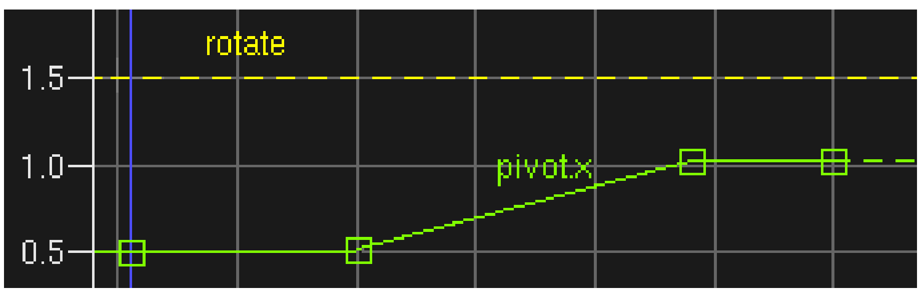
Animating Curves

A curve can be animated initially in the Node Panel and refined in the Curve Editor, or you can start in the Curve Editor and create keyframes to animate the curve.

A parameter that has not been animated is displayed in the graph as a dashed line. Its value remains constant across the entire frame range.

An animated curve is displayed as a solid line, with each keyframe represented by a point on the curve called a keypoint.

8.4 The rotate curve is not animated; the pivot.x curve is animated with four keypoints to create three separate curve segments.



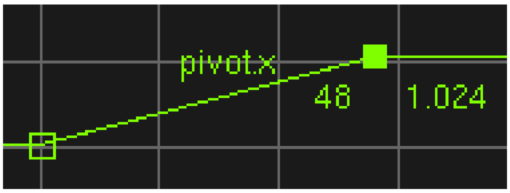
Every animated curve has a minimum of two keypoints, one at each end. You can add as many keypoints to a curve as necessary, and each section of curve between keypoints is referred to as a curve segment.

You manipulate curves in the graph by adding and deleting keypoints and changing their position and value, as well as by applying interpolation functions to curve segments between keypoints.

CREATING AND EDITING KEYPOINTS

To create a keypoint, hold down the Control key while clicking on the curve. You don't have to be precise about where you click; you can always adjust the position (frame number and value) of the point once it has been created.

To select a keypoint, click on it. To select multiple keypoints, drag a bounding box around them, or hold down the Shift key as you click on each of them in turn.



8.5 Selected keypoint (solid square) is at frame 48 and has a value 1.024. The unselected keypoint is a hollow square.

To change the position of a keypoint, select and drag it. Drag up or down to adjust the value; drag left or right to move the keypoint to a new frame. All selected points move when you drag. To shift an entire curve without changing its shape, select all keypoints in the curve and then drag any one of the points.

> **TIP:**
> To select all keypoints in a curve, click anywhere on the curve except on a keypoint. (Be sure to drag the entire curve by one of the selected points; you cannot drag by clicking on a line segment.)

DELETING KEYPOINTS

To delete a keypoint, select it and press the Delete key. For a curve with three or more keypoints, deleting the first or last keypoint shortens the curve. For a curve with two keypoints, deleting a keypoint returns the parameter to a non-animated state.

CONTROLLING INTERPOLATION OF CURVE VALUES

You specify the value of each keypoint on a curve when you create and edit it. Then RAYZ interpolates the values for the frames in between, based on the keypoint values.

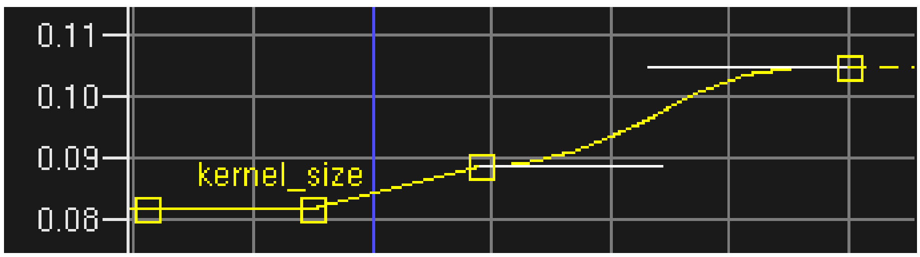
Linear interpolation is applied by default, but you can assign other interpolation functions to a curve segment. In this way you can control the rate and degree of change in the parameter value across time:

1. Right-hold on the curve segment you want to change and the Curve Actions menu will pop up.
2. Select a function from the menu list and when you release the mouse button, the function will be applied to that curve segment.

CURVE ACTIONS MENU

The following interpolation functions can be selected from the Curve Actions menu and applied to the curve segment under the cursor:

- **Bezier**: Interpolates in and out points by fitting a Bezier curve using slope and acceleration.
- **Constant**: Constant value of the in point.
- **Cubic**: Interpolates in and out points using slopes. Results in a cubic polynomial.
- **Ease**: Ease in smoothly at the start and ease out smoothly at the end.
- **Easein**: Ease in smoothly at the start.
- **Easeout**: Ease out smoothly at the end.
- **Expression**: Select this option to use an expression to control the curve.
- **Linear**: Straight line interpolation from start to end.
- **Quintic**: Interpolates the in and out points using slope and acceleration. Results in a quintic (degree 5) polynomial.



8.6 The first two segments of the curve use linear interpolation; the last segment uses a Bezier function (note the handles).

BEZIER HANDLES

When a Bezier function is applied, the appropriate keypoint in the curve will display handles you can use to adjust the slope and acceleration of the curve:

- Drag the handle radially around the keypoint to adjust the angle of the slope.
- Drag the handle in or out (the handle will shrink or stretch) to change the rate of acceleration.

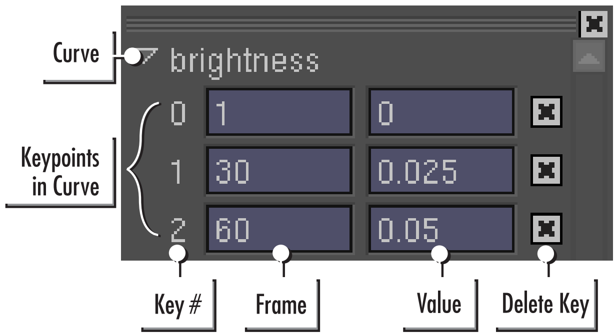
OPENING THE EXPRESSION EDITOR

Once Expression is specified for the interpolation, the Edit Expression command becomes available from the Curve Actions menu. Select it to bring up the Expression Editor panel, which is described in "Using the Expression Editor" in appendix C, p. 446.

MERGING FUNCTION SEGMENTS

When you delete a keypoint, the curve segments on either side are merged. If a different interpolation function had been applied to each segment, the merged segment will use the function from the segment that had been on the left (the preceding segment in time).

EDITING KEYPOINTS NUMERICALLY



8.7 Keypoint Viewer.

Use the Keypoint Editor to examine and edit the values of keypoints numerically. The Keypoint Viewer panel should displayed by default; if not, you can select it in the Curve Editor Tools menu.

All keypoints for the currently visible curves appear in the Keypoint Editor. Two data entry fields are provided for each keypoint, in which you can type a new value:

- The left field controls the frame number (horizontal axis).
- The right field controls the parameter value (vertical axis).

USING THE EXPRESSION FIELD

If the "expression" interpolation type has been applied to the curve, an additional field will be displayed in the Keypoint Viewer in which you can enter or edit an expression to control the value.

8.8 Expression field in Keypoint Viewer.

For more information, see also *Appendix C: Using Expressions in RAYZ* (p. 445). Step-by-step procedures are given in the section on "Entering Expressions in the Curve Editor" (p. 447).

COPYING CURVES

You can copy any curve to another curve using the Copy and Paste commands in the Curve Actions menu:

1. Right-hold on the curve you want to copy to access the Curve Actions menu and select Copy from the list.

2. Right-hold on the curve to which you want to paste the copied curve and select Paste from the Curve Actions menu.

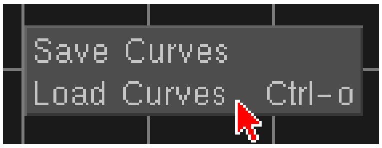
When you release the mouse button, the values of the first curve are applied to the second.

NOTE:

You can only copy entire curves, not individual keypoints.

EXPORTING AND IMPORTING CURVES

You can export a curve (save it in a file on disk) and then import it into any RAYZ project file.



8.9 Right-hold anywhere in the graph, except on a curve, to access the popup menu used to save and load curve files.

Curves are exported and imported using the Save Curves and Load Curves commands in the Graph Actions popup menu, which is accessed by right-holding in the graph. (Do not click on a curve, or the Curve Actions menu will pop up instead.)

SAVE CURVES

The Save Curves command saves all curves currently displayed in the Curve Editor graph into a RAYZ curve file. When you choose Save Curves from the Graph Actions menu, a dialog box will appear in which you can name the curve file and specify where to save it.

If you select an existing curve file, the new curve data will be saved to it, replacing the old curve data in the file.

RAYZ CURVE FILES

A RAYZ curve file is simply a text file containing a description of the curve attributes. A curve file is given the ".rzc" extension by default when you create it, and the dialog box used to load curves will display all files in a directory with this extension.

LOAD CURVES

You can load not only RAYZ curve files, but Boujou 2D track data and Maya parameter curve data.

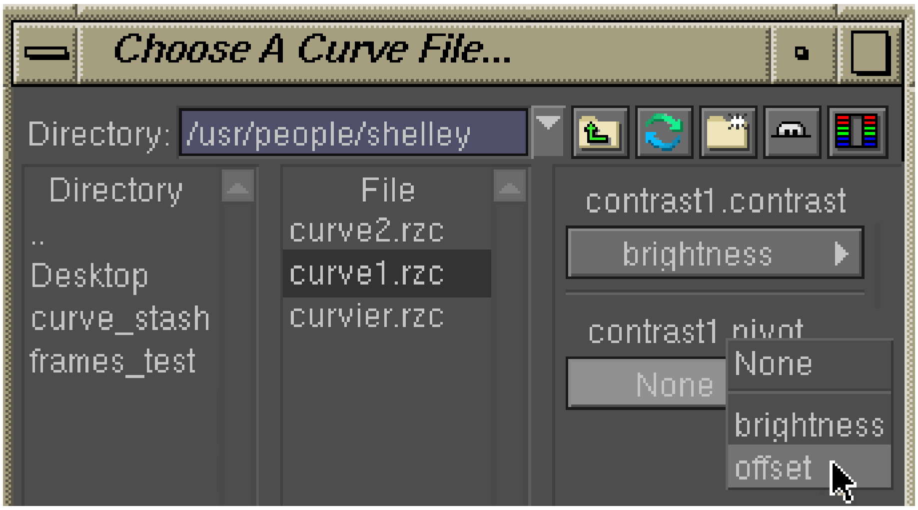
To import a curve, choose the Load Curve command (Ctrl-o) from the Graph Actions menu to open the Load Curve dialog box. Select a curve file from the File list in the middle pane. (Navigate the directory structure if necessary to locate the file.)

When you select a curve in the dialog, a menu will appear (on the right) for each curve available in the curve editor graph. Use the menus to specify whether or not the saved curve will be loaded into each curve in the graph.

Curve Selection Menus

When a curve file contains data for more than one curve, you can use the Curve Selection menus in the Load Curves dialog box to specify which curve in the file is imported to each curve in the graph. You can

- import any curve that was saved in the selected file.
- import the same saved curve into all curves in the graph.
- import a different saved curve into each curve in the graph.
- prevent any curve data from being imported to a curve in the graph by selecting "None" from the menu.



8.10 The "curve1.rzc" curve file is selected (middle pane). The Curve Selection menus (right pane) show that a saved curve named "brightness" will be imported into one graph curve, while an "offset" curve is being selected for import to the other.

chapter 9

MAIN MENUS

The Main Menus provide commands and options for the RAYZ interface and RAYZ project files. The Main Menu strip is located by default in the upper left corner of the RAYZ interface.

IN THIS CHAPTER

File Menu

The File menu provides commands to create, save, open, and close RAYZ project files, as well as to import and retime image sequences.

New (Ctrl-n)

This command creates a new, empty RAYZ project file called "untitled." You will be prompted to save the file currently open when you invoke the command, if the current file contains unsaved changes.

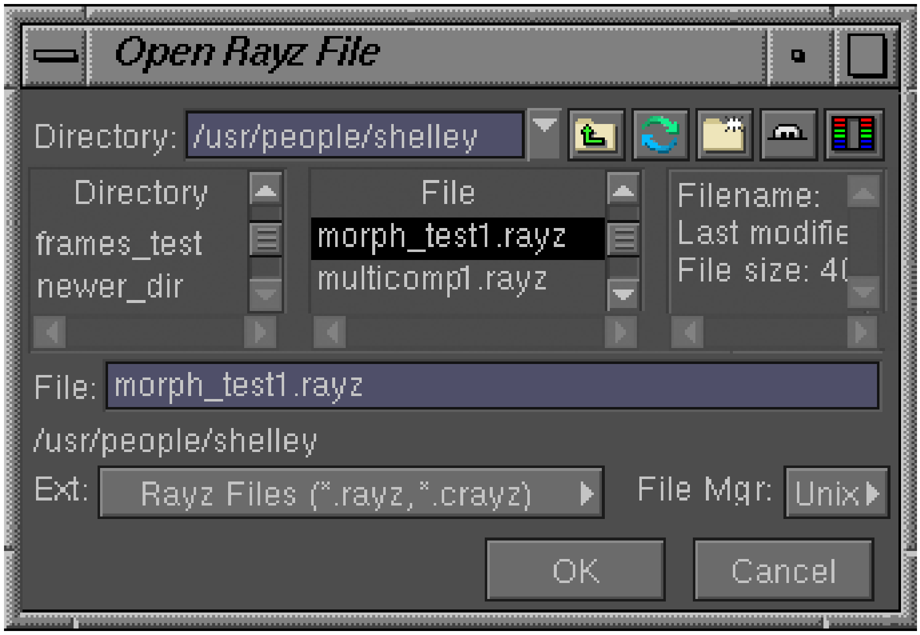
Open... (Ctrl-o)

This command brings up the RAYZ File dialog box, from which you can navigate the directory structure of your network to find the RAYZ project file to open.

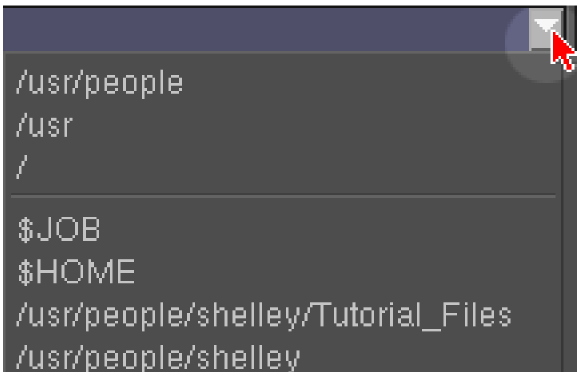
Open File Dialog Box

Use the Open File dialog box to navigate to the directory containing the file you want to open. Then double-click on a RAYZ file in the File list to open it. You can open both regular (.rayz) and compressed (.crayz) files in this manner. (See also "Saving Compressed RAYZ Files" on p. 118.)

9.1 Dialog box used to open RAYZ project files.



Directory Field The Directory field will default to the most recently used directory, however, you can select another recent directory from the pull-down menu associated with the Directory field, type a directory path into the field, or use the file navigation buttons to go to a directory.



9.2 Example of the contents of the Most Recently Used (MRU) directory menu.

Directory MRU The menu button on the right end of the Directory field can be used to access a list of most recently used files.

This menu also includes $JOB, which takes you to the current job directory, and $HOME, which takes you to your home directory. The JOB global variable can be redefined in the Project Settings panel, as described in "Defining $JOB" (ch. 13, p. 157).

Extension Menu The Extension menu defaults to displaying only RAYZ files (files with the ".rayz" or ".crayz" file extension). If you select All Files from the Extension menu, the File field will list all files in the directory.

FILE MANAGER MENU The default is to use the file manager for your operating system, however, you can select the FTP file manager instead.

IMPORT FILE...

The Import File command is used to import a multi-layer Photoshop file into a new project file in one step, without flattening the layers. After prompting you to save the current RAYZ file, if necessary, this command opens a dialog which you use to locate the Photoshop file.

Each layer of the Photoshop file you choose is automatically imported into a separate Image In node, and the Image In nodes are all connected to a Multi-comp node in the same layer order as in the Photoshop file, and with the same blending modes applied if possible.

IMPORT FOOTAGE... (CTRL-I)

The Import Footage command is equivalent to pressing the Import button in the node menu strip in the Worksheet. It opens the Import Footage dialog box, which you use to navigate to the directory containing the image files you want to import.

When you select an image sequence and press the Import button in the dialog box, RAYZ creates an Image In node with that sequence specified in it and places it in the Worksheet. You can import as many different image sequences as you want before closing the dialog. See also *Chapter 10: Importing Images* (p. 123), which covers the topic in general and describes the Import Footage dialog box components in detail.

RETIME FOOTAGE...

The Retime Footage command is used to access the RAYZ utility for retiming imported image sequences, which uses Cineon's Cinespeed method or frame averaging, as you choose. It opens the Retime Footage panel, in which you specify the sequence to retime and the retiming parameters to use. See "Using the Retime Footage Utility" in chapter 12 (p. 140) for more information.

SAVE (CTRL-S)

This command saves the changes you have made to the current RAYZ project file since the last save. If the file is a new, untitled file, the Save command works like the Save As command, described below.

SAVE AS...

This command is used to save a new, untitled RAYZ project file or to save an existing file under a new name.

SAVE FILE DIALOG BOX

The Save As command brings up the RAYZ Save File dialog box, in which you can specify the name and directory in which the file should be saved.

This dialog uses the same structure as the Open File Dialog Box shown in *Fig. 9.1* (p. 116).

Navigate to the directory in which you want to save the file. Then type a name into the File field and press the Enter key or click the OK button in the dialog box. The ".rayz" file extension will be appended to the filename automatically; there is no need to actually type it.

Saving Compressed RAYZ Files

You can compress RAYZ files using Bzip2 compression. For project files with extensive roto or other operations that generate a lot of file data, this can really reduce file size.

To compress a RAYZ file, use the Save As command to save the file with the .crayz file extension instead of .rayz. Type `.crayz` instead of `.rayz` and press the OK button.

To expand a compressed file, simply reverse the process by saving the .crayz file as a .rayz file. (It is not necessary to do so, however, to reopen and work in a file.)

Revert...

Use the Revert command to revert to the last version you saved of the currently open file. When you select the command, a prompt will ask you to confirm the action.

MRU File List

The File menu always lists the four Most Recently Used (MRU) project files for your convenience when reopening a current project.

The MRU list will appear in the menu between the Save As and Close Window commands.

Close Window (Ctrl-w)

This command is used to close the current window, when you have more than one RAYZ window open. If you select this command when only one RAYZ window is open, it works like the Quit command, described next.

Quit (Ctrl-q)

This command will quit the RAYZ application after prompting you to save any changes to the current file if necessary.

Edit Menu

The Edit menu includes the **Undo** (Ctrl-z) and **Redo** (Ctrl-r) commands, as well as providing access to the General Preferences and Project Settings panels, which are described in *Chapter 13: Setting Preferences* (p. 147).

TIP:

The Ctrl-z and Ctrl-r hotkeys are very convenient shortcuts for Undo and Redo. You may want to access these commands from the Edit menu instead, however, when you need feedback about exactly what action will be undone or redone. The Edit menu display is dynamic; for example, it might read “Undo parameter change” or “Redo node delete” rather than just “Undo” or “Redo.”

Levels of Undo and Redo

You can use the Undo command multiple times to step back through your actions in the order they were performed. There is no limit on the number; you can continue to undo your actions up to the last time you saved the file, unless memory constraints prohibit it. The same applies to redoing your actions.

What Gets Undone and Redone

You can undo (or redo) any change you make to a node parameter, whether in the node panel or in another view such as the Curve Editor. You can also undo and redo any change to the node network, including node creation, deletion, and connection.

The Undo and Redo commands are global, and cursor position is irrelevant when using the hotkeys for these commands. That is, if you change a parameter value in the Node Panel, add a node to the Worksheet, adjust a curve in the Curve Editor, and then change a node connection in the Worksheet, the Undo command will undo each action in order.

Changes to the Interface

You cannot use Undo and Redo for changes made to the interface. For example, if you split a view into two separate view panes, you cannot use Undo to change it back. The same is true if you change the display from RGB to alpha in an Image Viewer. In these cases you would simply use the standard interface controls to delete the extra view pane you created or to change the Viewer display from alpha back to RGB.

Views Menu

The Views menu is one way to add a view to the current layout. When you select one of the following options from the menu, a new view of that type is added to the bottom of the layout and the other view panes rearrange themselves accordingly:

Add Curve Editor
Add Image Viewer
Add Node Panel
Add Clip Editor

Add Worksheet
Add Render Control

For information about other ways to change the number and type of views in a layout, see "Changing the Layout" in chapter 4, p. 37.

Add New Window...

You can also create a new floating window by selecting Add New Window... from the Views menu. By default, the layout of the new window only has one view, the Image Viewer, which fills the window, although you can modify the layout just as you can in any RAYZ window.

See also "Creating a View in a Separate Window" in chapter 4, p. 36.

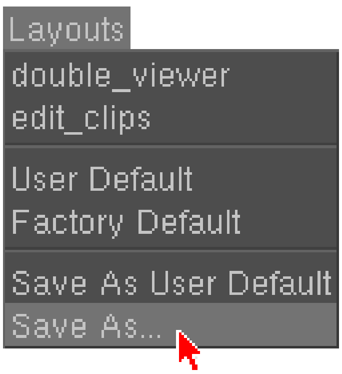
Tools Menu

The Tools menu is used to turn display of the Process strip, Status Bar, and Time Scooter on and off in the current layout.

These tools are described in the section on "Other Tools in the Layout" in chapter 4, p. 25.

Layouts Menu

The Layouts menu enables you to save and recall specific interface configurations. See also "Customizing Layouts" in chapter 4, p. 37.



9.3 Layouts Menu: The two items at the top are custom layouts created using the Save As command; the User Default item only appears in the menu after the Save As User Default option has been selected.

Factory Default (F10)

This command restores the default RAYZ layout configuration.

User Default

This item only appears in the menu if you elect to save a layout as the user default; select it to restore the saved user default layout.

Save As User Default

This command saves the current layout as the default layout that will be used for new files instead of the Factory Default. It does not replace the factory default, however; instead, a new item called User Default appears in the Layouts menu so that both layouts are available.

Save As...

This command is used to save the current layout to the Layouts menu as a custom layout.

9.4 Save As Custom Layout dialog box.

Just type the name you want to use for the layout in the dialog box that appears when you select this command.

The next time you pull down the Layouts menu, the new layout will appear at the top of the list.

Render Menu

The Render menu contains two commands: Render All and Open New Render Panel.

Render All

The Render All command will render image files to disk for all Image Out nodes in the file, using the current node settings for file format, location, etc. This command is equivalent to pressing the Render button in the Render Control view.

Open New Render Panel

This command opens a new window with a Render Control view in it. The Render Control view lists all Image Out nodes in the file, so you can use it to examine and modify the rendering parameters and specify which Image Out nodes to render before pressing the Render button.

See also *Chapter 11: Rendering Images* (p. 129) for detailed information about rendering images in RAYZ.

Help Menu

The Help menu is used to access RAYZ help options.

Help Tags

This command turns the Help Tag option on and off. Help Tags are contextual help text boxes that pop up when you mouse over elements of the interface. See also “Help Tags” in chapter 4, p. 39.

About RAYZ...

This command displays the version number of RAYZ in a floating panel. It is a good idea to jot down the version number before contacting Silicon Grail support with a question or problem.

RAYZ Manual... (F1)

This command opens the HTML documentation for RAYZ in the browser you have specified in the Help Browser Setting preference. See also “Settings” in chapter 13 (p. 149).

chapter 10

Importing Images

Image files are loaded into RAYZ by creating an Image In node for each sequence and using the file parameters in the Image In node panel to specify which sequence to load. However, a shortcut called "Import Footage" is available for importing multiple image sequences quickly.

You can import most popular image file formats and specify conversion options for many of them in the Image In Node Panel. For a complete list, see *Appendix B: Image File Formats Supported by RAYZ* (p. 441).

In addition, proxy resolution image sequences can be imported; each Image In node provides file parameters for full size images as well as medium and low resolution proxies.

You can even import a multilayer Photoshop file into a new RAYZ project file with each layer in a separate Image In node, as described in "Import File…" in chapter 9, p. 117.

IN THIS CHAPTER

Import Footage Shortcut

The basic way to import a sequence of image files is to create an Image In node in the Worksheet and then use the Node Panel to access a dialog box in which you can navigate to the directory containing the footage.

When you need to import dozens of different sequences, however, this method becomes tedious and inefficient. The fastest way to import a series of image sequences is to use the Import Footage command, which automatically creates an Image In node in the Worksheet for each sequence you select in the Import Footage dialog box.

The Import Footage dialog box can be accessed by

- selecting the Import Footage command from the File menu,
- pressing the Ctrl-i hotkey, or
- pressing the Import button in the Worksheet node menu strip.

Using the Import Footage Dialog Box

The Import Footage dialog box is similar to the standard file chooser that you access from an Image In node, in which you can navigate your directory structure to locate imagery to import.

The difference is that the Import Footage dialog includes the Import button, which creates an Image In node and imports the selected footage into it in one step.

In the Import Footage dialog box:

1. Navigate to a directory that contains a sequence you want to import. You can type the path into the Directory field or use the navigation buttons.

2. Click on any directory in the Directory list to display the files it contains in the File list.

3. Click on a sequence in the File list to select it and then press the Import button. (Alternatively, you can double-click the sequence, which is the equivalent of selecting it and pressing Import.)

As soon as you press the Import button, an Image In node is created and added to the Worksheet. The File parameter of the Image In node is automatically set to point to the sequence you specified, and the other node parameters are set to default values based on the format of the image files.

You can repeat the steps above to select as many additional image sequences as you need without closing and reopening the dialog box. Each time you select a new sequence and press the Import button, another Image In node is created for that sequence and added to the Worksheet.

Once you have finished importing imagery, press the Close button to close the dialog box. Then you can continue building the shot by adding other types of nodes, or you can change the default parameters in the Image In nodes for the imported imagery.

NOTE:

To import a Photoshop file with multiple layers, without having to flatten the layers into a single composite image or choose a single layer to use, see the description of "Import File…" in chapter 9, p. 117.

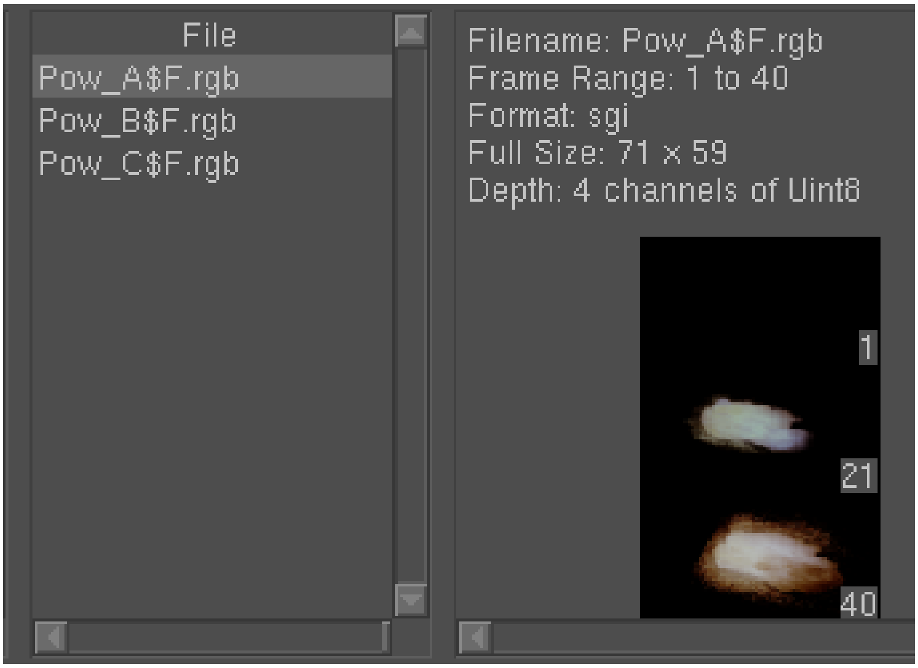
For a detailed description of the individual Image In node parameters, refer to section on the “Image In Node” in chapter 14 (p. 177).

TIP:

It may also save time to change some of the default Image In parameter settings (file conversion and pixel ratio options, e.g.) to suit the specific needs of a project. See “Editing Project Settings” in chapter 13, p. 156, for more information.

Getting File Info

The right side of the Import Footage dialog box displays information about the currently selected sequence in the File list, including the name, frame range, file format, size (spatial resolution), and bit depth (per channel). It also shows thumbnail images of the first, last, and middle frames in the sequence.



10.1 Click once on any image in the File list to see file statistics and thumbnail images.

Using the Sequence Filter

By default, the dialog box displays one entry per sequence in the File list. Take, for example, a three-second Cineon shot where each frame is a separate image file: “filename.0001.cin”; “filename.0002.cin” … and so on, through … “filename.0072.cin.” With sequence filtering on, the Files list will contain this single entry for the 72 image files: “filename.$F4.cin.”

10.2 The Sequence Filter toggle button is located in the top right corner of the dialog box.

Turn sequence filtering off (the Sequence Filter button is a toggle) to display all of the files individually. This enables you to import a single frame instead of the entire sequence, to view a thumbnail image of any frame before importing it, and to check the sequence for missing frames.

Using Proxies in RAYZ

All RAYZ Image In nodes provide the option to specify proxy files, which are lower resolution stand-ins for the final input frames. Proxies are often used when the full resolution inputs are very large files (2K or 4K film footage, for example). A proxy-resolution sequence can be much faster to view and modify as a shot is being built.

Selecting a Size in the Image Viewer

You can always view any image in RAYZ at a lower size than its native resolution by selecting Medium or Low from the Size menu in the Image Viewer. In such cases, RAYZ accesses the full resolution image and scales it down.

If a medium or low resolution image sequence has been specified in the proxy parameters of the Image In node, however, RAYZ will access these files when you choose Medium or Low in the Image Viewer's Size menu.

This can be considerably faster than scaling the full size imagery and also addresses the situation in which the full resolution files are not currently available, as when some of the background frames haven't yet been digitized or final versions of CG elements haven't finished rendering.

You can tell RAYZ what size the full resolution images will be and use the available proxies to build the shot. If the full size imagery is available, on the other hand, you can create proxies for it in RAYZ.

Creating Proxy Files

You can create proxy files in RAYZ for any image sequence you import into an Image In node. Connect the Image In node directly to an Image Out node, select Medium or Low from the Scale Factor menu, and press the Render button.

The files will be written to disk in the directory and format you specified in the Output Path parameter. When the files have finished rendering, you can specify them in the Image In node's Medium or Low file parameters.

Shortcut for Creating Multiple Proxy Sequences

To create proxies for multiple sequences, select all relevant Image In nodes in the Worksheet and drag and drop them into the Render Control view. This automatically creates an Image Out node connected to each, with corresponding entries in the Render Control list.

Each Render Control entry duplicates the Image Out node parameters, as described in *Chapter 11: Rendering Images* (p. 129), so that you can set the render resolution for all of the images in the Render Control list and then press the button at the top to render your proxies.

WORKING WITH PROXIES

You work the same way in RAYZ, with or without using proxy files, because RAYZ is designed so that all nodes can use proxy or full size images without adjusting parameters to accommodate a change in the resolution of the source image.

Parameter values, no matter what unit of measure is used to display them in the Node Panel, are stored internally at floating point precision; that is, as fractional values, to accommodate the switch from proxy to full size.

The flowchart of nodes you build in the Worksheet is a description of the shot. Image data is only calculated when you display it in the Viewer or render files to disk. Whenever you select Medium display in the Image Viewer, RAYZ uses your Medium proxy files, and when you select Full display, it uses the full size imagery.

NOTE:

The Status Bar displays the size data for the full size image, even if you are looking at Medium or Low images in the Viewer. Similarly, node parameters that display resolution data in pixel units, such as Translate and Scale, always show the pixel resolution of the full size image.

DEFINING MEDIUM AND LOW

As described above, you can select Full, Medium, or Low in the Image Viewer and the Render Control view. Full size is always 100 percent of the resolution of the node image. Medium and Low are fractions of full size: Medium is 50 percent by default, and Low is 25 percent.

You may want to adjust the default values that define Medium (0.5) and Low (0.25) depending on the actual sizes of your imagery. If so, open the Preferences panel, which is accessed from the RAYZ Edit menu:

1. In the Preferences panel, expand the Settings group in the left pane and select the Image Viewer Settings subgroup. This will display the Medium and Low preferences in the upper right pane.
2. Select Medium or Low to activate the associated parameter field in the lower right pane and type a new value into the field.

chapter 11

Rendering Images

The images in any node can be rendered to disk as a series of image files in the format and location you specify. Once you have designated the node imagery to render, you can initiate the actual rendering from within the RAYZ interface or from the command line.

IN THIS CHAPTER

You can render images in most popular file formats. For a complete list, see *Appendix B: Image File Formats Supported by RAYZ* (p. 441).

Adding Images to the Render List

You can designate the imagery in any node for rendering by connecting the node to an Image Out node.

The Image Out Node Panel provides parameters for specifying the name, filepath, and format RAYZ should use when writing the image files to disk. It also provides conversion options, if applicable, and format-specific compression options. For detailed information about these parameters, refer to the description of the "Image Out Node" in chapter 14, p. 189.

RAYZ automatically lists every Image Out node you create in the Render Control view. This means that you can control rendering of all Image Out nodes in one place.

The parameters in the Image Out Node Panel are identical to the corresponding parameter group in the Render Control. In fact, they are the same set of parameters, presented in different panels. When you make a change in an Image Out Node Panel, the change is reflected in the Render panel, where the corresponding parameter for that particular Image Out node will update automatically to match.

Multiple Render Entries for a Single Node

You can add the same node to the render list multiple times (which is the same as saying that you can connect a node to multiple Image Out nodes).

This enables you to specify different render parameters for each entry. For example, you might want to render the same imagery at different resolutions or quality levels, in different file formats, or render a different range of frames.

Be sure to give each entry a representative name, as illustrated by the first two entries to the Render Control panel shown in *Fig. 11.1* (p. 131).

Using Drag-and-Drop

You can drag-and-drop any node in the Worksheet into the Render Control view. This creates an Image Out node that is automatically connected to the node you dropped into the Render Control.

You can select as many nodes as you want and drag them all into the Render Control as a unit, which makes this method a great shortcut for adding multiple images to the render list at one time.

Rendering from the RAYZ Interface

There are several ways to render images in RAYZ:

- Press the Render button in the Image Out Node Panel.
- Press the Render button in the Render Control.
- Select the Render All command from the RAYZ Render menu.

During a render, a dialog box will appear to inform you of the progress of the render and let you know when rendering is complete.

Render Button in Image Out Node

To render the image sequence in a specific Image Out node, press the Render button in the Node Panel.

Using the Render Control Panel

You can press the Render button in the Render Control view to render every enabled entry in the render list.

To access the Render Control view, which is not displayed in the default layout, select Open Render Panel from the Render menu. This creates a new floating window with the Render Control view in it.

You can also add the Render Control view to the current layout by selecting it from the Views menu, or by replacing another view with it (press the F7 hotkey while the cursor is over a view and that view type will be replaced by the Render view).

Working With Render List Entries

You can modify render parameters for any entry before rendering. Expand the entry, if necessary, to access the parameters, which are identical to those in the corresponding "Image Out Node" (ch. 14, p. 189).

TIP:

Double-click the label for each entry to turn it into an editable text field, and give it a distinctive name. (The name of the corresponding Image Out node will update automatically to reflect the change.) This makes it easy to distinguish among multiple entries.

Disabling Entries

When you press the Render button, only those entries that are enabled will be rendered. You can disable any entry to exclude it from the next render by unchecking the box to the left of the entry name.

A red "x" icon will appear over the corresponding Image Out node to indicate that it has been disabled.

11.1 Render Control View: the "full_rez_comp" entry has been disabled, as checkbox next to entry name indicates; "low_rez_comp" is selected, as indicated by shading.

Removing Images from the Render List

To remove an image from the Render Control list, click the entry in the render list to select it first and then select Delete from the Layer Actions menu (right-hold on the selected entry to access). All currently selected entries will disappear from the list and their corresponding Image Out nodes will also be deleted.

Alternatively, you can delete the Image Out node directly. This will also remove the corresponding entry in the Render Control.

Render All Command

To render all images currently specified in the Render Control without opening the Render Control view, select the Render All command from the Render menu located in the Main Menu strip.

Rendering from the Command Line

To render imagery in a RAYZ file from a shell or Windows Command Prompt, without using the interface, type the following at the prompt:

```
rayz -render filename.rayz
```

In this example, “rayz -render” is the command that runs the render script and “filename.rayz” represents the actual name of the file that contains the imagery you want to render.

NOTE:

If you are not in the directory in which the RAYZ file is located, you must either change directories before typing the command or include the absolute pathname of the RAYZ file in the command.

This command is the equivalent of clicking the Render button in the Render Control view in the RAYZ application. All node imagery that has been added to the render list in that file will be written to disk in the file format and pathname specified therein (see also “Adding Images to the Render List” on p. 129).

Render Options

Several options are available to modify the render using this syntax:

rayz -render [options] filename.rayz

You can specify that multiple processors or tiling be used, and override some of the render settings in the RAYZ file.

COMMAND OPTION	DESCRIPTION
`-h or -help`	print a help screen of these options
`-p or -processors <number>`	specify number of processors to use for the render
`-log-directory <dirname>`	location of license log file (graild.log)
`-license-directory <dirname>`	where to search for license files (*.lic)
`-s or -start <number>`	specify start frame (applies to all image sequences rendered)
`-e or -end <number>`	specify end frame (applies to all image sequences rendered)

COMMAND OPTION	DESCRIPTION
`-i or -incr <number>`	specify increment (applies to all image sequences rendered)
`-L or -layer <name> <s> <e> <i>`	render specified sequence only
`-G or -global <name> <value>`	redefine global variable to use during render

The following command, for example, would render every other frame of every image sequence in the render list of a file named "my_comp.rayz":

```
rayz -render -i 2 my_comp.rayz
```

The short (`-i`) or the verbose (`--incr`) version of each command can be used interchangeably, but you must type a space between the option and the option value. For example, type "`-i 2`" or "`--incr 2`" to set the increment to 2.

RENDERING A SPECIFIC NODE SEQUENCE (-L)

The -L option is used to render a single sequence, when more than one image sequence in the RAYZ file has been specified for rendering.

This option should be followed by the name of the Image Out node (or the name of the entry in the Render Control list, which will be identical), along with values representing the start, end, and increment values.

To render an Image Out node named "full_rez_comp" in a file named "my_comp.rayz," you would type:

```
rayz -render -layer full_rez_comp 1 100 1 my_comp.rayz
```

OVERRIDING GLOBAL VARIABLE DEFINITIONS (-G)

You can override the definition of any global variable specified in a project file by using the following syntax with the -G option:

rayz -render -G <global name> <global value> filename.rayz

This option applies to the render only, it does not change the global value permanently. To change $JOB to a different filepath, for example, you might specify the following (note that you do not type the "dollar sign" in front of the variable):

```
rayz -render -G JOB /images/finals/shot1 my_comp.rayz
```

TILING OPTION (-T)

Use this option to reduce memory requirements during rendering. When the -tile option is invoked for a render, RAYZ will slice each image frame into tiles for processing, with the exact number of tiles being based on the available memory.

chapter 12

EDITING IMAGE SEQUENCES

There are several ways to change a sequence of image frames in RAYZ, depending on the type of changes you need to make.

You can use parameters in the Image In node to import a subset of the entire range of frames in the sequence and specify at which frame number in the shot the sequence should start. These parameters are explained in the section of the Image In node description on "Frame Range to Import" (ch. 14, p. 180). To perform more complex editing tasks, however, you can use the Clip Editor view or the Retime Footage utility.

IN THIS CHAPTER

Choosing an Appropriate Method

The Clip Editor displays all source node imagery as film strips on a timeline. You can shrink, stretch, or time-shift a clip or split it into segments of any length, each of which can be modified individually.

You do not have to import a sequence into RAYZ to modify it in Retime Footage. This utility is used to shrink or stretch a clip by editing a curve that controls the redistribution of frame data.

The Cinespeed option, which is available only in Retime Footage, analyzes how the image changes from frame to frame and uses this data along with the specified curve shape to retime the clip.

NOTE:

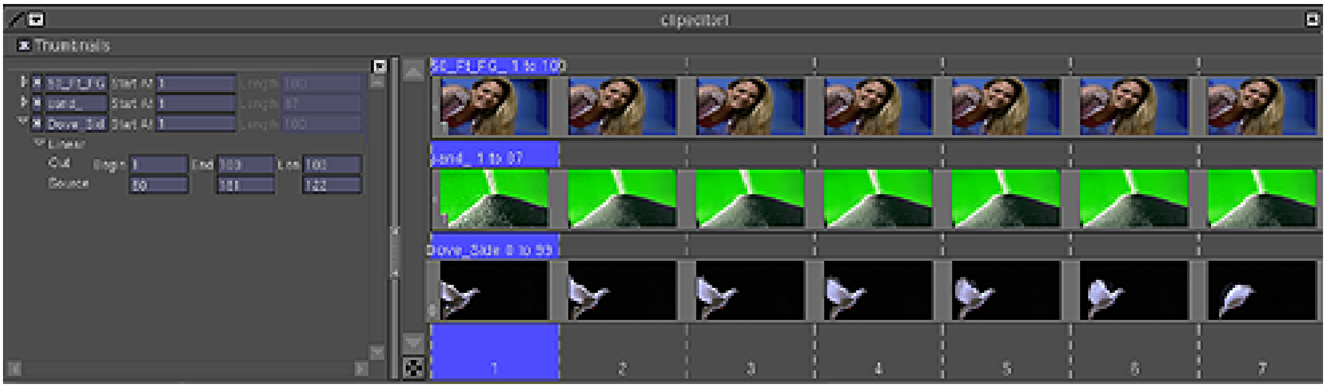
To re-sequence a clip in a downstream node (the output of a composite node, for example) use the "Sequence Node" (ch. 21, p. 415) instead.

USING THE CLIP EDITOR

The Clip Editor is used to re-sequence source imagery. The most basic, and most common, sequencing tasks can be done directly in the source node, without opening the Clip Editor. In the Clip Editor, however, you can view the sequence as a film roll and perform more complex edits.

In the Image In Node Panel, for example, you can specify the starting frame number to use for the sequence in the RAYZ network. You can also specify that only a subset of the total frames available on disk are imported. To split a sequence or add holds, on the other hand, you need to use the Clip Editor.

The Clip Editor view consists of a Clip Browser panel and a Timeline. All source nodes in the file are listed in the Clip Browser automatically. Only those clips you specify are displayed in the timeline, however.



12.1 The Clip Browser panel (left) can be used to control which clips are displayed in the Timeline (right).

THUMBNAIL DISPLAY

To display thumbnail images for each frame in the Timeline strip, check the Thumbnails box in the control strip at the top of the Clip Editor.

The “Main Clip Edit Controls” strip is accessed from the Tools menu on the left side of the Clip Editor title bar.

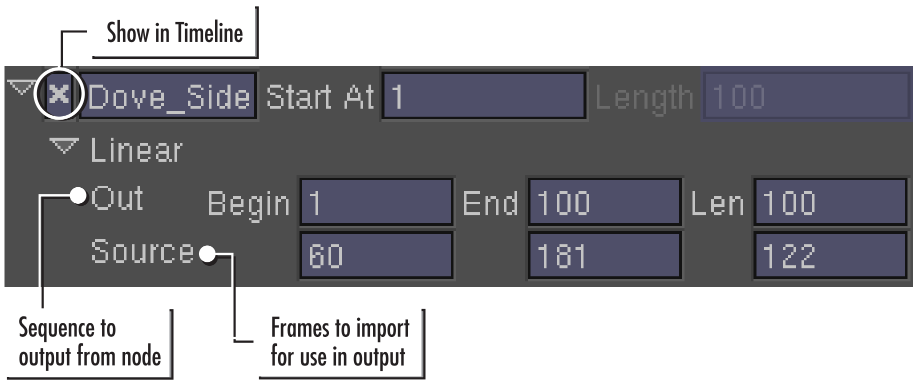
CLIP BROWSER

To display a clip in the timeline, check the associated box in the list of clips in the Clip Browser. All clips that have been enabled in the clip list appear in the timeline as frame strips. Each item in the clip list can be expanded to access the sequence parameters for that clip.

SEQUENCE PARAMETERS

Each sequence in the Clip Browser list can be expanded to access two rows of parameter fields:

- The top row is used to specify the beginning and ending frames and length of the output sequence in RAYZ.
- The bottom row also features Begin, End, and Length fields, but here they are used to specify which source frames on disk to assign to the output sequence.



12.2 Clip Browser entry for an Image In node named "Dove_Side."

SEGMENT PARAMETERS

When you split a clip in the Timeline, the clip list parameters for that clip are duplicated so that you can edit each segment individually.

12.3 This clip has been split into two segments and an ease function applied to the second segment.

For example, you might split a clip and reassign the same range of source frames to both segments to create a loop. Or you could assign a single frame to a segment and set the segment length to however many frames you wanted to hold the frame.

TIMELINE

The timeline in the Clip Editor is used to examine the frames visually and to re-sequence clips interactively. Any changes you make in the timeline are also reflected in the sequence parameters for that clip in the Clip Browser list.

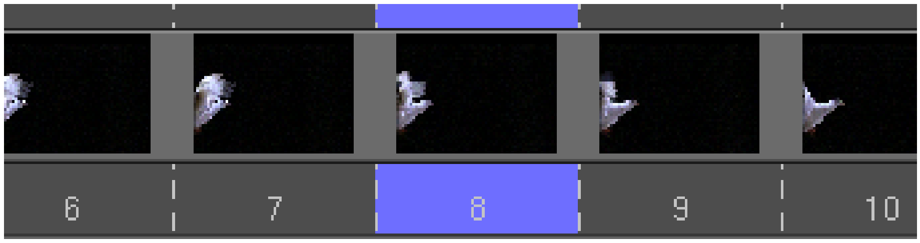
As explained in the previous section on the "Clip Browser" (p. 136), any clip that is checked in the clip list will appear in the timeline. The timeline displays each clip as a horizontal strip of frames and the clips are stacked vertically.

Like the Worksheet, you can move the timeline around in the view frame by dragging with the middle mouse button or scale it by holding down the Control key as you drag the scroll bar located at the bottom.

Current Frame Indicator

The blue bar that runs vertically through the timeline is the current frame indicator. Drag the blue bar to change the current frame. This updates the data displayed in the Image Viewer and Node Panel.

12.4 Current frame is 8, as indicated by blue bar.



> **TIP:**
>
> Press the **Backspace** key to snap the viewspace back to its default position with the first frame lined up on the left side of the timeline.

Thumbnail Display of Frame Images

To see thumbnail images of each frame, check the Thumbnails box in the control strip at the top of the Clip Editor.

Editing in the Timeline

The timeline is used to edit a clip by dragging the entire clip back and forth across the timeline to reposition it in time, or by dragging one end of the clip by the edit handle to change its length. Nonlinear interpolation functions can be applied to a clip, and the clip can be split into segments, each of which can be edited separately.

> **TIP:**
>
> Any edit you make in the timeline can be fine-tuned in the associated "Sequence Parameters" (p. 137) in the clip list.

Time-Shift

Drag the entire clip back and forth in the Clip Editor to time-shift it.

Time-Stretch/Shrink

Drag the edit handle (see *Fig. 12.5* below) at either end of the clip or segment in the direction that will stretch or shrink it.

You have two options for stretching a segment:

- You can stretch the length of the output without increasing the number of source frames used: just drag the edit handle.
- You can extend the length of the segment and the number of source frames used for it: hold down the Shift key as you drag the edit handle.

SPLIT

Ctrl-click between two frames or anywhere on the preceding frame to split a clip into two separate segments at that point. Edit handles will appear, separating the two frames, and new parameter group will be created for the segment in the clip list entry.

12.5 Example of a clip split into separate segments between frames 3 and 4. Note the edit handles that appear between the split frames.

SINGLE-FRAME SEGMENTS To split the first frame in a sequence from the rest of the frames (as when you want to add a hold), Ctrl-click on the frame. To split the last frame, Ctrl-click on the preceding frame. For a frame in the middle of a sequence, Ctrl-click on both sides of the frame.

REMOVING A SPLIT

To delete a segment, drag its edit handle until the segment disappears, or set the Out Length value for the segment to 0 in the clip list.

ADD HOLDS

To hold a frame, start by splitting it into a separate segment, as described above. Once the frame has been split, right-hold on the frame to access the popup function menu and select “Hold.” Then you can drag the frame by its edit handle to stretch the hold, or edit the length field for the new segment in the clip list.

APPLY FUNCTION

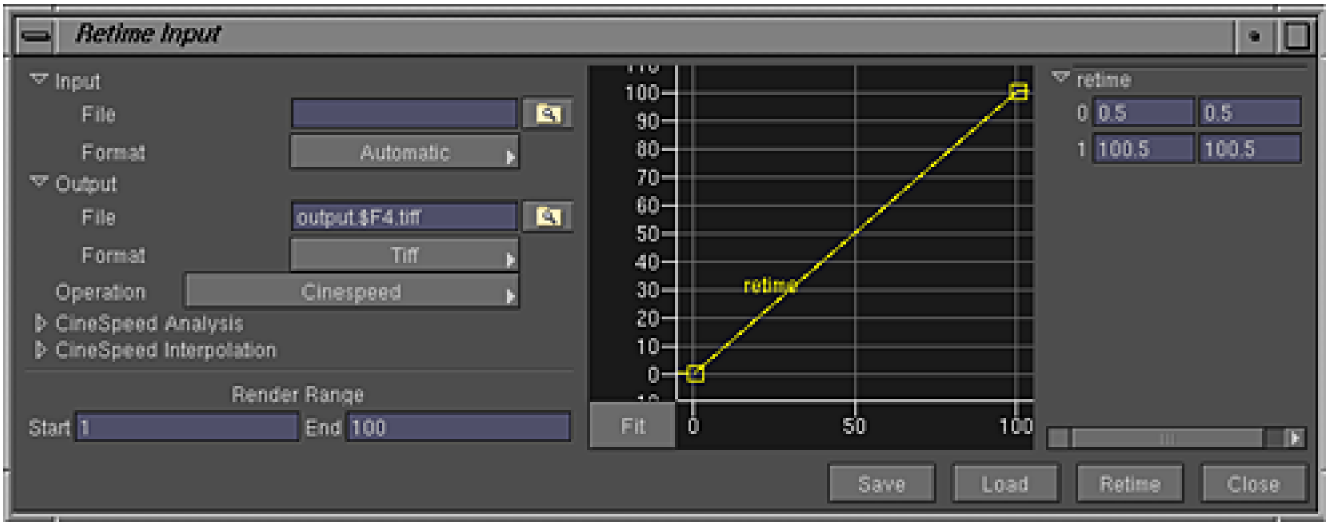
To apply a nonlinear interpolation such as an ease function, right-click and hold on any frame in the segment to access the popup menu, which includes the following functions:

- Linear
- Hold
- Ease
- Ease In
- Ease Out

The function you select from the menu will be applied to the clip (or to the current segment of the clip in the case of a split clip).

Using the Retime Footage Utility

The Retime Footage utility is a panel, accessed from within RAYZ, that is used to retime a sequence of image files stored on your local hard drive or on a shared storage device on your network.



12.6 The Retime Footage panel.

The retimed frames are written to disk in the location you specify, and you can then import the retimed footage into RAYZ for further modification and integration into a shot.

How Retime Footage Works

Retime Footage works by using a distribution curve to plot the rate of change from input to output. There are three retiming methods to choose from:

- Cinespeed (the default)
- Frame Averaging
- Frame Rounding

They all use the distribution curve to specify the retiming, but each uses a different method to calculate how the input image data is handled.

Frame Rounding

The fastest method is Frame Rounding, which duplicates the previous or next frame when lengthening a clip and throws away frames when shortening a clip. It chooses the frame to duplicate or discard based on its interpolation value in the retiming curve, which is rounded to the nearest whole number. If, for example, the value is 5.7, frame 6 will be duplicated; if the value is 5.2, however, frame 5 will be duplicated.

Frame Averaging

Frame Averaging, unlike Rounding, creates new frame images when necessary by averaging the pixels in the adjacent frames. The contribution of each adjacent frame is weighted based on the interpolation value in the retiming curve. If, as in the frame rounding example above, this value is 5.7, 70 percent of frame 6 and 30 percent of frame 5 will be used to generate the new frame.

Cinespeed

The Cinespeed option incorporates the Cinespeed algorithms used in the Cineon Time Warp utility. Cinespeed actually compares and analyzes the image data from frame to frame, creating vector/occlusion files that represent the areas and magnitude of change. This data is used, rather than simple averaging, to determine how frame images are blended.

These vector files can take a while to process, but they can be saved and reused if you need to fine-tune the shape of the distribution curve.

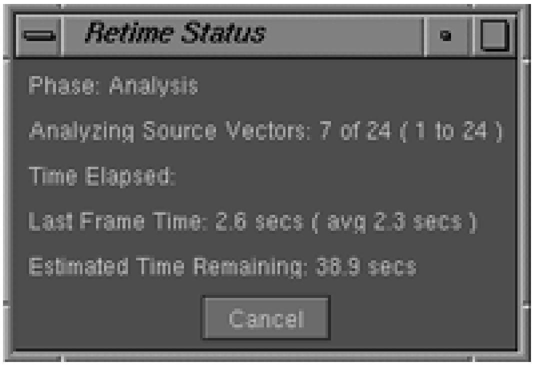
NOTE:

Unlike the other retiming operations, Cinespeed does not operate on four-channel images. You can specify RGBA image files for retiming with Cinespeed, but the alpha channel will not be processed and the retimed footage will be RGB only.

The Retime Footage Panel

To retime a sequence, open the Retime Footage panel and follow these steps:

1. Specify the image sequence to be retimed in the Input parameters.
2. Specify where the retimed footage should be saved, and in what format, in the Output parameters.
3. Select the method to use from the Operation menu: Averaging, Rounding, or Cinespeed. (If you select Cinespeed, refer to the following section on "Setting Cinespeed Analysis Options".)
4. Adjust the retiming curve, as described in "Adjusting the Retiming Curve" on p. 143.
5. Set the Render Range to match the sequence, or the subset of frames you want to retime (avoid processing unnecessary frames).
6. Press the Retime button, located at the bottom of the panel, to render the retimed footage.



12.7 Status information updates during the render.

While the retimed footage is being rendered, a status box pops up to display the current frame being rendered and the estimated time remaining.

When Cinespeed is used, it also indicates the phase being executed: Analysis, when the vector files are created, and Retime, when the retimed footage is being rendered.

SETTING CINESPEED ANALYSIS OPTIONS

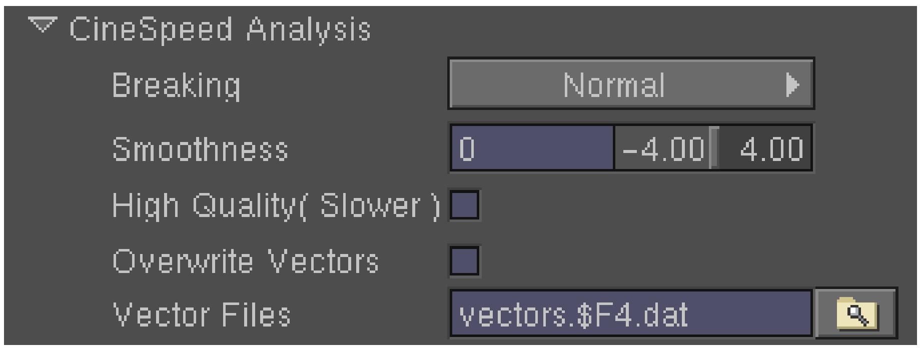
At minimum, you always want to set the Vector Files parameter in the CineSpeed Analysis parameter group. Optionally, you can also adjust the default values for the other parameters in this group.

VECTOR FILES

Use the Vector Files parameter to specify the name and location to which the vector files generated by this method should be saved. Vector files are given the name "vector.$F4.dat" by default. Whatever name you use, be sure to include the ".dat" extension.

OVERWRITE VECTORS

You can adjust the retiming curve and then reprocess a clip without rewriting the vector files by turning off the Overwrite Vectors option. However, if you need to change an analysis option, such as Breaking, you will have to overwrite the old vector files.



12.8 Cinespeed Analysis parameters in the Retime Footage panel.

BREAKING

Select an option from the Breaking menu to correspond to the amount of movement in the scene being retimed. The default Breaking option is Normal, however, you can also choose Harder, Soft, and Very Soft.

The busier the scene—the more moving objects it has—the "harder" the option should be. Be aware, however, that a harder option increases the time needed to process the sequence.

SMOOTHNESS

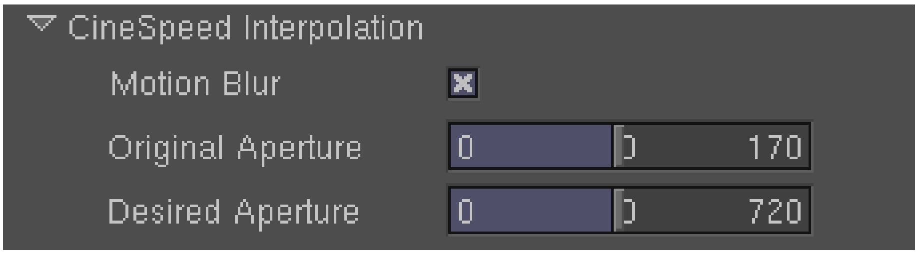
Use the Smoothness parameter to specify how important smoothness should be. The default value, 0, is effectively neutral. Setting a positive value indicates that smoothness is more important; a negative value, that it is less important.

HIGHER QUALITY (SLOWER)

By default, the combined luminance values of the RGB image are analyzed. To analyze the red, green, and blue channels individually, check the Higher Quality box. As the label states, this option will take much longer to analyze.

SETTING CINESPEED MOTION BLUR OPTIONS

To add motion blur to the retimed footage, check the Motion Blur box.



12.9 Cinespeed Motion Blur parameters in the Retime Footage panel.

Additional parameters, Original and Desired Aperture, will become active which can be used to adjust the default aperture angles for the motion blur.

ORIGINAL APERTURE

Use the Original Aperture parameter to specify the aperture angle of the input in a range of 0–170 degrees. This parameter value should be set to match the angle used when the footage was shot, if known; otherwise leave it at the default value of 0. For CG imagery, use 0.

DESIRED APERTURE

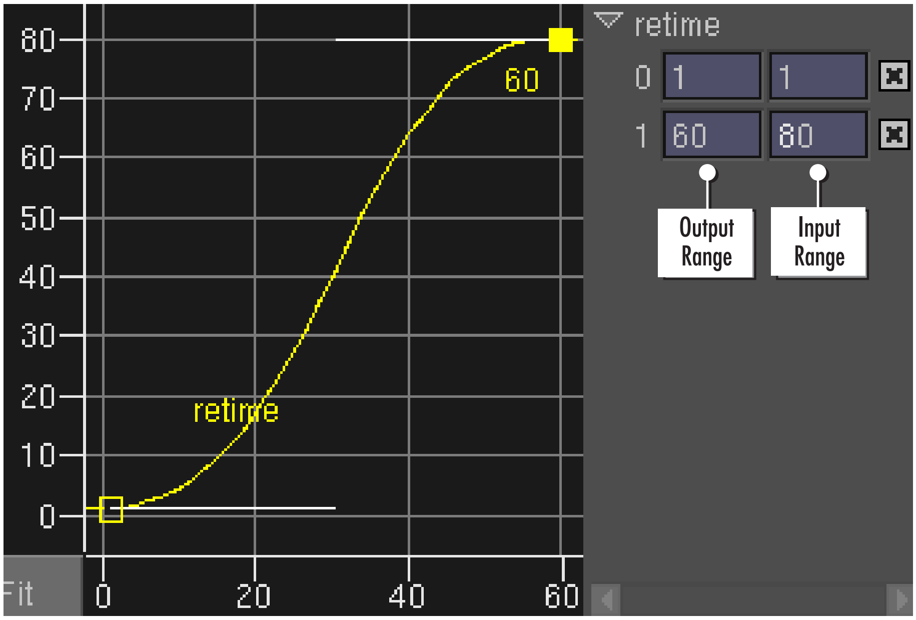
Use the Desired Aperture parameter to specify the aperture of the motion blurred output in a range of 0–720 degrees.

ADJUSTING THE RETIMING CURVE

You can manipulate the curve directly in the graph, or use the associated Keypoint Viewer to edit the curve values numerically.

You can add keypoints to the curve and specify different distribution functions for each curve segment:

- To add a keypoint, Ctrl-click on the curve.
- To change the default distribution function, which is linear, right-hold on a segment and select a function from the popup menu.



12.10 Retime curve graph (left) and Keypoint Viewer (right). The vertical axis of the graph represents the input frame range; the horizontal axis, the retimed frame range.

You can save retiming curves, just as you would animation curves in the Curve Editor, and reuse them on different sequences. Right-hold in the graph, anywhere except on the curve, to access the popup menu used to save and load retiming curves.

The curve graph interface works like the one in the Curve Editor view. For detailed information about manipulating a curve, refer to “Creating and Editing Keypoints” in chapter 8, p. 109, “Controlling Interpolation of Curve Values” (ch. 8, p. 109), and “Editing Keypoints Numerically” (ch. 8, p. 111).

Render Range Parameters

In most cases you will set the Render Range parameters to fit the range of the retimed output sequence. However, you can also specify a subset of the retimed output when applicable.

To adjust the Render Range parameters, simply type new values in the Start and End fields.

TIP:

To run a clip backward in time, enter the last frame in the Start field and the first frame in the End field.

SAVING RETIME FOOTAGE SETTINGS

You can save the current panel settings to a file and reload them into the Retime Footage panel at any time by using the Save and Load buttons located at the bottom of the panel.

Press the Save button and a dialog box will appear in which you specify the name and location of the file, which should have a ".retime" file extension. RAYZ will append this extension by default when you save the retime settings file.

To load the saved settings, press the Load button and select a saved retime settings file in the dialog box, navigating to the appropriate directory if necessary.

RETIMING SEQUENCES FROM THE COMMAND LINE

You can retime an image sequence from the command line, if the settings were saved as a ".retime" file, by using the `rayz -retime` command. Specify the start and end frames of the retimed sequence and the name of the .retime file:

rayz -retime -start <number> -end <number> filename.retime

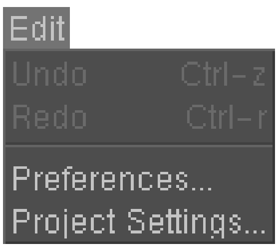
COMMAND OPTION	DESCRIPTION
`-input frames <input images>`	specify sequence to retime
`-output frames <output images>`	retimed output sequence
`-curve <curve string>`	specify a saved retiming curve
`-frameavg`	use frame average method
`-frameround`	use frame rounding method
`-cinespeed <breaking> <smoothness> <vector files>`	use Cinespeed method
`-vector-overwrite`	overwrite existing Cinespeed vector files
`-highquality`	use Higher Quality Cinespeed option
`-motionblur <original aperture> <desired aperture>`	add Cinespeed motion blur

chapter 13

Setting Preferences

You can set preferences in RAYZ to customize the interface and specify default parameter settings. General Preference settings apply to all RAYZ project files, while Project Settings apply only to the current file.

In This Chapter



13.1 The RAYZ Edit menu.

Preferences can be edited in the Preferences panel and in the Project Settings panel, both of which are accessed from the RAYZ Edit menu.

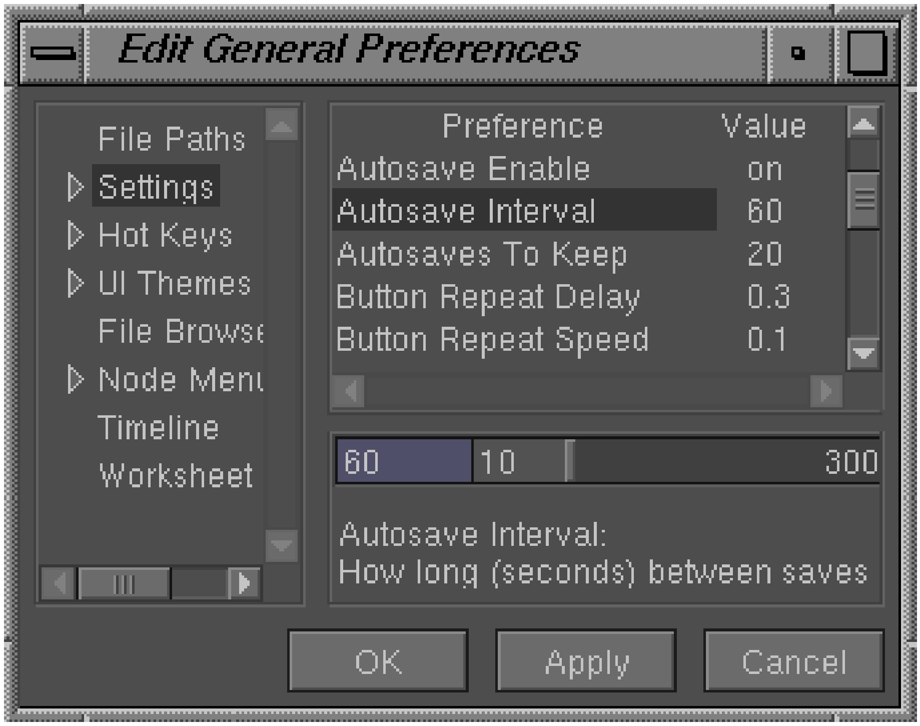
Using the Panel Interface

The Preferences and Project Settings panels use the same interface, which is divided into three main panes:

- The pane on the left lists preference categories in alphabetical order.
- When you select a category in the left pane, a list of preference items for that category appears in the upper right pane.
- When you select an item in the upper right pane, editable parameters for that item appear in the lower right pane, along with a description of the purpose of that preference.

The first two panes are used to select the preference to be set or modified, and the third pane is where you specify the preference value.

13.2 Example of Preferences Panel being used to set Autosave Interval.



Editing General Preferences

The Preferences panel is used to customize the look of the interface and edit default search paths, hotkey assignments, and file browser behavior. You can also specify how nodes are grouped into menus.

Where Preference Settings Are Saved

The file that holds the current Preferences Panel settings is located by default in .rayz/2.2/Preferences/Defaults.pref.

To change the path to the Defaults.pref file, set an environment variable by typing the following at the prompt in a shell, replacing "[pathname]" with the actual path to use:

```
setenv RAYZ_PREFERENCES_PATH /[pathname]
```

To restore the factory default preferences, delete the Defaults.pref file and RAYZ will create a new one with the factory default settings.

You can copy the Defaults.pref file from one RAYZ workstation to another to share preference settings with other users.

File Paths

This category specifies the file path RAYZ will use to search for various directories and files. To change a default path, select a File Path preference. A text entry field will appear in which you can type a new search path.

To enter multiple file paths into a search path preference, use the pipe character (|) to separate each pathname.

SEARCH PATH	DESCRIPTION
Temporary Directory	path to directory RAYZ should use to write temporary cache, if necessary, while application is running
Layout Search Path	path to directory of custom layouts saved by the user
Help Search Path	path to directory containing the RAYZ HTML manual used for online help
Plugin Search Path	path to directory containing RAYZ plugins
Node Preset Search Path	path to directory containing node presets saved by the user
Custom Node Search Path	path to directory containing custom nodes saved by the user
System Search Path	path to directory containing system binaries
Font Search Path	path to directory containing fonts used by RAYZ

SETTINGS

This category specifies autosave and help options, as well as how mouse events should be interpreted. Most of these preferences offer numeric data entry parameters.

SETTING	DESCRIPTION
Autosave Days to Keep	specifies how long to keep autosaved files
Autosave Directory	specifies directory in which autosaved files should be stored
Autosave Enable	turns autosave on and off
Autosave Interval	specifies the interval between autosaves
Button Repeat Delay	specifies the interval before buttons that repeat start repeating
Button Repeat Speed	once a button starts repeating, specifies the interval between repeats
Double Click Timeout	maximum time allowed between mouse clicks that should be interpreted as a double-click
Help Browser	specifies the HTML browser application in which to open the online help files
Help Tag Enable	sets the default state of the help tag toggle
Help Tag Start Timeout	specifies the delay time before help tags pop up

SETTING	DESCRIPTION
Marking Menu Delay	specifies the delay time before menus pop up
Mouse Sensitivity	specifies how close to an object the cursor must be to affect it
Nudge Size (Large)	Number of pixels to move when using Shift modifier with the nudge (arrow) keys
Nudge Size (Small)	Number of pixels to move when using the nudge (arrow) keys
Window Border Sensitivity	specifies how far past a window border the cursor can range while autoscrolling without being interpreted as being outside the window

IMAGE VIEWER SETTINGS

The Image Viewer settings are accessed by expanding the Settings group. They include preferences for Magni-zoom and the color of the background area in the Image Viewer as well as the scale factors to use when viewing an image at other than full size.

Click on the Image Viewer group to select it, rather than expanding it, and the Magni-zoom and Outside Image Color preferences will appear in the right pane.

MAGNI-ZOOM PREFERENCES

There are two Magni-zoom preferences, one for the scale factor to use in the zoom box and the other to set the size of the zoom box.

VIEWSPACE COLOR

This preference specifies the color RAYZ should use to fill in the viewspace in the background of the Image Viewer. By default the viewspace color is set to black. To change it, click the Outside Image Color preference to access the standard RAYZ color parameters, which are described in detail in “Using the Color Parameters” in chapter 14 (p. 168).

MEDIUM AND LOW SCALE FACTORS

Expand the Image Viewer group to access the Scale Factors item. When you click on Scale Factors, the Medium and Low preferences appear in the right pane. These preference settings specify the scale factor to use when Medium or Low is selected from the Size menu in the Image Viewer.

By default, Medium is set to 50 percent of full size, and Low to 25 percent, where “full size” refers to the native spatial resolution of the image.

UI THEMES

A UI theme is a set of values that define the current look of the RAYZ interface, including colors to use for backgrounds, foregrounds, borders, buttons, text, and other elements, as well as the type of bevel and border width to use for buttons, menus, etc. Themes also specify the font to use for labels and other text.

Current Look Settings

The Current Look Settings are used to edit the values assigned to the currently selected theme. You can specify the color to use for interface elements throughout RAYZ by selecting a look preference from the list and adjusting the associated color parameters.

These color parameters work like the equivalent parameters available in a number Node Panels. For more information about them, refer to the section on "Using the Color Parameters" in chapter 14 (p. 168).

The look preferences that set a value other than color, such as font or border style, provide menus in which you can choose a value.

File Browser

The file browser is the dialog box that appears when you open or save RAYZ files or import image files in the Image In node. The File Browser preferences affect the default contents of the navigation menus in the file browser, as well as how files are listed and displayed.

Node Menu Group

These preferences are used to customize node menus by changing the order in which node menus appear and changing the menu to which any node is assigned.

Timeline

This group of preferences controls size and spacing of filmstrips in the timeline of the Clip Editor. You can adjust the horizontal and vertical size of the frames and the spacing of the vertical strips between frames.

With smaller frames and spacers, you can see more frames at once, but larger frames and spacers are easier to manipulate.

Worksheet

The Worksheet preferences are used to specify the size of the nodes in the Worksheet, the size of the (unattached) node inputs, and the size of the invisible grid to which new nodes align when placed in the Worksheet.

Text Only Node Menu Strip

In addition, to save space in the Worksheet, you can change the Icon Node Menu preference from On (the default) to Off to display a text-only node menu strip in the Worksheet in place of the iconic node menu.

Straight Lines

Another Worksheet preference you can change is how flowlines are drawn. The Straight Lines preference is off by default, which means that flowlines are drawn with right angles when they connect nodes that are not located on the same horizontal. If you turn Straight Lines on, the flowlines are drawn diagonally in this case.

HOTKEYS

The Hotkeys category in the General Preferences panel is used to create or modify keyboard shortcuts—keys that can be pressed to issue commands in place of using the mouse to manipulate the RAYZ interface.

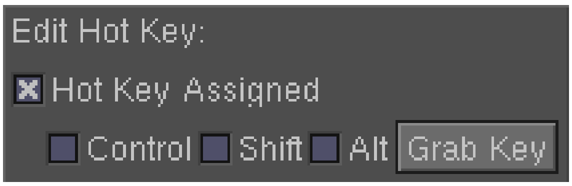
Virtually every command in RAYZ can be assigned a keyboard equivalent, or hotkey. And default hotkey assignments can be modified or eliminated.

Hotkeys are divided into two categories, hotkeys for Global commands and hotkeys for Context Sensitive commands:

- **Global** commands are common to multiple views. For example, you can copy and paste nodes in the Worksheet and text wherever text fields appear, using the same hotkeys in every case.
- **Context sensitive** commands are specific to a view or task, such as the channel display commands in the Image Viewer, which have no application in other views.

HOTKEY EDITOR

When you select a command from the Hotkeys list, a hotkey editor is displayed. The top line of the editor consists of a checkbox that turns the hotkey assignment on and off for the command. When the box is checked, the controls on the second line become active.



13.3 Hotkey Editor in the Preferences Panel.

MODIFIER KEYS

The first three checkboxes are used to assign an optional modifier key: Control, Shift, and/or Alt. Modifier keys are held down while the assigned key is pressed.

Typical examples would be the ubiquitous cut, copy, and paste hotkeys: Ctrl-x, Ctrl-c, and Ctrl-v.

GRAB KEY BUTTON

The final button is used to assign the hotkey. This button is labeled “Grab Key” unless a hotkey assignment has already been made, in which case it is labeled with the current key assigned.

Assigning a Key

1. Check the "Hotkey Assigned" box to activate the controls.
2. Check the box for any modifier keys you want to use (this is optional).
3. Click the Grab Key button to tell RAYZ that the next keyboard key you press should be assigned as the hotkey.
4. Press the key on the keyboard that you want to assign.

If that key (or key combination, if you also specified one or more modifier keys) is already assigned to another item, a dialog box will pop up to inform you of this. You then have the option to reassign the key to the current item or to cancel, in which case you can try another key.

Default Hotkeys in RAYZ

The single most useful hotkey in RAYZ may be the **Space bar**, which can be held down while working in any view to access the popup node menu, as described in "Creating and Connecting Nodes" in chapter 5, p. 46.

The other "factory default" hotkey assignments for common commands are listed below in functional groupings.

FILE COMMANDS (GLOBAL)	HOTKEY
New File	Ctrl-n
Open File	Ctrl-o
Close Window	Ctrl-w
Quit RAYZ	Ctrl-q
Help	F1
Import Footage	Ctrl-i

The cut, copy, paste, and delete commands in the following list apply to selected text (including numerical values) in parameter fields, as well as to curves in the Curve Editor and nodes in the Worksheet.

Selected or targeted nodes in the Worksheet can be deleted, as described in "Deleting Nodes" in chapter 5, p. 53. The undo and redo commands are described in "Levels of Undo and Redo" in chapter 9, p. 119.

EDIT COMMANDS (GLOBAL)	HOTKEY
Cut	Ctrl-x
Copy	Ctrl-c
Paste	Ctrl-v
Undo	Ctrl-z
Redo	Ctrl-r
Delete	Del

The frame navigation commands change the current frame selected in the Time Scooter and Flipbook, which in turn affects the data displayed in the views:

Frame Navigation Commands (Global)	Hotkey
Go to Previous Frame	, (comma key)
Go to Next Frame	. (period key)
Go to First Frame	Shift-,
Go to Last Frame	Shift-.

For more information on working with the following view options, see "Changing the Layout" in chapter 4, p. 37 and "Expanding and Collapsing Grouped Controls" (ch. 4, p. 35).

View Commands (Global)	Hotkey
Replace Current View with Clip Editor	F5
Replace Current View with Curve Editor	F2
Replace Current View with Image Viewer	F3
Replace Current View with Node Panel	F4
Replace Current View with Worksheet	F6
Replace Current View with Render Control	F7
Maximize View	F11
Restore Default Layout	F10
Expand/Collapse All Toggle	o
Expand/Collapse Toggle	Shift-o

The following commands affect the scale and position of the image displayed in the Viewer and the node network in the Worksheet:

Image Viewer and Worksheet Commands	Hotkey
Center in View (also works in Clip Editor)	Backspace
Center and Reset Scale to Normal (also works in Clip Editor)	Home
Zoom In (scale up)	=
Zoom Out (scale down)	-
Nudge Up	up arrow
Nudge Down	down arrow
Nudge Left	left arrow
Nudge Right	right arrow
Nudge in Larger Increments	Shift-arrow

The next group of commands control how image data is displayed in the Image Viewer.

IMAGE VIEWER COMMANDS	HOTKEY
Display RGB (Color) Channels	c
Display Alpha Channel	a
Display Red Channel	r
Display Green Channel	g
Display Blue Channel	b
Display Other (Fifth) Channel	o
Display Luminance Channel	l
Source Image Display (cycle thru each Source menu option for current node in turn)	s
Toggle Buffer Display (front buffer/reference buffer)	F8
Auto Update Mode	Shift-a
Continuous Update Mode	Shift-c
Update Image (when in Manual Update Mode)	y
Play Flipbook Forward	k
Play Flipbook Backward	j
Magni-zoom Toggle	m
Full Screen Mode Toggle ("blackout" mode)	n

The node commands apply to the node you target (position the cursor over) when the command is issued, with the exception of Find Nodes, which can be invoked when the cursor is anywhere in the Worksheet:

NODE COMMANDS	HOTKEY
Create Separate Image Viewer Window	1
Create Separate Node Panel Window	2
Create Separate Curve Editor Window	3
Pass Thru Toggle (also works in Node Panel)	p
Thumbnail Display Toggle	t
Icon Display Toggle	i
Find Nodes	Ctrl-f

The following commands apply to the Node Panel parameter over which the cursor is positioned:

NODE PANEL COMMANDS	HOTKEY
Open Expression Editor	Shift-e
Export Parameter to Group Node Panel (applies to nodes within a Group node)	e
"Un-export" Group Parameter (remove from Group Node Panel)	u

Editing Project Settings

Project settings are preferences that apply to the current project file only, such as default values for node parameters and swatch values to use in color palettes. Many of these settings are used to set defaults that match the type of imagery you are using on a project.

Project Settings are saved in the project file itself.

The Project Settings panel interface is described in the section on "Using the Panel Interface" (p. 147). Specific project settings are described next.

Current Colors

Expand the Current Colors group to specify the default values of colors used to display various interface elements, including

- overlays in the Image Viewer,
- underlays in the Worksheet, and
- curves in the Curve Editor.

And you can use the Image Sample Color Palette preferences to set the default fill colors assigned to swatches in the Color Picker.

The individual color preference values are set in the same way for all of the Current Color preferences, by using the standard RAYZ color parameters. For more information, refer to the section on "Using the Color Parameters" in chapter 14 (p. 168).

Node Defaults

Select Node Defaults in the left pane to access a list of default node parameter settings in the right pane. For example, the default formula for computing luminance is film standard. For a video project, however, you could change the Default Luminance Model preference to NTSC.

TIP:

Whenever you find yourself repeatedly changing the default setting in a Node Panel for parameters that control basic setup or output options, consider changing the default for that item in the Project Settings panel.

Settings

Click on the Settings item itself to display a list of default settings relating to the type of imagery you are using on a project, such as the default size and bit depth to use for new Color, Gradient, and other source nodes.

Time Display Style

This is where you specify the units in which time will be displayed in the sliders of the Time Scooter and Flipbook. You can display time in Frames (the default), Seconds, Edge Code, or Time Code.

Playback Rate

Use the Frames per Second or Frames per Foot settings to specify the default playback rate in the Flipbook.

Scale Factors

You can expand the Settings group to access the Thumbnail Scale Factors parameters, which specify how thumbnail node images, if displayed, are scaled when you zoom out in the Worksheet.

Globals

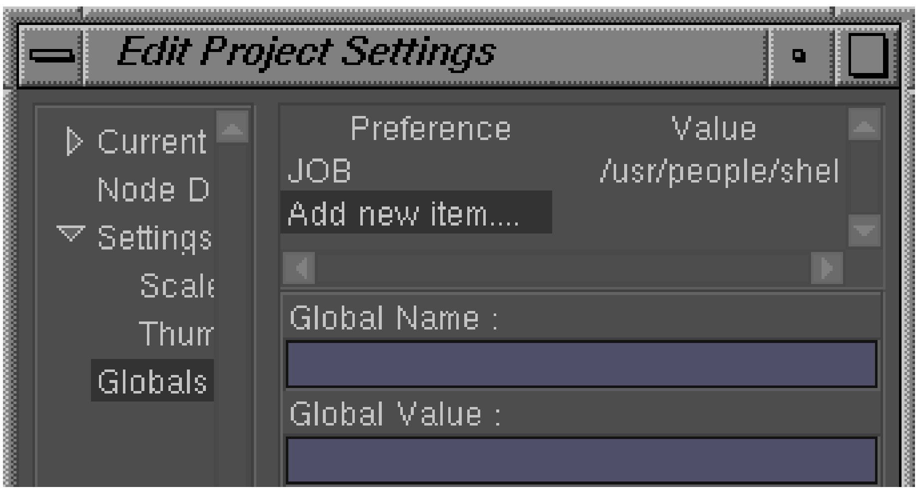
Select the Globals category to get a list all global variables currently defined.

Defining $JOB

The $JOB global is set by default to the directory from which RAYZ was launched. To redefine it, click on the JOB item in the list in the upper pane and type a new path into the text field in the lower pane.

Creating a New Global

Click "Add New Item" to display the Globals editor in the lower pane.



13.4 Globals Editor is used to create new global variables and redefine existing ones.

GLOBAL NAME Type the name of the global in the first text field, using all uppercase letters (A to Z; no numbers or non-alphabetic characters). Do not type the "dollar sign" that is used to signify the global when typing command strings.

GLOBAL VALUE Define the global in the second field.

DELETING A GLOBAL

13.5 Press the Delete button to delete the currently selected global.

The Delete button becomes active whenever a global is selected in the Preferences list. Press the button to delete it.

PART

Three

NODE REFERENCE

Part III provides complete descriptions of each node in RAYZ, organized in chapters according to node category.

IN PART III: NODE REFERENCE

Concepts, techniques, and parameters that are common to nodes of a particular category are explained at the beginning of that chapter. For example, the introduction to a node reference chapter may include guidelines for choosing the most suitable node for a task.

To find the description of any individual node without knowing what default category it belongs in, use the following alphabetical node listing.

A-Z LIST OF ALL NODES

chapter 14

SOURCE NODES

The Source node menu contains the nodes used to create and import imagery. The Image In node is used to import digital image data from disk for modification. Other source nodes are used to create new elements, such as color fields and gradients.

The Source menu also includes the Image Out node, which is used to specify that the image data flowing into it will be rendered to disk.

IN THIS CHAPTER

FRAME ATTRIBUTE PARAMETERS COMMON TO SOURCE NODES

The source nodes in RAYZ provide a set of parameters used to specify basic attributes of the output frame:

- size (spatial resolution)
- bit depth (per channel)
- channels
- frame range
- backing color

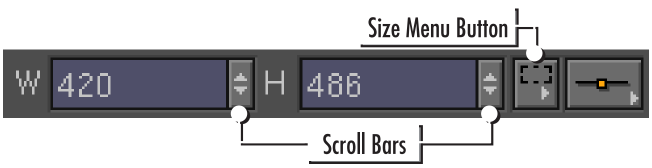
14.1 Frame Attribute Parameters.

The exception is the Image In node, which imports existing image files from disk rather than creating images from scratch. The following descriptions of the individual frame attribute parameters will note if there are any differences in a parameter among the nodes.

SIZE PARAMETER

This parameter enables you to specify the width and height of the frame that will be output by a source node.

You can key a pair of values into the Width and Height fields, or you can use the associated menu to access a list of commonly used film and video resolutions. When you select a resolution from the list, the corresponding XY values are entered into the parameter fields.



14.2 Drag a scroll bar up to increase field value; drag down to decrease. To change the value in finer increments, drag the scroll bar to the left or right—the value will change more gradually.

Use any of the following methods to adjust the size parameters:

- Type new pixel values into the width (W) and height (H) fields.
- Drag the scroll-arrows up or down to increase or decrease the value.
- Select a spatial resolution from the Size menu, which contains a list of many video, HD, film, and large format film resolutions.

Size Exceptions

The **Color Bars** node does not include a Size parameter, as the output size is limited to the resolution associated with the specified video type (NTSC or PAL).

In the **Image In** node, the Size parameter specifies the resolution of the full size imagery when proxies are used in lieu of full size images, rather than in conjunction with them, as explained in “Specifying Full Size” (p. 181) in the Image In node description. (To change the resolution of imported imagery, connect the Image In node to an appropriate node such as Resize.)

Bit Depth Menu

The Bit Depth menu enables you to select 8-bit, 16-bit, or floating point (“32-bit”) per channel as the bit depth of the output.

In the Image In node, you can convert the bit depth of imported images by using the “Conversion Parameters” (p. 181).

Channels Menu

The Channels menu enables you to select the number and type of image channels to output: RGB, RGBA, or Alpha only. The alpha, or matte, channel contains the data that determines the transparency level of the RGB elements of each pixel.

In the Image In node, you can add an alpha channel to imported imagery by checking the Alpha parameter, or you can connect the Image In node to a “Channel Swap Node” (ch. 16, p. 248) to make other adjustments.

Frame Range Parameters

The source nodes include parameters in which you can specify the frame number to start at and the total length of the sequence. The default values are set to match the current range specified in the Time Scooter. To adjust the range, simply type new values into the Start At and Length fields.

TIP:

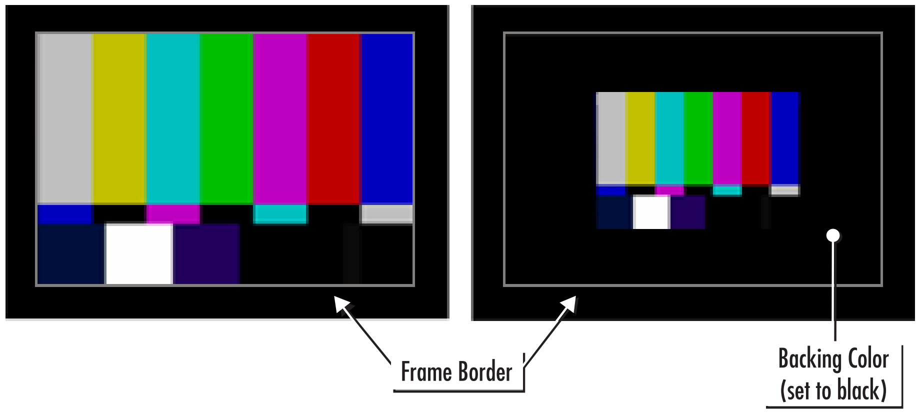
Use the Clip Editor to do more complex re-sequencing, such as looping an animated ramp in a Gradient node.

Frame Range Exceptions

The frame range parameters in the Image In node can be expanded to specify which frames in the sequence to import, as explained in the section of the Image In node description on "Frame Range to Import" (p. 180).

Backing Color Parameters

The Backing Color parameter value only becomes relevant in the event that an image is scaled or otherwise transformed such that some areas within the full size frame border no longer contain image data. For example, if you scale down an image in a Transform node, RAYZ will fill the empty pixels inside the frame border with the designated backing color.



14.3 The backing color becomes relevant when an image (left) is scaled down (right) or transformed in any other way that leaves part of the frame "empty."

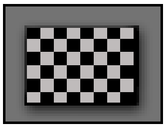
By default the backing color is black (0) for all channels, and in most cases this is appropriate. In fact, you may never need to adjust the default value. If you wish to assign another color to the backing area, however, you can do so in the source node. The backing color assigned in the source node is used throughout the node network.

The Backing Color parameter controls work exactly like those described in the section on "Using the Color Parameters" (p. 168) in the Color node description.

Backing Color Exceptions

The Backing Color parameters for the Image In node are located within the "Image Description Parameters" (p. 186).

CHECKER NODE



The Checker node creates a checkerboard pattern in the colors and dimensions you specify.

CHECKER PARAMETERS

You can specify frame size, bit depth, channels, and range of the output as described in the section on "Frame Attribute Parameters Common to Source Nodes" (p. 162).

The other parameters in the Checker Node Panel, which are described below, are used to set the size of the squares that make up the checkerboard (the Square Size parameter) and the two colors that distinguish the alternating pattern of squares (Square Color and Background Color).

The Square Color and Background Color parameters use the standard RAYZ color controls, which are described in the section on "Using the Color Parameters" on p. 168.

SQUARE COLOR

Use the Square Color parameter controls to specify the color of the squares. The square color is set to white by default, including the Alpha channel, assuming you have specified RGBA output in the Channels menu.

BACKGROUND COLOR

Use the Background Color parameter controls to specify the color of the background squares. The background color is set to black by default, including the Alpha channel, assuming you have specified RGBA output in the Channels menu.

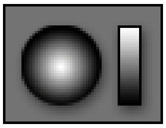
If you want the entire checkerboard pattern to be opaque, remember to set the Alpha channel value in the Background Color parameters to maximum value.

SQUARE SIZE

Use the X and Y fields of the Square Size parameter to specify the size, in pixels, of the squares that make up the checkerboard pattern. The default size is 64 x 64.

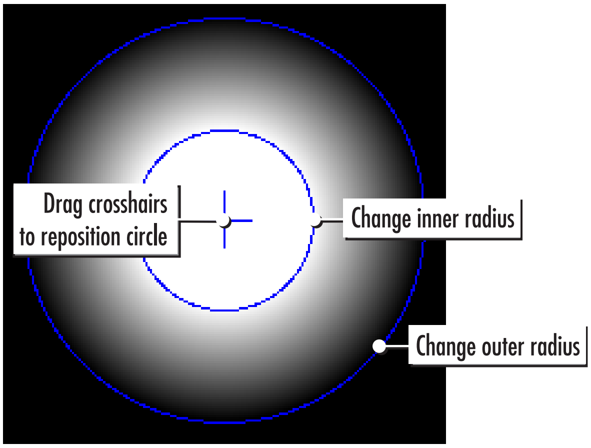
The squares do not actually have to be square; if you enter a different size in each field, you will create a pattern of rectangles.

CIRCLE RAMP NODE



The Circle Ramp node creates a circular gradient in the color and dimensions you specify. For example, you could use the Circle Ramp node to create a vignette mask by defining a circle with graduated opacity from center to edge and applying it to the alpha channel only.

You can use the Circle Ramp overlay in the Image Viewer to adjust the position and radius interactively and the corresponding parameter values in the Node Panel will update accordingly.



14.4 Circle Ramp Overlay: Drag the overlay controls in the Image Viewer to adjust the ramp characteristics.

CIRCLE RAMP PARAMETERS

You can specify frame size, bit depth, channels, and range, as described in the section on "Frame Attribute Parameters Common to Source Nodes" (p. 162). The other Circle Ramp parameters are described next.

CENTER AND EDGE COLOR PARAMETERS

The Center Color and Edge Color parameters are used to set the color values of the center and edge of the radius. The values in between are interpolated linearly.

Both parameter groups feature identical color controls which are standard to RAYZ. If you need more information, these controls are described in detail in the section on "Using the Color Parameters" (p. 168) in the Color node description.

NOTE:

When an alpha channel has been selected for output, use the Alpha channel parameter within each color group to set the opacity. By default, the Center Color alpha value is set to 1 and the Edge Color alpha is set to 0. Change the Channel display in the Image Viewer to Alpha to see the result.

Center

This parameter is used to position the circle in the frame. The center of the circle is positioned at the *x,y* coordinates entered into the Center parameter fields. The coordinate values are expressed as a percentage of the total width and height of the output frame, so that the default values of 0.5 for both fields will center the circle in the frame.

The Center parameter is not bounded—you can set values outside the range of 0 to 1 to center the circle outside of the frame.

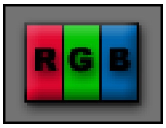
Radius

This parameter sets the size of the circle by specifying its radius. A pair of fields are used that represent the inner (I) and outer (O) limits of the radius, expressed as a percentage of the *width* of the output frame. The outgoing radius is not bounded; it can be set to values greater than 1.

Aspect

This parameter controls the aspect ratio of the circle, which can be adjusted from a true circle to an ellipse. To flatten the circle into a horizontal ellipse, decrease the Aspect value from its default of 1; to create a vertical ellipse, increase the value.

COLOR NODE



The Color node enables you to create a solid field of color using the values you specify in the Node Panel.

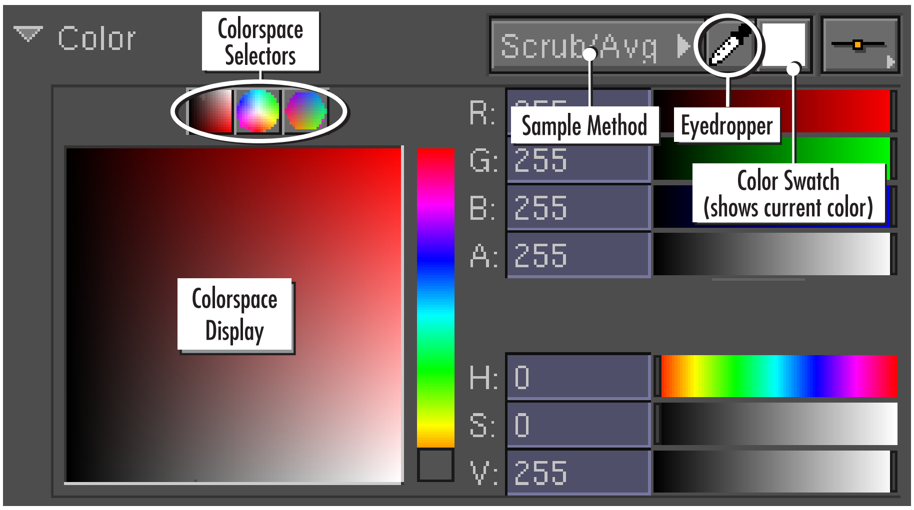
The output of the Color node will be RGB(A) color channels, but you can choose the model in which to specify the color: RGB, HSV, or Constant Luminance. You can also control the value of the alpha channel output, if any.

The Color Node Panel consists of the Color parameter group and the Frame Attribute parameters. The frame parameters, including size, bit-depth, channels, range, and backing color, are described in "Frame Attribute Parameters Common to Source Nodes" (p. 162). The color parameters are described next.

USING THE COLOR PARAMETERS

The Color parameter group in the Color Node Panel is also available in other nodes in which you might need to specify color values:

- You can specify a color by selecting it from the popup spectrum bar, which is accessed from the color swatch, or you can use the eyedropper tool to sample pixel values in any image or swatch.
- You can also expand the Color group to enter numerical values or to select a color interactively by clicking in a colorspace display (RGB, HSV, or Constant Luminance).



14.5 Color parameter group expanded, with the RGB tools selected. Note the colorspace selector buttons, which are used to display the RGB, HSV, or Constant Luminance controls.

COLOR PICKER TOOLS

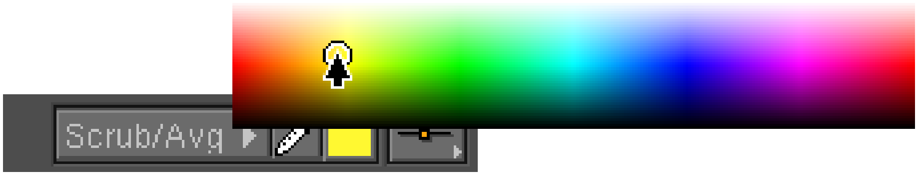
The Color Picker tools located in the top row can be used to select a color from a popup spectrum bar, to sample a color in an image, or to pick a color stored in any other color swatch in RAYZ.

There is also an animation menu that can be used to animate the color values over time, as described in "Animation Menu Options" in chapter 7, p. 100.

Color Swatch

The Color Swatch shows the currently specified color (the default is white; that is, the maximum value for the specified bit depth, such as 255 for 8-bit color).

You can also use the swatch to specify a new color: right-hold on the swatch to access a popup color spectrum, drag across the spectrum to the color you want use, and release the mouse button.



14.6 One way to specify a color is to select it from the popup spectrum strip accessed by right-holding on the Color Swatch.

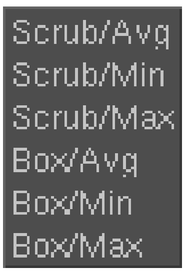
Eyedropper

Use the eyedropper tool to fill the Color node with the value of a specific image pixel or of another RAYZ color swatch. Color swatches are available in any other node that uses color parameters and in the Image Viewer's Color Picker.

The eyedropper tool samples both the RGB values and the alpha channel value, if any.

Click the eyedropper tool and the cursor will change to an eyedropper icon. Move the eyedropper cursor over the pixel you want to pick and click, scrub or drag a bounding box, depending on the sample method selected from the associated menu.

Sample Method Menu



14.7 Options available in the Sample Method menu. The option selected determines how image pixels are sampled and whether to use the average, minimum, or maximum value of the sampled pixels.

Use this menu to select the sample method to use when picking pixels in an image. The default method lets you click an individual pixel or scrub across an area of pixels and gives you the average value of the sampled pixels. However, you can also choose Scrub/Min or Scrub/Max to get the minimum or maximum value of the sampled pixels.

The other method lets you drag a bounding box around the pixels to be sampled. As with the scrub method, you can select whether to use the average, minimum, or maximum value of the sampled area.

Colorspace Selectors

Select the colorspace model you want to use to specify the color values: RGB, HSV, or Constant Luminance. The interactive display and corresponding numeric fields are linked—changing one updates the other accordingly. The interactive display is dynamic; that is, the nature of the display changes to represent the selected color model.

RGB

This is the default colorspace setting, as shown in *Fig. 14.5*. You can enter specific RGB values into the fields, or you can use the interactive display to select a color.

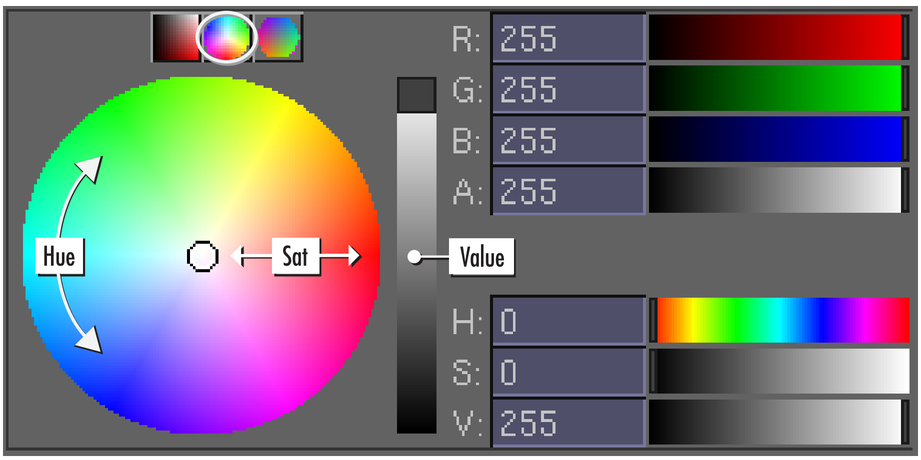
In the interactive display, start by dragging the vertical color slider to select a color from the strip. The range for that color will appear in the box to the left of it. Then click a specific point in the range.

The RGB parameters are set in a range of 0–max, where max represents the highest value at the selected bit depth (0–65535 in 16-bit, e.g.). The range updates automatically to reflect the current selection in the Bit Depth menu.

HSV

Select HSV to display a hue-based color model. Use the color wheel to select a specific hue at a specific saturation. The closer to the center, the higher the saturation. Drag the vertical slider to the right of the color wheel to adjust the Value parameter.

14.8 HSV color tools.

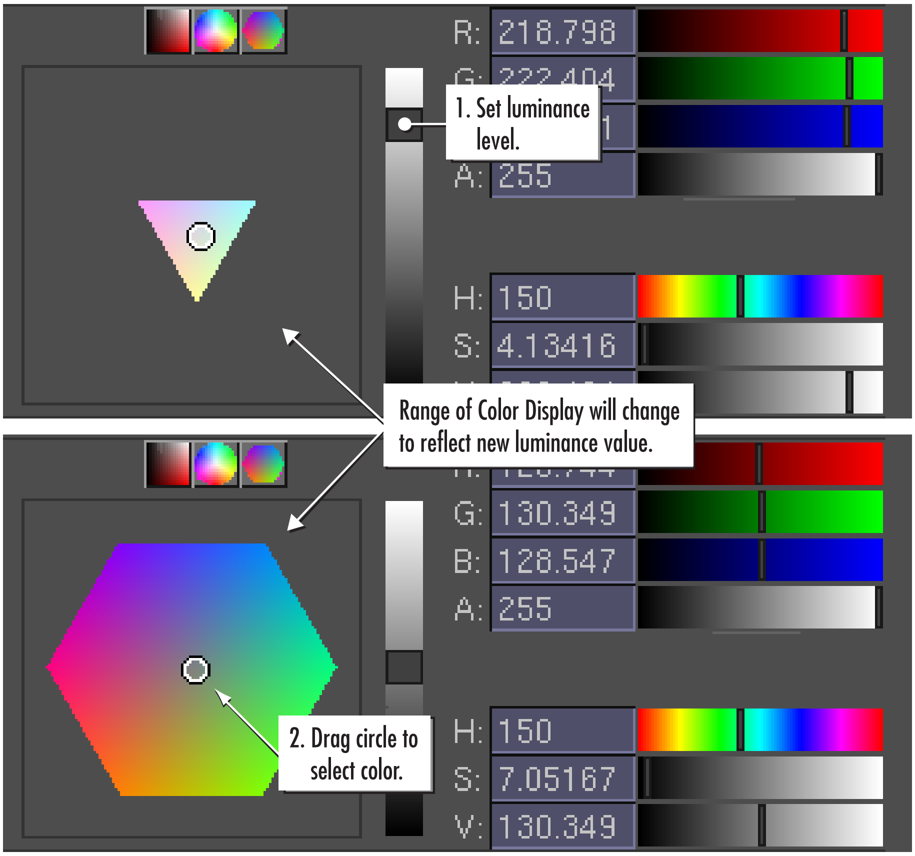


The Hue parameter range is from 0 to 360, representing a color wheel. The Saturation and Value parameters are set in a range of 0–1. (For more information about working in HSV space, refer to the description of the "Hue Adjust Node" in chapter 16, p. 265).

Constant Luminance

Select this option when you need to restrict the color selection range to a specific luminance value.

Use the vertical slider in the center to select the luminance level, and the available color range at that luminance will update accordingly. Then select a color from the range by clicking on it.

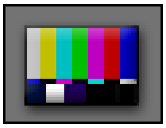


14.9 Constant Luminance color tools.

NOTE:

The luminance value used is computed, not perceptual.

COLOR BARS NODE



The Color Bars node generates a test pattern of color bars for broadcast applications.

Like other source nodes in RAYZ, the Color Bars node also provides parameters to set the bit depth, channels, frame range, and backing color of the output. Refer to "Frame Attribute Parameters Common to Source Nodes" (p. 162) if you need more information about setting these parameters.

NOTE:

Channel selection in the Color Bars node is limited to RGB or RGBA output. And unlike the other source nodes, the size is limited to the choices in the Standard menu, which is described below.

STANDARD

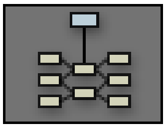
The size of the output frame is limited to the size of the CCIR broadcast standard you choose in the Standard menu:

- NTSC is 720 x 486 (the default)
- PAL is 720 x 576

INTENSITY

The Intensity menu enables you to change the intensity from the default level of 75 percent to 100 percent.

File Group Node



The File Group node is used to import a RAYZ file that has been configured as a customized group. This way, a particular procedure or effect can be distributed to multiple compositors with consistent results. In addition, if any changes are made to the master group file, all File Group nodes pointing to that file can be updated to match the latest version.

The File Group node has no inputs or outputs when it is created. Once a RAYZ file has been imported into the node, it will have as many inputs and outputs as were specified in the imported file.

File Group Parameters

When a new File Group node is created, the only parameters in the Node Panel are Edit Group Inputs/Outputs, Reload Project, and File.

Once you use the File parameter to load a RAYZ project file, however, the Node Panel will contain any parameters that have been exported to it, which may be from any node in the file.

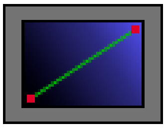
File

The File parameter must be specified before you do anything else—the node is just an empty shell until you do. You can type the full pathname of the project file you want to access into the File field, or you can click the file folder button to bring up a dialog box in which you can navigate your directory structure to locate the file.

Reload Project

Click this button whenever a change has been made to the master file referenced by the File parameter to force RAYZ to update the contents of the File Group node.

GRADIENT NODE



The Gradient node enables you to create a color gradient using the values you specify. Most commonly, the Gradient node is used to create matte or mask data with smoothly graduated changes in alpha values.

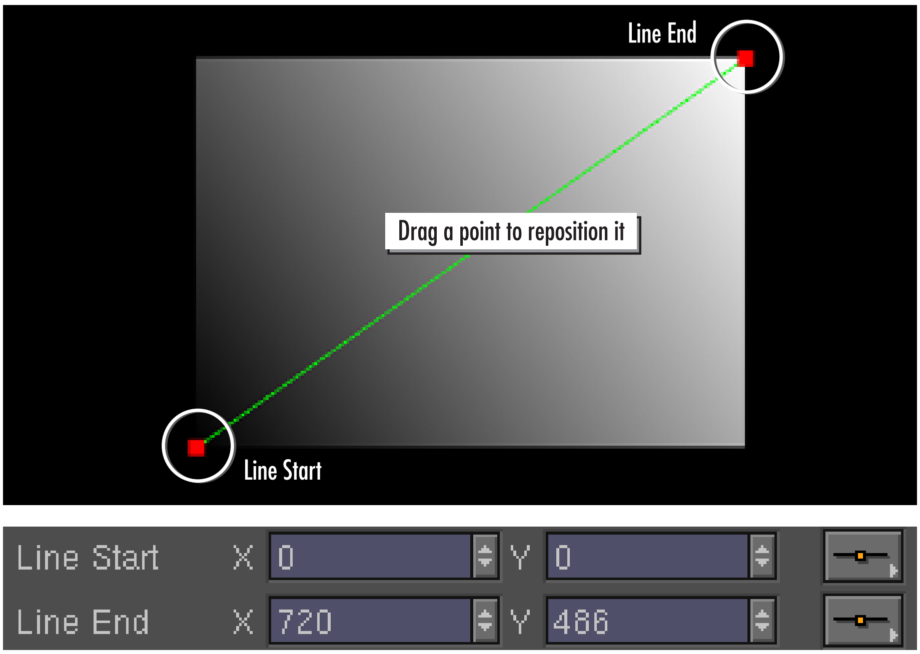
The Gradient Node Panel provides controls for creating the gradient and for specifying the output frame attributes, including size, bit depth, channels, frame range, and backing color. The frame parameters are described in "Frame Attribute Parameters Common to Source Nodes" (p. 162). The parameters specific to creating a gradient are described next.

NOTE:

To create a circular gradient, use the "Circle Ramp Node" (p. 166).

USING THE GRADIENT PARAMETERS

By default, the Gradient node creates a grayscale linear ramp from transparent to opaque. However, you can adjust the direction and length of the gradient vector by using the interactive vector line in the Image Viewer or by entering specific *x,y* coordinates into the Line Start and Line End parameters in the Node Panel. These controls are interdependent; when you change the line position in the Image Viewer, the parameters in the Node Panel update automatically, and vice versa.

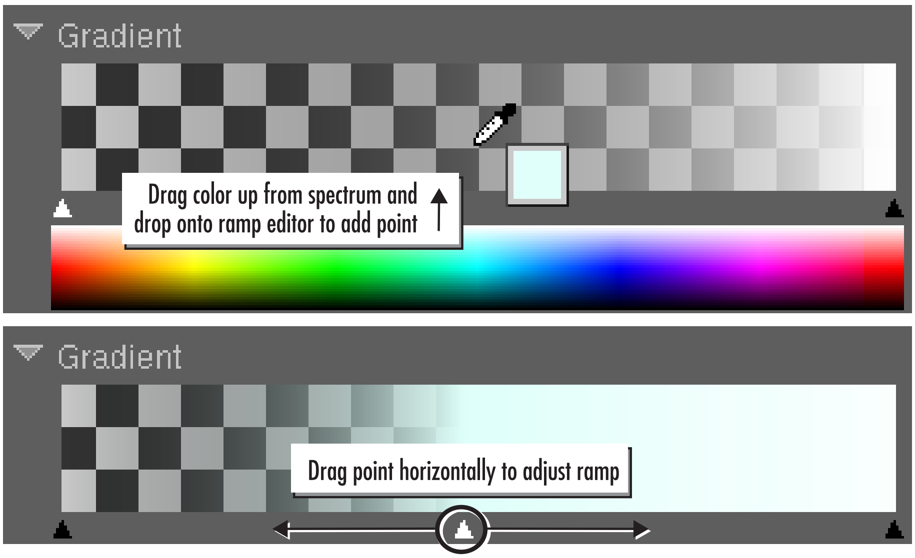


14.10 The Gradient overlay in the Image Viewer is shown on top; below it are the corresponding parameters in the Gradient Node Panel.

You can also use the controls in the Gradient Node Panel to adjust the opacity and color of the ramp, and you can add adjustment points to further manipulate the color frequency distribution.

Adding Adjustment Points to the Ramp

To add points and adjust their color and opacity, use the Ramp Editor in the Node Panel. To add a point, select a color from the Spectrum strip (located directly below the Ramp Editor) and drag it into the Ramp Editor at the desired location along the ramp. When you release the mouse button over the ramp, a new point is created, which is indicated by a triangle pointer at the bottom of the Ramp Editor display.



14.11 The top image shows how to use the spectrum strip to create a new point in the Ramp Editor; the bottom image shows how to adjust a point.

Fine-Tuning the Ramp

You can adjust the position, color, and opacity of the currently selected point. Click the triangle pointer for any point to select it:

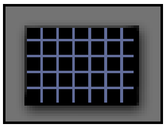
- To adjust its position, simply drag the pointer horizontally.
- To adjust its color, use the Color parameters, located directly under the spectrum bar. These controls are described in "Using the Color Parameters" on p. 168.
- To adjust its opacity, use the Alpha parameter in the Color parameters.

TIP:

To more easily manipulate closely spaced points, zoom in on the Ramp Editor. Position the cursor over the Ramp Editor and use the "+" and "-" hotkeys to zoom in and out.

Alternatively, you can use the maximize hotkey (F11; or a quick tap on the Space bar) to temporarily fill the interface with the Gradient Node Panel. Use the hotkey again to return to normal layout.

GRID NODE



The Grid node creates a grid pattern in the colors and dimensions you specify.

GRID PARAMETERS

You can specify frame size, bit depth, channels, and range of the output as described in the section on "Frame Attribute Parameters Common to Source Nodes" (p. 162).

The other parameters in the Grid Node Panel, which are described below, are used to set the width and spacing of the grid (Line Width and Line Gap), as well as the color of the grid lines and the background (Line Color and Background Color).

The Line Color and Background Color parameters use the standard RAYZ color controls, which are described in the section on "Using the Color Parameters" on p. 168.

LINE COLOR

Use the Line Color parameter controls to specify the color of the grid lines. The line color is set to white by default, including the Alpha channel, assuming you have specified RGBA output in the Channels menu.

BACKGROUND COLOR

Use the Background Color parameter controls to specify the color of the background behind the grid lines. The background color is set to black by default, including the Alpha channel, assuming you have specified RGBA output in the Channels menu.

If you want the entire grid pattern to be opaque, remember to set the Alpha channel value in the Background Color parameters to maximum value.

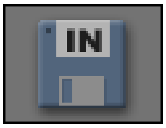
LINE WIDTH

Use the Line Width parameter to specify the width of the grid lines, in pixels. The default line width is 5 pixels.

LINE GAP

Use the Line Gap parameter to specify the size of the gap between grid lines, in pixels. The default gap size if 50 pixels.

IMAGE IN NODE



The Image In node enables you to load an image sequence from your local hard disk or from a remote directory on a shared volume. Most RAYZ networks begin with one or more Image In nodes.

TIP:

The Import Footage (Ctrl-i) command is a great shortcut for importing images. It opens the same file chooser that you access from the Image In Node Panel, and when you select a sequence it creates the Image In node for you automatically with that sequence specified in the File parameter.

You can select as many sequences, and create as many new Image In nodes as you want, all without ever closing the dialog. For more information, refer to *Chapter 10: Importing Images* (p. 123).

USING IMAGE IN

You specify the image sequence to import by using either of the following methods:

- Type the complete pathname of the sequence directly in the File field.
- Click the folder button at the right end of the File field to display a Choose File dialog box and use it to navigate your file system until you find the sequence you want to load.

Once the imagery has been loaded into RAYZ, you can adjust various parameters in the Image In Node Panel if necessary. As explained in the following sections of this node description, you can

- add an alpha channel,
- adjust the frame range,
- flip the imagery vertically,
- designate the imagery as having a non-standard pixel ratio or gamma,
- modify the conversion of certain file formats, and
- designate proxies.

ABOUT FILE FORMATS

You can import virtually any image format into RAYZ. For a complete list, see *Appendix B: Image File Formats Supported by RAYZ* (p. 441).

For some image formats you may want to take advantage of certain conversion options when the files are imported. RAYZ automatically provides the parameters for such options when the relevant file type is imported.

You can convert the bit depth and distribution type (nonlinear to linear) of the image data, as described in "Conversion Parameters" on p. 181.

NOTE:

RAYZ automatically converts Cineon log files to 16-bit linear. However, you can change this conversion using the appropriate parameters in the Conversion group.

You can also specify the true pixel ratio for "squeezed" anamorphic imagery, and override the default setting for premultiplication, as described in the section on the "Image Description Parameters" (p. 186).

ABOUT PROXIES

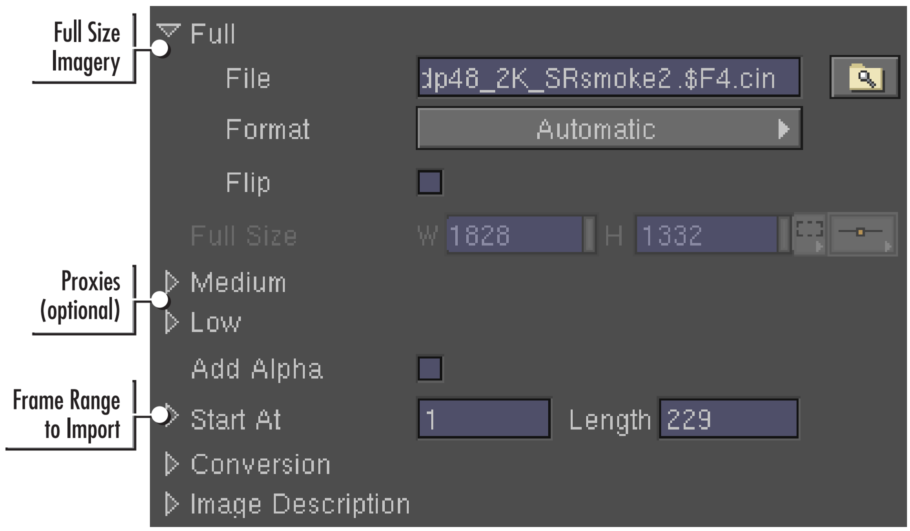
A single Image In node is used to specify both the full size version of an image sequence and any proxy files that are available for that sequence.

If proxy files have been specified in the Image In node, RAYZ will use them whenever Medium or Low resolution is selected from the Size menu in the Image Viewer. If proxies have not been specified, RAYZ will scale down the full size imagery and send the scaled-down image data through the network. The parameters used to specify proxies are described in the section on "Proxy Parameters" (p. 181).

FILE PARAMETERS

The Image In node parameters described in this section are used to specify full size and proxy images to import, to re-sequence frames, and to add an alpha channel. However, the node also offers parameters to control conversion, specify certain image characteristics, and set format-specific options when applicable. These parameters are described in the sections on "Conversion Parameters" (p. 181), "Image Description Parameters" (p. 186), and "Format-Specific Import Options" (p. 187).

14.12 Image In parameters in the Node Panel.



Full Parameter Group

The Full parameter group is used to specify the full size imagery to be loaded into the node and includes the File, Format, and Flip parameters.

File

This parameter specifies the location of the image files to be imported. You can type the complete pathname into the field, or you can click the folder button associated with the Full Path field to access a File dialog box.

The File dialog enables you to navigate through directory hierarchies and examine lists of files to find a file or sequence to import. Once you locate it, click on its name in the file list to select it and then click the Accept button. (Alternatively, you can double-click on its name in the file list.) The dialog box will close and the selected images will be loaded into RAYZ.

The File field and the other Image In Node Panel parameters will update with information specific to the imagery that has been loaded.

NOTE:

You will frequently encounter the "$F" variable in image filenames, as in "filename.$F4.cin." The "$F" indicates that the file references a sequence of images, and the "4" indicates that each frame file includes a four-digit frame number padded with zeros, as in "filename.0001.cin." The ".cin" is the file format extension that identifies the frames as Cineon format.

Format

You may never need to use this menu. The Format menu lists the file formats that can be imported by RAYZ. The default choice is "Automatic," which indicates that RAYZ will examine the image files to determine their format. If it cannot determine the file type for any reason, an error message will appear. In such a case, use the Format menu to tell RAYZ the file type of the image you are attempting to import.

Flip

Certain file formats invert the orientation of images. Click the Flip checkbox to flip the imagery vertically. (This box can be checked or unchecked at any time to change the orientation of the image.)

Full Size

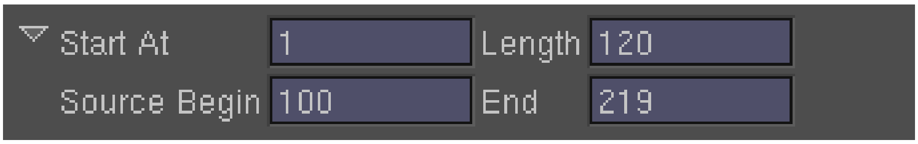
The Full Size parameter is grayed out (unavailable) unless you import proxy images without specifying a full size image source. For more information, refer to the "Proxy Parameters" (p. 181) section of this node description.

Add Alpha

Check this box to add an alpha channel to the imagery. The alpha is a uniform field set to full opacity (white). To create an alpha using any other criteria, connect the Image In node to an appropriate node, such as Channel Swap, Chromakey, Roto, or an Ultimatte node.

Frame Range to Import

The frame range parameters in the Image In Node Panel are used to specify the frame number in RAYZ at which the sequence should start and to specify the range of frames to import. They are set by default to values that represent the sequence being imported, however, they can be changed by typing new values in the fields.



14.13 You can expand the frame range group to access the Source Begin and End parameters.

The **Start At** parameter is used to specify at which frame number in the shot the imported sequence should start (the default is frame 1). The **Length** parameter specifies the total number of frames.

You can expand the parameter group to access the **Source Begin** and **End** parameters, which are used to import a subset of the total frame range available on disk.

The fields are interdependent; for example, if you change the End value the Length value will update accordingly.

When to Use the Clip Editor

The frame range parameters in Image In are the fastest way to make several of the most common adjustments to a sequence, such as changing the start-frame number. To do more extensive re-sequencing, however, use the Clip Editor, which displays each clip as a film strip in an interactive timeline. This is especially useful when you want to

- see each frame represented as a thumbnail image, or
- perform more complex edits.

NOTE:

The Clip Editor settings override the parameter settings in the Node Panel. This means that once you edit a sequence in the Clip Editor, the frame range parameters in the Node Panel are deactivated.

For more information, see "Using the Clip Editor" in chapter 12, p. 136.

PROXY PARAMETERS

The Medium and Low proxy parameters are used only when you want to specify lower resolution files to use in place of the full resolution imagery. You can specify Medium proxies, Low proxies, or both.

The Medium and Low parameter groups can be expanded to reveal the same set of parameters available for the full size input. These parameters work exactly the same way to specify the file path and flip the input, as described previously in the section on the "Full Parameter Group" on p. 179.

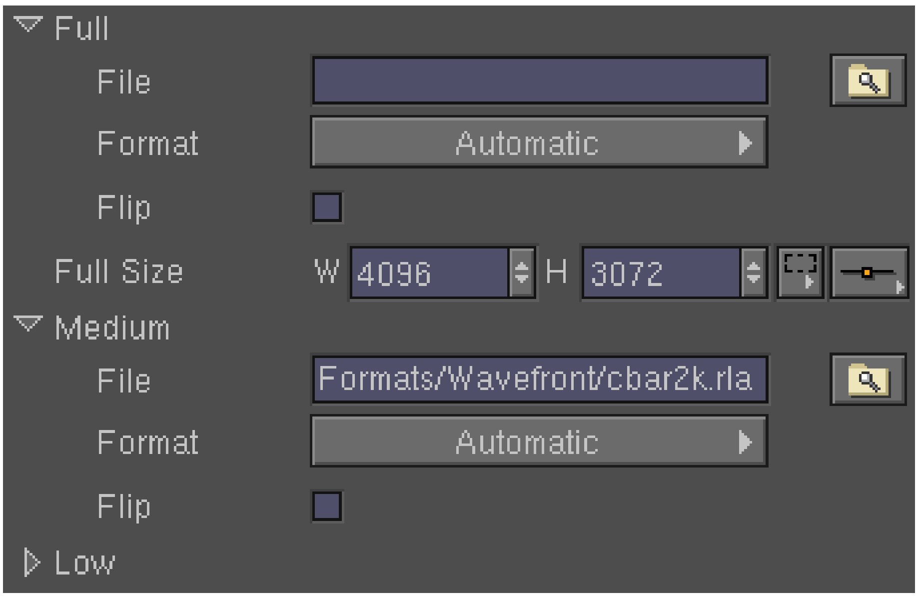
NOTE:

The Add Alpha and frame range parameters in the Image In Node Panel apply to all imagery specified in the node, proxies and full size.

SPECIFYING FULL SIZE

You can import proxy files and start building a shot even when the full size imagery is not yet available. All you need do is specify what resolution the full size imagery will be in the Full Size parameter, which becomes active whenever you use the proxy parameters to import imagery without first specifying an image sequence in the Full parameter group.

You can type width and height values into the entry fields or use the menu to select a resolution from the list.

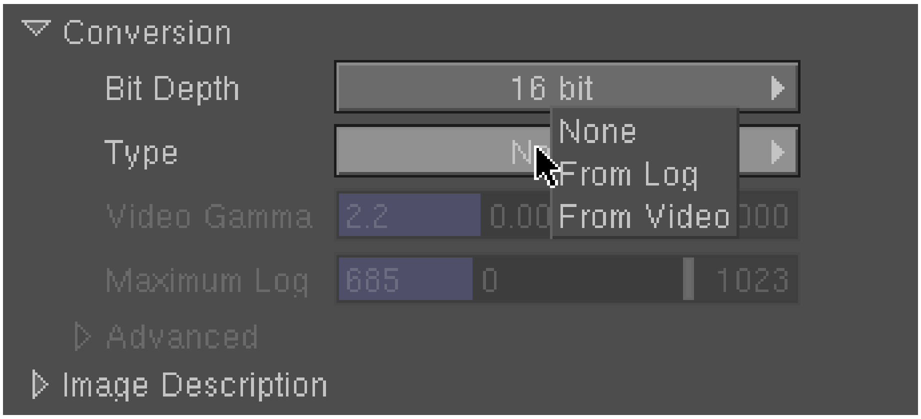


14.14 This example shows 2K proxy files imported into the Medium File parameter and the Full Size parameter set to 4K resolution. Note that full size image files have not actually been specified.

CONVERSION PARAMETERS

The Conversion parameters enable you to perform linear-to-linear conversions using the Bit Depth menu or nonlinear-to-linear conversions using the Type and Bit Depth menus. Based on the current selection in the Type menu, additional parameters may become active.

14.15 Expand the Conversion group to access the specific parameters relevant to the imported image format.



The parameters in the Conversion group default to neutral settings, which do not convert the imagery, with the exception of Cineon 10-bit log files, as explained next.

About Cineon File Conversion

By default, Cineon log files (Fido or DPX) are converted to 16-bit linear colorspace. However, you can opt instead to convert the imagery to floating point or 8-bit linear (using the Bit Depth menu).

You also have the option not to convert the log data at all (by selecting None from the Type menu), as when you want to perform color corrections in log space. You can always convert the color corrected log image to linear later by using a "Log To Lin Node" (ch. 20, p. 407) downstream in the node network.

NOTE:

By default, log image data is displayed in the Image Viewer using Raw Log display conversion. If you prefer Cineview emulation, select it in the Viewer's "Display Conversion" (ch. 6, p. 72) parameters.

When converting a Cineon image from log to linear, you can also adjust the default settings used to remap the color values. The easiest way is probably to change the "Maximum Log" (p. 183) parameter value, which automatically changes the relevant log conversion parameters accordingly. For even more control, however, you can expand the "Advanced Cineon Parameters" (p. 184) group and adjust any of the individual log conversion parameters.

Bit Depth Menu

You can convert imagery to 8-bit, 16-bit, or floating point precision per channel by selecting a bit depth from the menu. For linear image data, the Bit Depth menu defaults to the native resolution of the imported files.

TYPE MENU

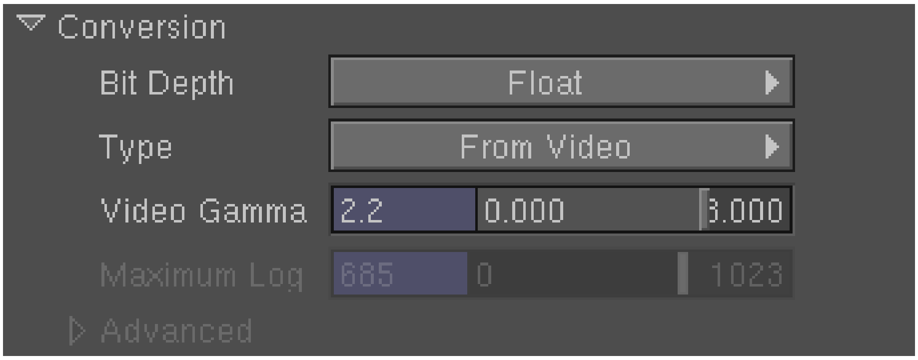
The Type menu is used to convert nonlinear data distributions to linear:

- Cineon log files (by using the **From Log** menu item)
- Video with a nonlinear gamma correction (by using the **From Video** menu item)

With the exception of Cineon Fido and YUV files, the Type menu defaults to None (no conversion). Cineon files default to "From Log" and YUV files default to "From Video."

VIDEO GAMMA

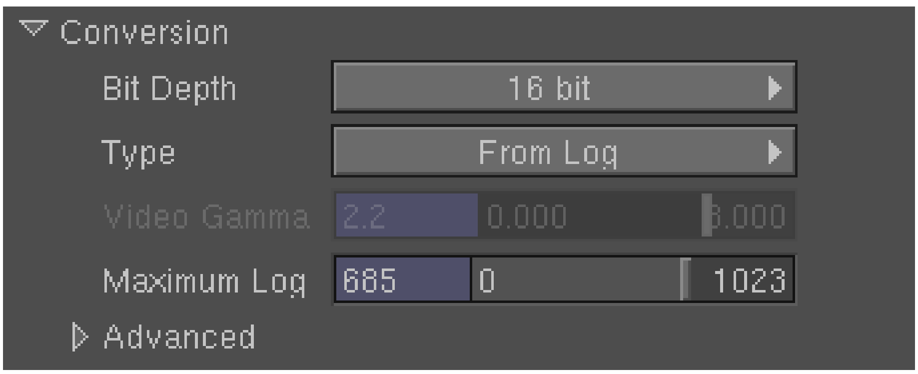
This parameter becomes active when From Video is selected in the Type menu. It is used to specify that a nonlinear gamma correction has been applied to the imagery being imported. RAYZ will then convert the color channels of the image from the specified gamma to linear gamma (1.0). The Alpha channel, if there is one, is not affected.



14.16 The Video Gamma parameter defaults to 2.2, to match the gamma value commonly encoded in YUV imagery.

MAXIMUM LOG

This parameter becomes active when From Log is selected in the Type menu. RAYZ uses the Maximum Log value to compute the linear white value used in the conversion. (Conversely, if you modify the log parameters in the Advanced group, the Maximum Log parameter is updated accordingly.)



14.17 When From Log is selected from the Type menu, the Maximum Log parameter and the Advanced parameters become active.

The default value for Maximum Log is 685, however you can change it to reflect the actual maximum value in the image being converted. If the highlights in the image seem "blown out," for example, raise the Maximum Log value until they look right.

When you raise the Maximum Log value, RAYZ recalculates the optimal Linear 90% White value, lowering it from its default of 65535 to create headroom in linear space to accommodate specular highlights and other extremely bright image data captured in the original photography.

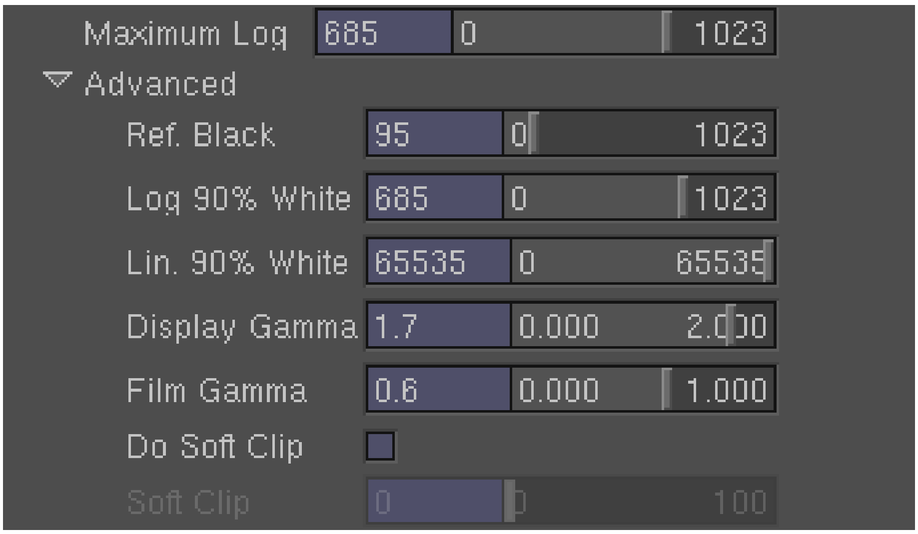
Be aware, however, that the lower the Linear White value, the smaller the range becomes into which the majority of the image data is remapped. Another option is to convert the log data to linear float, which will not clip the highs.

NOTE:

To examine the log image in the Image Viewer and determine the actual maximum value, you can set the Type menu to None temporarily to output the unconverted log data. Otherwise, you will be viewing the converted image data.

ADVANCED CINEON PARAMETERS

The Advanced parameter group provides complete control over the Cineon log conversion process. The Advanced parameters become available when From Log is selected in the Conversion Type menu.



14.18 Expand the Advanced parameter group to access the full set of Cineon log conversion variables.

By default, Image In remaps Cineon log files to 16-bit linear by stretching the log data in a constant slope to fill the 16-bit space, using standard values of 685 for log 90% white and 65535 for linear 90% white.

NOTE:

The default values for Cineon conversion are based on specifications published by Kodak Motion Picture & Television Imaging and Cinesite Digital Film Center in "Grayscale Transformations" (1993) and "Conversion of 10-bit Log Film Data to 8-bit Linear or Video Data" (1995).

These documents can be downloaded in Acrobat PDF format, which is suitable for printing, from Silicon Grail at ftp://ftp.sgrail.com/pub/reference/cineon. They are also posted on the Technical Documents page of the Cinesite Hollywood website at http://www.cinesite.com/la/scanrec/techdocs.html.

Depending on the nature of the specific Cineon files you are importing, you may want to adjust the Advanced parameters:

Reference Black: This parameter specifies the value used for reference black in the conversion operation. The default value is 95, which represents Dmin (the minimum printing density, or blackest black that can be recorded, about equivalent to the 1% black card).

Log 90% White: This parameter specifies the value used for log 90% white in the conversion operation. The default is 685, which represents the code value of the 90% white card for a normally exposed film negative.

Linear 90% White: This parameter specifies the value in linear space to which the log 90% white value will be mapped in the conversion operation. The default is 65535 (in 16-bit), which will clip any values above 685 in the original log file. When converting to linear float, however, values are not clipped at 1.

Display Gamma: This parameter specifies the value used for display gamma in the conversion operation. The default value is 1.7.

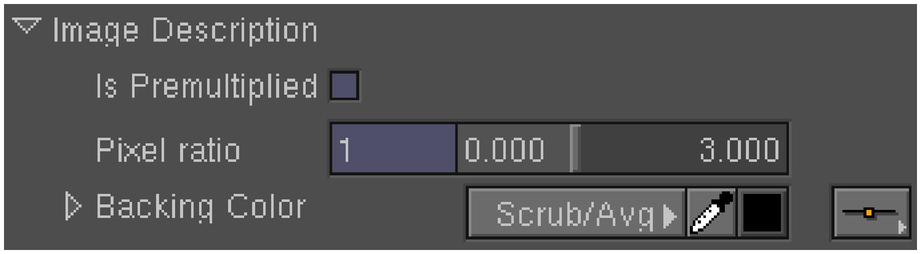
Film Gamma: This parameter specifies the value used for film gamma in the conversion operation. The default value is 0.6.

Softclip: Softclip can be used, if necessary, to reduce the effects of harsh clamping at the high end when remapping log data. Check the Softclip box to enable this option and set a positive softclip value in the associated parameter field.

The Softclip parameter can be adjusted in the range of 0–100. This value is subtracted from the Log 90% White value to create a breakpoint below peak white. The slope of the distribution curve above the breakpoint becomes nonlinear to remap the highs more gradually.

IMAGE DESCRIPTION PARAMETERS

This parameter group is used to specify various characteristics of the imagery being imported, including the premultiplication status, pixel ratio, and backing color of the imagery. RAYZ can then use this information in other nodes to adjust default parameters or Image Viewer display settings.



14.19 Expand the Image Description group to access parameters that identify whether an RGBA image is premultiplied as well as whether an image has a non-square pixel ratio.

IS PREMULTIPLIED

The Is Premultiplied checkbox tells RAYZ whether or not the imagery has been premultiplied. RAYZ sets this parameter automatically based on the number of channels in the image being imported: an RGBA image is assumed to be premultiplied and an RGB image, unpremultiplied. You can use this checkbox to override the setting if necessary for a particular image.

The premultiplication status of an image is important when an image is composited, because premultiplying the RGB channels of an input by the alpha channel is an initial step in many compositing operations. See "About Premultiplication" in chapter 18 (p. 328) for more information.

PIXEL RATIO

This parameter specifies the pixel ratio of the imagery. The pixel ratio (as distinct from the aspect ratio) refers to the ratio between the number of pixels used vertically and horizontally, per equal unit of measure, to encode the image data.

Most digital imagery has a square pixel spacing ratio (1:1); that is, the same number of pixels are used per inch both vertically and horizontally, regardless of how many inches high or wide a particular image may be.

Certain image formats, however, are encoded with a non-square pixel spacing ratio, typically YUV video (0.9:1) or "squeezed" anamorphic film footage (2:1). RAYZ uses the Pixel Ratio parameter value to display such imagery properly in the Image Viewer as well as to correctly perform rotations, blurs, and other image operations that may use image size data in the computation.

RAYZ sets the Pixel Ratio parameter automatically, based on the imagery being imported, but you can override the setting if RAYZ makes the wrong guess. The parameter value represents the variance, if any: for CG images and most film footage, the default value is 1; for video, it is 0.9; and for anamorphic, it is 2.

Backing Color

You will probably never want to change the default value (0) of this parameter. Common to all source nodes, it is described at the beginning of this chapter in “Backing Color Parameters” on p. 164.

Format-Specific Import Options

Some image file formats encode data in ways that require RAYZ to ask for additional information to interpret them properly. For example, some formats allow additional channels of data to be encoded at a different bit depth from the RGB image data in the same file.

When these files are imported into RAYZ, format-specific parameters become available as described below. Expand the parameter group to access the controls.

14.20 Format-specific parameters only become active when an image of the corresponding type is imported—a Photoshop image in this example.

Adobe Photoshop

The Adobe Photoshop parameters become available when importing Photoshop (.psd) images, which may have been saved with multiple layers.

Read Flattened Checkbox: This parameter, which is checked by default, is equivalent to the “Flatten Image” command in Photoshop. It creates a single-layer image composed of all the layers in the Photoshop file. If you uncheck the box, RAYZ will read in only the layer specified in the associated Layer parameter.

Layer Parameter: This parameter becomes active when “Read Flattened” is unchecked. It enables you to specify which layer to read into RAYZ. Layers are specified by number, starting with 0 for the background layer.

Maya Formats

The Maya parameter becomes available when importing Maya IFF imagery, which may include a channel of depth data, because Maya image files store RGBA data as 8- or 16-bits per channel and z-depth (ZBUF channel) data as floating point.

Read Data Menu: Use this menu to select whether to import the RGB(A) image or the z-depth channel into the current Image In node.

TIP:

To access all channels of a Maya image (RGBZ or RGBAZ), select the same file sequence in two different Image In nodes and specify RGBA in the Read Data menu of one node, and ZBUF in the other.

WAVEFRONT RLA/RLB

The Wavefront parameter becomes available when RLA or RLB files are imported, which may include specialized channels of data such as z-depth that are encoded at a different bit depth from the RGBA image data.

READ DATA MENU: Use this menu to select which data to import into the current Image In node, the RGB(A) image, Z-Buffer channel, Effects channel, or Object channel.

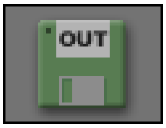
YUV

YUV files are converted from YUV colorspace when imported into RAYZ. By default, they are converted to floating point data to avoid quantization.

This is because YUV data is encoded in a way that can result in values greater than 1 and less than 0 when converted to RGB, so the increased precision is necessary to avoid losing color information that would be clamped otherwise.

REPRESENT AS MENU: Use this menu if you need to change the bit depth from floating point to 8-bit or 16-bit per channel data.

Image Out Node



The Image Out node is used to render the imagery flowing into it. Whenever you choose, the node will write image files to disk in the format and location indicated in the Image Out Node Panel. An Image Out node can be used at any point in a network, and you can add as many Image Out nodes to a network as you need.

Simply creating an Image Out node does not initiate rendering; however, you can start the render at any time by pressing the Render button at the top of the Image Out Node Panel. Alternatively, you can execute a render in the Render Control panel, which automatically lists all existing Image Out nodes in the project file and enables you to coordinate rendering of multiple sequences.

Each listing in the Render Control panel duplicates all the parameters in the Image Out Node Panel; in fact, the two sets of parameters are actually the same data, displayed in two different places. This means that you can make a change to a parameter in the Render Control panel and it will be reflected automatically in the Node Panel of the corresponding Image Out node, and vice versa.

See also *Chapter 11: Rendering Images* (p. 129) for more information.

Image Out Parameters

The Image Out parameters are used to specify the filename and directory location, the file format, and the frame range to use for each render. In addition, you may be able to specify certain conversion or compression options for the selected file format.

TIP:

Take advantage of the ability to save node presets to create Image Out presets for the combinations of parameter settings (such as a specific file path, format, and compression level) that you use frequently. Then you can select the preset from any Image Out node and the parameters will update accordingly. See also "Presets Menu" in chapter 7 (p. 88).

Render Controls

The **Render button** at the top of the Node Panel is used to actually initiate a render. When you press the Render button, the node data is written to disk at the size and quality specified in the Scale Factor and Quality menus, and using the criteria specified in the other Image Out parameters.

SCALE FACTOR

The Scale Factor menu specifies whether the images should be rendered at full size or a fraction thereof. Full size is selected by default, however, you can choose to render Medium or Low size images instead.

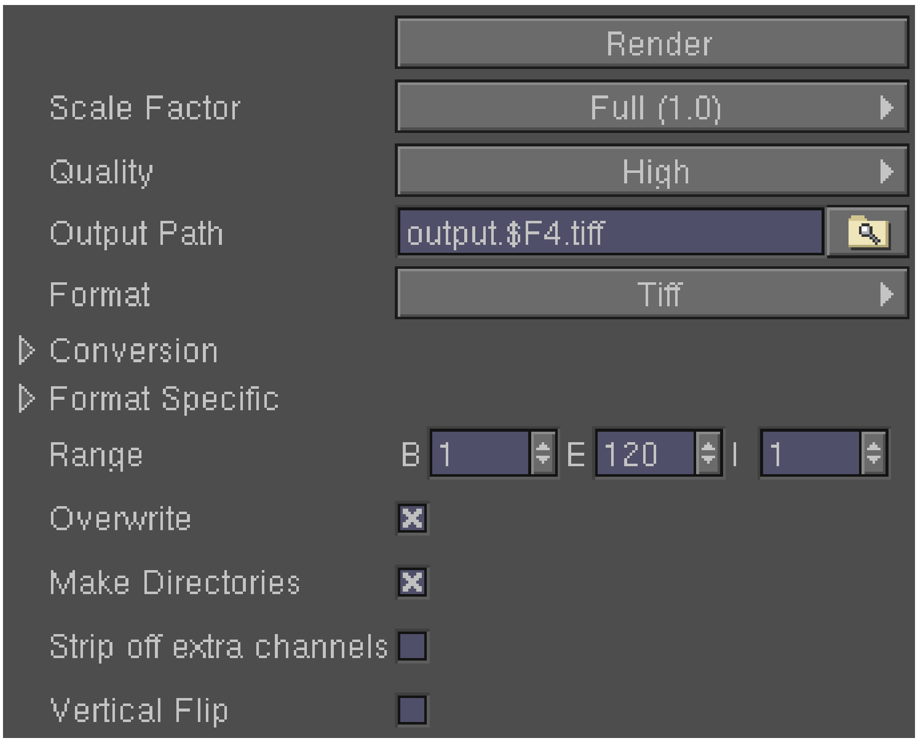
By default Medium is 50 percent and Low is 25 percent of full size; however you can change these percentages in Edit > Project Settings > Settings > Scale Factors.

QUALITY

The Quality menu is used to specify a lower image quality render if necessary to reduce rendering time for preview comps. Quality is set to High by default, which renders the images at full quality.

The Medium and Low settings affect any filtering that has been specified in upstream nodes to reduce aliasing and related image artifacts, whether such filtering is user-set in a node parameter or is intrinsic to a node's operation:

- Medium sets filtering to bilinear interpolation for any upstream nodes that use such filtering, such as nodes that perform spatial transformations. It also reduces the level of anti-aliasing done on roto shapes.
- Low turns off all such filtering (point sampling only), and no anti-aliasing is done on roto shapes.



14.21 Parameters in the Image Out Node Panel. When you press the Render button at the top, RAYZ writes image files to disk in the location and format specified in the other parameters.

Output Path

This parameter specifies the filename and location of the rendered image frames. You can type the complete pathname into the field or you can click the folder button associated with the field to access the file dialog and navigate the directory structure to the desired location:

- If you enter only the filename, without specifying the directory path, the files will be written to the directory from which RAYZ was launched.
- The file extension, such as ".cin" for Cineon format files, is appended to the filename automatically when you select a new format from the Format menu.

Sequentially Numbering Frame Files

Use the **$F variable** to sequentially number each frame file. To pad the frame numbers with leading zeros, append a number to $F to specify the number of digits. For example, using "comp.$F4.cin" as the filename would number the frames sequentially and pad each number with leading zeros as necessary to create a four-digit number: the first frame would be "comp.0001.cin," the second, "comp.0002.cin," and so on.

NOTE:

If you output a sequence as a QuickTime or FRP file, however, do not use "$F" in the filename since a single movie file of the sequence is being created instead of a series of frame files.

$JOB Directory

You may also want to take advantage of the JOB global variable, which can be used in the Output Path field as shorthand for the directory into which the images should rendered.

For example, if you render a 200-frame sequence to disk and define the output path as "$JOB/my_file.$F4.tiff," RAYZ will create 200 files named "my_file.0001.tiff" through "my_file.0200.tiff" in the directory specified for $JOB.

The JOB global defaults to the directory from which you started RAYZ. You can redefine it to a different directory path in Edit > Project Settings > Globals. For more information about defining global variables, see "Globals" in chapter 13, p. 157.

Format

The Format menu specifies the image file format of the rendered imagery. Tiff format is the default; however, you can select any format from the menu. When you select another format, RAYZ will replace the .tiff extension in the File field with the appropriate extension for the chosen file type.

The same formats that can be imported into RAYZ can be specified. For a complete list, refer to *Appendix B: Image File Formats Supported by RAYZ* (p. 441).

Be sure to review the sections below describing the Conversion parameters, which are used when the node imagery is linear and the render format is nonlinear, and the Format Specific parameters, which are used to set compression and other options when the corresponding render format is selected.

NOTE:

To write log imagery to disk in any format other than Cineon Fido or DPX, convert it to linear first using the Log To Lin node. To render linear image data in Cineon log format, however, it is not necessary to use a conversion node; linear-to-log conversion can be specified in Image Out.

Conversion

The Conversion parameters are used, when necessary, to specify how the node imagery will be converted to the file format selected in the Format menu. These parameters correspond to those governing the conversion of imported imagery in the Image In node.

- **Linear to linear:** To render linear imagery at a different bit depth, select an option from the Bit Depth menu and make sure that the Type menu is set to "None."
- **Linear to log:** To render linear imagery in Cineon log format, select a Cineon format in the Format menu and the "To Log" option in the Type menu will then be selected automatically.
- **Log to log:** To render log imagery in Cineon format, select a Cineon format in the Format menu and the "None" option in the Type menu should be selected automatically.
- **Log to linear:** To render log imagery in a linear format, convert the imagery to linear first in a Log To Lin node and connect the Log To Lin node to Image Out. Then follow the guidelines above for linear-to-linear conversion.
- **Video gamma:** To add a nonlinear gamma to linear imagery destined for certain video devices, select "To Video" in the Type menu.

Bit Depth Menu

Use this menu to specify the bit depth per channel at which to render linear imagery. The menu defaults to the bit depth of the node data, so you only need to use it when you want to change the bit depth.

Type Menu

As delineated in the guidelines under "Conversion" above, the Type menu is used to convert linear imagery to nonlinear, in conjunction with the file format chosen in the Format menu. The choices are as follows:

None: Select None unless you are converting linear node data to a nonlinear format.

To Log: Select To Log to convert linear node data to Cineon 10-bit log. (You must also select Cineon Fido or DPX in the Format menu.)

The Advanced parameters for Cineon conversion will become active, which are described in detail in the section of the Image In node description on "Conversion Parameters" (p. 181).

To Video: Select To Video when you want to encode a nonlinear gamma correction in linear imagery to be rendered in a video format such as YUV. The Video Gamma parameter becomes active when this menu option is selected to enable you to specify the gamma value (the default is 2.2).

About Writing Cineon DPX Files

Unlike Cineon Fido format, which must be 3-channel, 10-bit log data, Cineon DPX files can be written that have 1, 3, or 4 channels, and they can be 8- or 16-bit linear, encoded with a non-linear video gamma, or they can be 10-bit log.

To write DPX files using the specifications you need, select the appropriate combination of conversion options from the Bit Depth and Type menus.

Format Specific

Expand this group to access parameters for controlling compression options. A parameter within the group becomes active when the corresponding file format is selected from the Format menu.

When rendering files in Adobe Photoshop, SGI, Softimage, or Targa format, you have the option to turn compression on or off (compression is on by default). When you render in JPEG, PNG, QuickTime or Tiff formats, you have additional compression options:

JPEG Format Compression: The JPEG format uses, naturally, JPEG compression. As JPEG is a lossy compression method, an additional parameter is provided to set the image quality level (the range is 0–100; the default value is 90). Image quality and compression level are inversely proportional: the lower the quality value, the higher the compression. For most images, the actual useful range is approximately 35–90 (with an image compressed to 35 being a very low quality image).

PNG Format Compression: The Portable Network Graphics format uses lossless Zip compression, with the option to set the level. Higher compression levels create smaller files but take longer to render.

QuickTime Format Compression: Uncompressed is selected by default, however you can choose Animation, YUV2, JPEG, or Motion JPEG-A from the QuickTime Compression menu.

Tiff Format Compression: In addition to turning off compression, you can choose from Zip, LZW (the default), or JPEG compression. Zip and LZW are lossless, with Zip providing the option to set the level of compression. The higher the compression, the longer the files take to render. If you choose JPEG to compress the files, you have the option of setting the quality level for this lossy compression method.

Range

The Range parameters are used to specify the frame range to render. You can specify the start and end frame and the increment. By default the Range parameters are set to the range of the input sequence, with an increment of 1.

Overwrite

If the Overwrite box is checked (which it is by default), RAYZ will overwrite any files with the same name and location as the name you assign to the output files.

NOTE:

RAYZ will write temporary files when the Overwrite option is checked, only replacing the previous versions once the render is complete. If the render fails for any reason, the old files will not be destroyed.

Make Directories

The Make Directories box, when checked (which it is by default), lets RAYZ create new directories, as needed, based on the directory path specified in each entry.

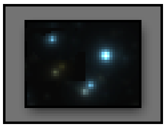
Strip Off Extra Channels

Some image file formats encode only color data; they do not support imagery with a separate alpha channel or other data. If you try to render node imagery containing more channels than the selected file format supports, RAYZ will generate an error message unless you check the Strip Off Extra Channels box. If this option is checked, however, RAYZ will ignore any extra channels and render the rest.

Vertical Flip

Certain file formats invert the orientation of images. Click the Vertical Flip checkbox to invert the imagery when writing the file to disk. This parameter is the complement to the Vertical Flip parameter for the Image In node.

Stars Node



The Stars node generates a star field based on actual astronomical data. You can set the node parameters to generate an accurate representation of the night sky that would be visible at a particular time and location, or you can simply experiment with the controls, judging the results by eye.

The data used to generate the positions, proper motions, and magnitudes of the stars is derived from the Smithsonian Astrophysical Observatory catalog. For more information about the SAO Star Catalog, refer to the following website:

http://tdc-www.harvard.edu/software/catalogs/sao.html

The frame parameters are described in "Frame Attribute Parameters Common to Source Nodes" (p. 162). The parameters specific to creating the star field are described below.

Stars Parameters

Start by using the Position parameters to specify the time and location from which the sky would be viewed and then use the Look parameters to modify the appearance of the star field.

Position Parameter Group

Use the Position parameters to specify from where and when the sky is being viewed (if an accurate representation is unnecessary or irrelevant, you can always use the default values). Start by selecting the coordinate system to use from the Coordinates menu and then set the other parameters, animating them as necessary:

- The Coordinates menu specifies which coordinate system the other applicable Position parameters modify.
- The Month and Day parameters specify the viewing date.
- The Latitude or Declination parameter works with Hour or RA to specify an area of sky in celestial coordinates.
- The Tilt, Pan, and Roll parameters change the POV from the starting point specified in the other Position parameters.

Tilt (X), Pan (Y), Roll (Z)

The Tilt, Pan, and Roll parameters enable you to animate the star field, perhaps to "fly the camera" through space or to simulate changes in the sky over time from a single vantage point.

The Tilt (move in X) parameter can be adjusted in a range of -90 to 90 degrees, while you can Pan (move in Y) and Roll (move in Z) in a range of 0 to 360 degrees.

Rotation Order Menu
This menu specifies the order in which the above transformation operations will be computed. The default order is Pan, Tilt, Roll.

Month
This parameter specifies the month of the year, from 1 (January) through 12 (December).

Day
This parameter specifies the day of the month, from 1 through 31.

Hour or RA
Right Ascension, abbreviated to RA, is the equivalent in the celestial sphere of longitude on Earth, but it is measured in hours, minutes and seconds instead of degrees.

The 0 hours line of RA marks the point where the sun crosses the celestial equator at the vernal equinox (the first moment of spring) in the northern hemisphere. Hours of RA are measured eastward from this point, up to 23 hours, 59 minutes. One minute later the cycle starts over at 0 hours.

Latitude or Declination
Declination is the celestial equivalent of latitude. It specifies the angular distance, measured in degrees, above or below the celestial equator. It ranges from 0 at the celestial equator to 90 degrees at the north pole and -90 degrees at the south pole.

About RA and Declination
RA (Right Ascension) and Declination are the celestial equivalents of longitude (the lines that run East-West) and latitude (the North-South lines), the coordinate system used to specify locations on Earth.

Declination is a straightforward projection of the Earth's latitude into space. Terrestrial longitude, however, cannot simply be projected into space because the sky appears to rotate (15 degrees per hour, as the Earth rotates on its axis) with respect to any line of longitude on Earth.

For this reason, astronomers use a measure for celestial longitude called right ascension (abbreviated RA), in which the sky is divided into 24 fifteen-degree increments, each of which is one hour of right ascension.

TIP:

You can get RA and Declination coordinates for any location on earth. Click a location on the interactive map on the following web page, which will take you to a current star chart for that earthly latitude (the star chart has the RA and Declination coordinates on it):

http://www.learnwhatsup.com/astro/charts/worldmap.shtml

Rotation Offset
This parameter creates an offset of the Y rotation value. The offset value range of 0–1 represents the percentage of one complete rotation (360 degrees).

Coordinates Menu
Choose from Earthbound - Up (the default); Earthbound - North; Astronomical; or Ecliptic (Zodiac).

Look Parameter Group
Use the Look parameters to modify the appearance of the star field. The fastest way to increase or decrease how "starry" the field looks is to adjust the Pre Gamma parameter. Related parameters include Magnitude, Over Exposure, and Softness. You can simulate lens distortions and similar effects using Focal Length, Film Size, Warp, and Pixel Ratio. The Saturation parameter controls the colorfulness of the stars.

Focal Length
Use this parameter, along with Film Size and Warp, to simulate the affect of changes in focal length or similar lens distortions. (Focal Length and Film Size are interdependent.)

Film Size
The Film Size parameter is a scale factor for the Focal Length parameter, which means that when both parameters are at their default values (35), they have no effect on the node output. Changing the Film Size value creates a zoom effect.

Warp
The Warp parameter value changes the apparent depth of the star field, effectively adding curvature to the field.

Softness
Increasing the Softness value blurs and spreads the stars. This parameter works with the Anti-Aliasing parameter.

Over Exposure
Use the Over Exposure parameter to change the average apparent magnitude values for the stars obtained from the database.

Saturation
The Saturation parameter specifies how colorful the stars will be in a range of 0–1. The default value of 1 represents full saturation.

Magnitude
This parameter specifies a magnitude threshold value for the star field. Stars with magnitudes (as determined by the SAO database) below this threshold will not be represented in the output image.

Pre Gamma

This parameter changes the average brightness and size of the stars. It increases or decreases the magnitude values obtained from the database, which means that more stars will become visible as the Pre Gamma value is increased. (The Over Exposure parameter value will then further modify this characteristic.)

Pre Gamma Pivot

This parameter changes the default pivot value used by the Pre Gamma parameter.

Pixel Ratio

Modifying this parameter will change the apparent width of the star field, squeezing or stretching it.

Render Quality

This parameter can be used to quantize the color values in each image channel when you want to reduce the total number of colors used per channel.

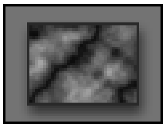
Anti Aliasing

Use this parameter to specify the level of anti-aliasing used.

Algorithm Menu

This menu specifies which anti-aliasing method to use: Runge's Original, which refers to the algorithm used by the author of this node, Dirk Runge, or a Gaussian blur. Try Runge's Original first; it usually produces an optimal result for a star field.

TURBULENCE NODE



The Turbulence node generates a pattern simulating turbulence from a solid field of color by altering the value and opacity of each pixel as specified in the Node Panel.

Turbulence includes controls for creating the turbulence pattern and for specifying the output frame attributes. The frame parameters are described in "Frame Attribute Parameters Common to Source Nodes" (p. 162). The parameters specific to creating the turbulence pattern are described next.

TURBULENCE PARAMETERS

Use the Type, Speed, Detail, and Seed parameters to create the pattern and the Color parameters to change the default color.

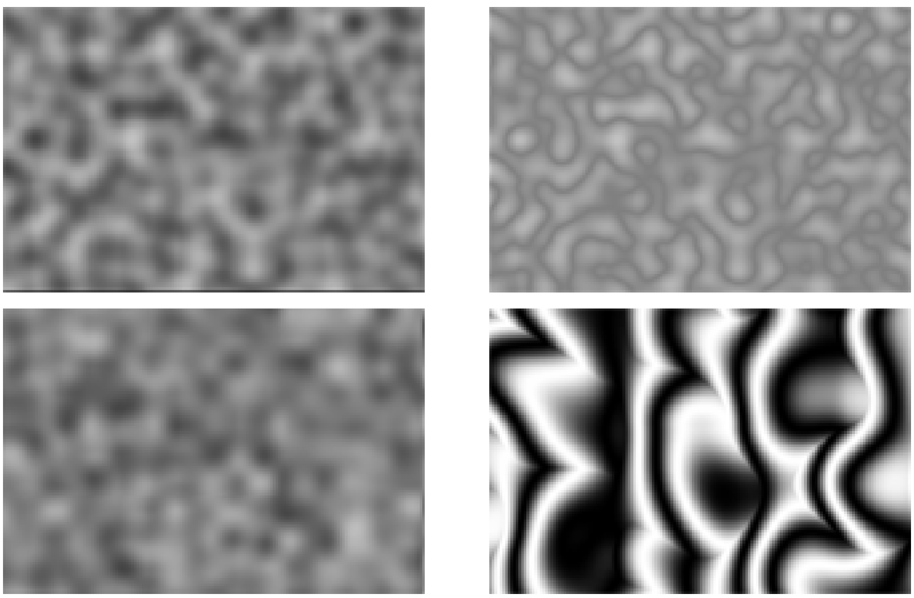
COLOR

The Color parameters are used to change the default color field from which the other parameters generate the turbulence. The default color is white (the maximum RGBA values for the specified bit depth), which results in a grayscale turbulence pattern.

Whichever color you use, each channel is adjusted equally for each pixel so that only the luminance changes, not the hue. For more information about using the controls, see "Using the Color Parameters" on p. 168.

TYPE

Select the type of turbulence to create from the Type menu: Simple (the default), Formica, Clouds, or Marble.



14.22 Examples for the four types of turbulence, with other parameters at their default settings. Clockwise, from upper left: Simple, Formica, Marble, and Clouds.

Speed

Use the Speed parameter to change the slope of the “hills and valleys” of the turbulence pattern. Try experimenting with this control until you see the desired result.

Detail

When any type of turbulence other than Simple is selected, the Detail parameter becomes active. As the name implies, increasing the value of this parameter increases the level of detail in the pattern; decreasing it lowers the level of detail.

Scale

Use the Scale parameter to increase the default scale factor from 1; the range is 1–10. Larger scale values will, in effect, “zoom out” of the turbulence pattern.

Seed

The Seed parameter is used to animate the starting value for the turbulence pattern to produce a changing pattern across time.

chapter 15

MATTE NODES

The Matte nodes enable you to create and modify matte channel data for use in compositing and other image processing operations.

In RAYZ, the term “matte” is generally used synonymously with “alpha” to describe the image channel that holds the opacity values for each RGB triplet and is necessary to most compositing operations. When appropriate, *alpha* may be used to denote a channel of image data and *matte* to describe the channel functionally.

IN THIS CHAPTER

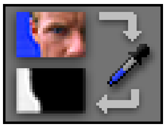
The **Chromakey** and **Lumakey** nodes pull mattes from RGB imagery by using color and luminance values respectively, while the **Roto** node enables you to create matte shapes by drawing them.

The **Ultimatte** nodes provide a suite of tools for working with bluescreen (or greenscreen or redscreen) imagery: they correct flawed backing, kill grain, create a processed foreground with an integrated alpha ready for compositing, and match colors in the foreground with equivalent objects in the background (or between any two images).

It is recommended that you start with the overview to find out which Ultimatte node to use for what operation and the best order in which to use them.

The **Despill** node can be used to remove color spill contamination from a bluescreen shot. **Erode Dilate** is used to shrink or expand edges, operations most commonly performed on mattes.

Chromakey Node



The Chromakey node creates an alpha channel matte based on the colors of the input image, within a narrow range of hue, saturation, and luminance that you specify in the Chromakey Node Panel.

You can output all channels or the alpha channel only.

Chromakey accepts two inputs, the primary input image to be chroma-keyed and an optional second input that is used for a garbage matte. For more information, refer to "Using a Garbage Matte" (p. 224) in the Ultimatte node description.

NOTE:

You can connect an input that already has an (unsuitable) alpha channel. The Chromakey node will discard the incoming alpha channel values.

Using Chromakey

Start by specifying the range of HSL values to key on. Pixels in the input image that fall within the ranges specified for hue, saturation, and luminance in the Node Panel are assigned alpha channel values of 1 (that is, the maximum value for the color depth) and all other pixels in the alpha channel become black (0).

You can select a range of values in the Chroma wheel in the Node Panel or use the Chromakey eyedropper to sample an area of the RGB image in the Viewer.

Then switch the Image Viewer display to Alpha to examine the resulting matte channel, adjusting the Key parameters as necessary to pull an acceptable matte:

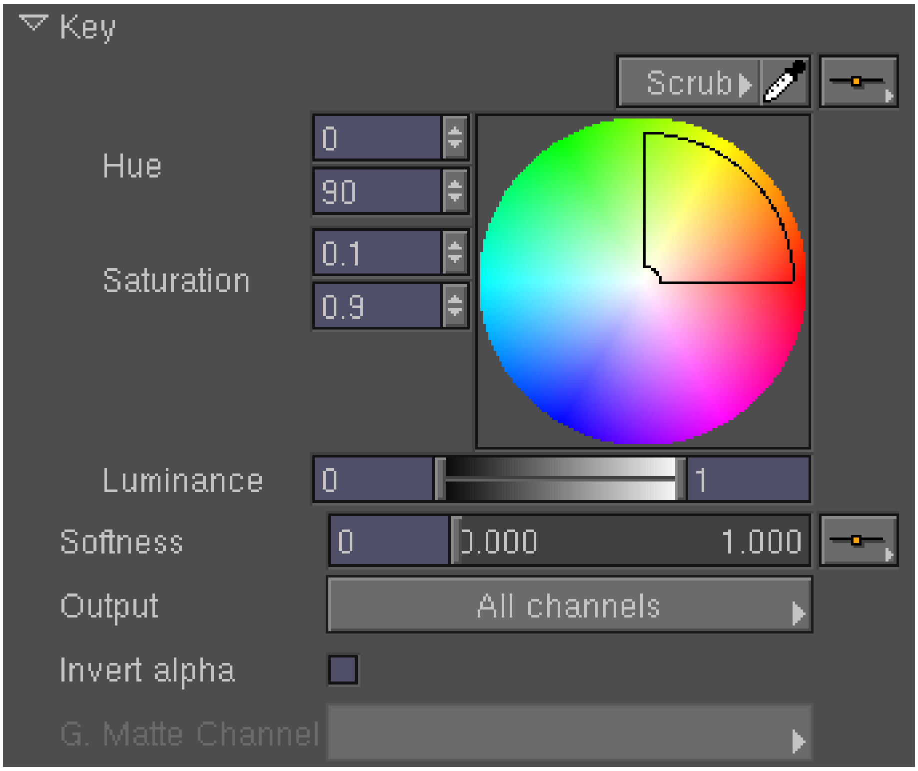
- You can invert the matte if you are keying on image areas that should become transparent, such as the backing area of a foreground sequence shot against a bluescreen.
- You can also use the Softness parameter to decrease the overall opacity of the resulting matte channel.

Chromakey Parameters

Key Parameters

The Key parameter group includes the Hue, Saturation, and Luminance parameters that specify the range of image values to use in the Chromakey operation. All three parameters have a pair of fields: one specifies the low end of the range and the other, the high end.

You can set these parameters by typing numerical values into the fields or by using the associated eyedropper to sample an area of the RGB image in the Viewer. The Hue, Saturation, and Luminance parameters will update accordingly. The Hue and Saturation values can also be set using the Chroma Wheel.



15.1 Chromakey parameters.

Using the Eyedropper

Select Scrub or Drag Box from the eyedropper menu and click the eyedropper button. Move the eyedropper cursor over the image in the Viewer and scrub an area or drag a bounding box around it to sample it.

The minimum and maximum values of the sampled pixels will be used to set the Hue, Saturation, and Luminance parameter values.

Using the Chroma Wheel

You can specify a range of Hue and Saturation values by adjusting the selection box in the chroma wheel. The selection box and its borders can be dragged to resize and reposition the box in the wheel:

- Drag a border radially (around the wheel) to adjust the total width of the selection box and thus the hue range.
- Drag axially (toward or away from the center of the wheel) to adjust the total length of the selection box and thus the saturation range.
- Drag inside the borders to reposition the box in the wheel.

Hue

The Hue parameter defines the range of hues to use within a range of 0 to 360 degrees, which represents the circumference of the color wheel.

To key on a typical bluescreen blue, for example, you might specify a range of 220 to 240 and then adjust it as necessary for your imagery.

Saturation

The Saturation parameter defines the range of saturation to use within a range of 0 to 1, which represents the scale from completely desaturated to fully saturated.

Using a bluescreen shot as an example, you might specify a range of 0.5 to 1, which represents the saturation range typical of bluescreen backings.

Luminance

The Luminance parameter defines the range of luminance to use within a range of 0 to 1. For some imagery, the luminance values of the input image can be used effectively to help pull the matte.

15.2 Luminance slider.

The Luminance slider control has two vertical slider bars. The one on the left is used to set the minimum luminance value and the one on the right, the maximum luminance value. You can also drag the area between the bars to shift the range up or down without changing its magnitude.

Softness

The Softness parameter can be used to soften the alpha channel matte generated by Chromakey. By default, the Softness value is 0, which creates a hard, opaque matte in which all pixels are either white or black. As you increase the Softness value, the matte becomes more transparent.

Output

Select All Channels (the default) from the Output menu to output an RGBA image, or Alpha Only to output only the alpha channel.

Invert Alpha

Check the Invert Alpha box to invert the values of the alpha channel created by the Chromakey node: white pixels become black, and vice versa.

G: Matte Channel

This menu becomes active when an optional garbage matte input is connected to the Chromakey node. It is used to specify which channel of this input to use as the garbage matte, when the image has more than one channel. For more information, see also “Using a Garbage Matte” (p. 224) in the Ultimatte node description.

DESPILL NODE

The Despill node is used to remove color spill from blue-, green-, or red-screen footage.

Using a greenscreen shot as an example, light reflected from the green backing onto objects in the foreground may add a greenish color cast to certain areas of the foreground. In addition, all that green light can bounce around inside the camera where it causes the entire frame to take on a green haze or tinge. Strictly speaking, this is an example of 'flare' rather than 'spill,' but the Despill node algorithm can ameliorate both problems.

Despill works by examining the RGB channel values of each pixel in turn and removing any excess of the backing color. Using greenscreen as an example, the node limits the Green channel value of each pixel to the average of the Red and Blue channel values. Whenever Green is higher than the average of Red and Blue, Green is clipped to the average value; otherwise, the original Green channel value is used.

For a bluescreen shot, on the other hand, the Blue channel value is limited to the average of Green and Red.

The Despill node accepts one input, which may be RGB or RGBA, although the Despill node only operates on the color channels.

DESPILL PARAMETERS

At minimum, you need to specify the Backing Color of the sequence, as this controls the algorithm used for the despill operation. Then, if necessary, you can also adjust the Proportion parameter value to tweak the result.

BACKING COLOR MENU

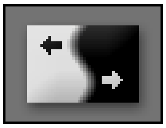
Use the Backing Color menu to specify the screen, or backing, color used to shoot the footage: Green (the default), Blue, or Red.

PROPORTION

The Proportion parameter can be used to increase or decrease the amount of spill removal. Proportion adjusts the values of the two channels being averaged (for a greenscreen shot, this would be the Red and Blue channels), increasing or decreasing them both equally.

The parameter range is 0 to 1; with a default value of 0.5; larger Proportion values increase spill removal, smaller values decrease the effect.

ERODE DILATE NODE



The Erode Dilate node will erode or dilate edges in an image, as you specify. Any or all channels can be adjusted, but this node is used primarily to modify matte edges in an alpha channel for better composites.

The dilate operation works by dilating, or spreading, brighter areas in the image while it erodes, or shrinks, the darker areas. The erode operation, on the other hand, erodes brighter areas and dilates the darker areas.

NOTE:

The “Ultimatte AE (AdvantEdge) Node” (ch. 18, p. 342) is designed specifically to provide matte edge controls when compositing an Ultimatte foreground image over a background. You may want to use Ultimatte AE rather than Erode Dilate in such cases.

Erode Dilate accepts two inputs, the image to be processed and an optional mask input used to control which pixels of the primary input are processed. For more information, see “Using Mask Inputs” (ch. 7, p. 102).

ERODE DILATE PARAMETERS

OPERATION

The Operation menu is used to select Erode or Dilate for the operation. Erode is selected by default.

MAGNITUDE

The Magnitude parameter controls the size of the effect. The range of the slider control is 0 to 0.05, to allow fine control over the adjustment, although the upper end of the range is unconstrained. The default value of the Magnitude parameter is 0.01.

SOFTNESS

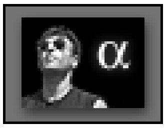
The Softness parameter uses subpixel accuracy to modify the softness, or falloff, of the effect in a range of 0–1. When the Softness parameter is set at 0, the original edge pixel values are used to erode or dilate the edge by the specified Magnitude value.

Nonzero Softness values will gradate the pixel values from the original edge inward (for erode) or outward (for dilate) to the new edge. The greater the Softness value, the steeper the slope, or falloff. A Softness value of 1 produces a circular gradient from the original edge to the new edge.

CHANNEL SELECT

Use this parameter to specify which channels of the input image will be processed by the node. By default, only the alpha channel is selected.

LUMAKEY NODE



The Lumakey node creates an alpha channel matte based on the luminance of the input image, within a narrow range of black and white that you specify in the Node Panel. You can output all channels or the alpha channel only. You also have the option of inverting the outgoing alpha matte.

Lumakey accepts two inputs, the primary input image to be lumakeyed, and an optional mask input that controls which pixels in the primary input are used in the lumakey operation. For more information, see "Using Mask Inputs" in chapter 7 (p. 102).

NOTE:

You can input an image that already has an (unsuitable) alpha channel. The Lumakey node will discard the incoming alpha channel values.

LUMAKEY PARAMETERS

OUTPUT

Use this menu to specify whether the Lumakey node should output all channels (the default) or the alpha channel only.

INVERT MATTE

Check this box to invert the matte channel created by the node.

KEY ON

Use this menu to select the type of luminance to key on:

- Red Channel
- Green Channel
- Blue Channel
- Film Luminance (the default)
- NTSC Luminance
- PAL Luminance

You can use one of the existing color channels, or have the node generate a luminance channel from the RGB image. For the formulas used, refer to *Appendix A: How RAYZ Computes Luminance Values* (p. 439).

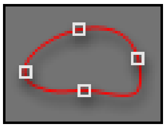
BLACK RANGE

Use the Black Min and Black Max parameters to set the range of blacks to use in the Lumakey operation.

WHITE RANGE

Use the White Min and White Max parameters to set the range of whites to use in the Lumakey operation.

Roto Node



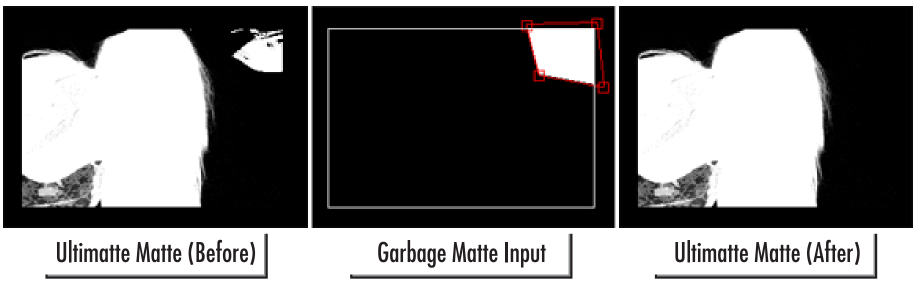
The Roto node is used to create and manipulate closed spline shapes, typically in the matte channel, although you can apply them to any image channels you choose. You can draw, edit, scale, rotate, invert, and animate these shapes. Most commonly this functionality is used to create garbage mattes or mask images.

NOTE:

The section of this node description on “Using the Roto Node” (p. 210) explains how to draw and edit shapes using the Roto tools in the Image Viewer, while the section on “Shape Controls in the Roto Node Panel” (p. 215) explains how to use the Node Panel parameters provided for each shape you create.

Creating Garbage Mattes

Garbage mattes are used to remove unwanted elements from an image before compositing it with other imagery. For example, a garbage matte could be used to remove lights or other equipment from a bluescreen shot before the subject was composited over a background scene.



15.3 Nodes with dedicated garbage matte inputs, such as Ultimatte, assume that pixels in the garbage matte with non-zero values represent areas that should be matted out of the image being processed.

The Ultimatte and Chromakey nodes accept an optional garbage matte input, as described in “Garbage and Holdout Matte Inputs” on p. 224.

The output of a Roto node can just as easily be used as a holdout matte input to an Ultimatte node to prevent print-through in a foreground element. To create a holdout matte you would draw matte shapes defining the areas that should be opaque when composited.

Creating Masks

A Roto matte can also be used as a mask; that is, to control which areas of an image are processed in another node, such as a color correction node. For example, a matte could be drawn to isolate an automobile that was filmed in black-and-white. The original shot could then become the primary input to an Indexed Color node, and the output of the Roto node would become the mask input. This would enable you to tint only the car while leaving the background monochrome.

For more information about using mask inputs in RAYZ, refer to "How Masks Control Node Processing" in chapter 7 (p. 102).

INPUTS AND OUTPUTS

The Roto node accepts one input, which may or may not have an alpha channel:

- If the input has an alpha channel, you can use it as is, invert it, or clear it to black or white by selecting the appropriate option from the Alpha menu in the Node Panel. The alpha channel will be cleared to Black by default.
- If the input does not have an alpha channel, the Add Alpha checkbox in the Node Panel becomes active. It will be checked by default. You can use the Alpha menu to specify whether it should be set to black (the default) or white.

NOTE:

If you will be working on the RGB channels only, you can prevent the node from creating an alpha channel for an RGB input by unchecking the Add Alpha box.

The node will output all of the image channels, or the alpha channel only, as you specify in the Output menu in the Node Panel. The default is to output All Channels.

15.4 The Roto Node Panel provides menus to control the incoming alpha channel and the channels to output.

USING THE ROTO NODE

The Roto node provides tools directly in the Image Viewer for drawing closed splines around portions of an image, so you always start by displaying the Roto node image in a Viewer.

15.5 Roto node tools available in the Image Viewer.

NOTE:

If the Roto tool strip doesn't appear in the Image Viewer when it is displaying the Roto node image, turn on the Node Controls option (right-hold anywhere in the viewspace to access the Viewer Actions menu).

Display the RGB image in the Viewer to use it as a reference when drawing a spline shape that will be used to isolate a specific object or area in the image.

Display the Alpha channel to view the roto matte and adjust its transparency level or edge width. To evaluate an animated matte, use the Flipbook controls in the Viewer to play the animated roto sequence.

You can create as many roto shapes as you need for an image. For each shape you create, a corresponding entry is created in the Roto Node Panel. The Node Panel list enables you to control overall characteristics of each shape individually, as described in "Shape Controls in the Roto Node Panel" on p. 215.

AUTOKEY STATUS

If you intend to animate the outline or position of a shape over time, you can turn on Autokey before you start. This will animate the shape automatically by creating a keyframe whenever you go to a new frame in the sequence and modify the shape in any way. You can also use the Animation menu for the shape entry in the Node Panel. See also "Animating Parameter Values" in chapter 7, p. 99.

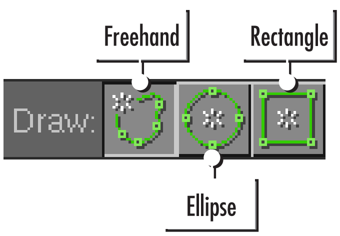
You can add, delete, and modify shapes and points at any frame. For example, you could draw a shape at frame 1, reposition it at frame 15, and add a point at frame 25 to reshape the outline. And you could create another shape starting at frame 7 that held its position until you deleted it at frame 100, and so on.

TIP:

To obtain the smoothest effect when animating a matte shape, set as few keyframes as possible. This will help to avoid the "rotoscope flicker" effect that can occur at the matte edges when the interpolation from keyframe to keyframe is too abrupt.

DRAWING A SHAPE

The Roto tool strip offers buttons used to specify drawing and editing modes. Freehand drawing mode is selected by default when a new Roto node is created, however, you can select Ellipse or Rectangle instead. Once you have closed a shape, the node automatically switches to an edit mode. To draw another shape, select a draw mode again.



15.6 The default Draw mode is Freehand, in which you draw one segment at a time in order to follow the outline of an object in the image. The other two drawing modes, Ellipse and Rectangle, enable you draw entire shapes in one motion.

Freehand Drawing

Click the Freehand button to select it, if necessary.

Linear Spline Segments To draw a linear spline (straight line), click anywhere in the image area to create an anchor point, move the mouse, and click again to create a second anchor point. RAYZ will draw a linear spline segment connecting the two points. Continue moving and clicking the mouse to add additional segments.

Curved Spline Segments To draw a curved spline (with point handles), drag in the image to create the anchor point instead of clicking (the farther you drag, the longer the control handle on the point). Release the mouse button and a spline will appear "attached" to the cursor. Move the cursor to wherever you want to place the next point and either click (to start a new linear segment) or drag again (to start a new curved segment).

To close the shape, click on the first point you created. Once the shape is closed, you can manipulate the shape as a whole or any of its points as described in "Editing a Shape" (p. 212).

> **TIP:**
>
> For maximum efficiency, use as few points as necessary to create a shape.

Creating Ellipses and Rectangles

In addition to drawing a shape freehand, you can select one of the other two drawing modes to create an ellipse or rectangle. With the appropriate drawing mode selected, drag in the image to draw the shape in one motion.

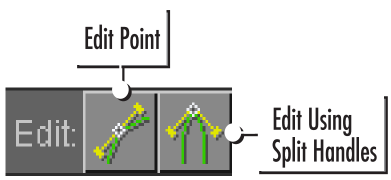
> **TIP:**
>
> To constrain the proportions of the shape to a circle or square, hold down the Shift key while dragging.

Editing a Shape

Once you have drawn a shape, you will probably want to adjust it. One of the editing modes should be selected by default; if not, click an Edit Mode button.

Choosing an Edit Mode

There are two point editing modes to choose from, however, the choice is only relevant when you are manipulating Bezier handles (the handles on points that control curved spline segments).



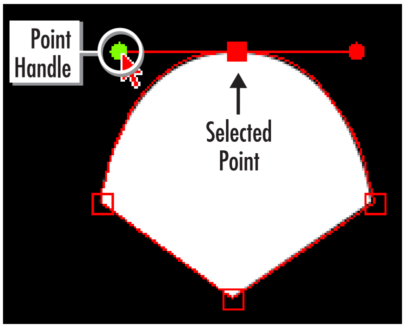
15.7 Edit Mode Buttons.

In the default point editing mode (the Edit button on the left), dragging a point handle adjusts the curve on both sides of the point equally. In split mode (the Edit button on the right), each end of the point handle is adjusted independently.

SELECTING POINTS AND SHAPES

You can modify an entire shape or shapes, or any individual point or points, depending on what is currently selected:

- To select a point, click it.
- To select multiple points, hold down the Shift key as you click each in turn, or drag a selection box around them.
- To select the entire shape, click on any spline segment. (To drag the selected shape, however, drag by a point rather than a spline.)



15.8 A selected point is solid, while an unselected point is hollow. Point handles appear when a point that controls a curved segment is selected.

As you move the cursor over the image, the line, point, or point handle currently under the cursor will highlight in a contrasting color so that you can tell which object you will be selecting when you click.

MANIPULATING POINTS AND SHAPES

To change the outline of a roto shape, adjust the position, type, and number of points in the shape:

- To **move** a selected point, simply drag it across the image.
- To **delete** a selected point, press the Delete key.
- To **add** a new point to a line, hold down the Control key and click the line. A new point will appear where you clicked, which can then be adjusted like any other.

To reposition a selected shape, drag it by any of the selected points. (You can't drag a shape by its splines.)

NUDGE HOTKEYS Use the arrow keys on the keyboard to nudge a selected point or shape in one-pixel increments in the direction indicated by the arrow. You can also hold down the Shift key as you press an arrow key to move in larger increments (4 pixels).

To change the default increment values, go to Edit > Preferences > Settings and select Nudge Size (Small) or Nudge Size (Large).

TIP:

To duplicate a shape, use the Layer Actions menu to copy and paste its entry in the Node Panel, as described in "Copying, Pasting, and Deleting Layers" in chapter 7, p. 94.

USING POINT HANDLES

You can also adjust the slope and acceleration of a curved spline by manipulating the point handles, which appear whenever the point is selected. To adjust the slope and acceleration of a curve, drag the handles that extend from the anchor point:

- Rotate a handle about the point to adjust the slope of the curve.
- Shorten or lengthen a handle, by dragging it toward or away from the point, to change the acceleration of the slope.

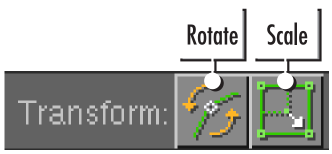
To adjust the segments on each side of a spline point separately, switch to Split Handle mode, as described in "Choosing an Edit Mode" (p. 212).

CHANGING A POINT TYPE

As described above in "Freehand Drawing" (p. 212), you create linear or curved segments as you draw a shape. However, you can change any control point after the shape has been drawn. Click the point to select it, and then hold down the **Control key** and click it again: Linear points will become curve points and vice versa.

TRANSFORM MODES

Click the Rotate or Scale buttons to choose one of these modes. When an entire shape is selected, the rotation or scaling applies to the entire shape. When an individual point is selected, the operation applies to the selected point (or points) only.



15.9 The Transform modes are Rotate, which rotates selected points or shapes, and Scale, which scales selected shapes up or down. (You can translate selected points or shapes in Edit mode.)

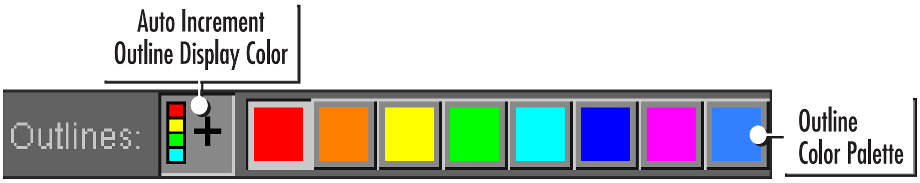
In Rotate mode, a pivot point marker appears in the image. This indicates the pivot around which the selected items will rotate. By default, the pivot appears in the center of the image. To change the pivot, drag it to a new location.

TIP:

When animating a shape across time, you may want to use Rotate mode to reposition the shape (or individual points) in a smooth arc to mimic certain types of motion.

OUTLINE COLOR CONTROLS

The Roto tool strip also provides buttons to control the color of a shape's outline; that is, the color used to display the spline segments and points of the shape overlay.



15.10 You can change the color of any shape outline to make it easier to distinguish over the image, or to color-code related shapes.

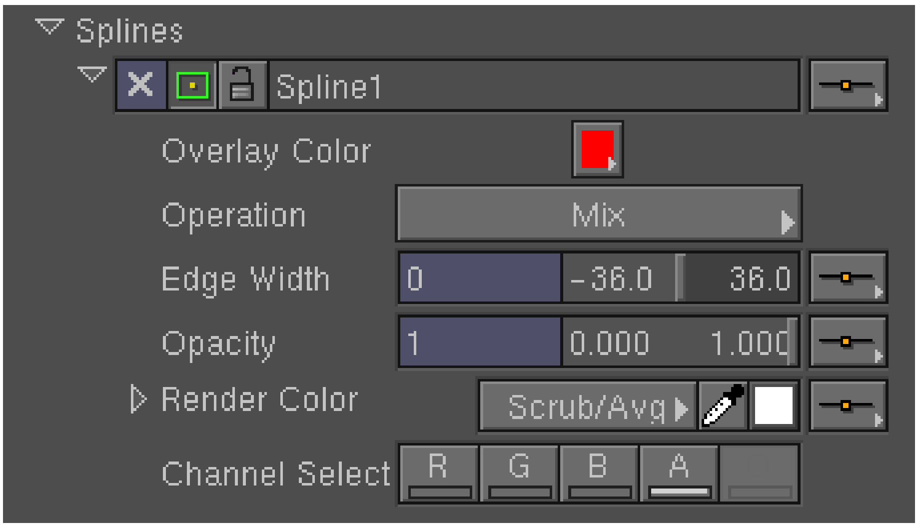
The Outlines button toggles auto-increment mode on and off. In auto-increment mode, each new shape you create is assigned the next color in the palette. If you turn this feature off, however, each new shape is assigned the color currently selected in the palette.

NOTE:

After a shape has been created, you can change the color of its outline display by selecting a different color in the Overlay Color menu for the shape, which is located in the Roto Node Panel.

Shape Controls in the Roto Node Panel

An entry is created in the Splines list of the Node Panel for each shape you create.



15.11 Shape-specific parameters available in the Roto Node Panel for every shape you create.

Using the controls in the top line of the shape entry, you can

- reorder the shapes in the list, to control how they are layered over each other (this is applicable when shapes overlap). Click once on the shape name to select the entry and then drag the entry up or down.
- hide a shape temporarily, which turns off the shape display in the image. See "Hiding/Disabling Layers and Overlays" in chapter 7, p. 94 if you need more information.
- hide the shape overlay, which hides the outline, points, and handles for a shape. See "Hiding/Disabling Layers and Overlays" in chapter 7, p. 94 if you need more information.

- lock a shape, which prevents you from accidentally changing a shape as you work in the Image Viewer. Click the button with the lock icon.
- change the default name to something more descriptive. Double-click on the shape name to turn it into an editable text field.
- use the Animation menu to add and delete keyframes or change the type of interpolation between keyframes. (See also "Animating Parameter Values" in chapter 7, p. 99.)
- duplicate a shape (by using the Layer Actions menu to copy and paste it). See "Copying, Pasting, and Deleting Layers" in chapter 7, p. 94.
- delete a shape (using the Layer Actions menu). See "Copying, Pasting, and Deleting Layers" in chapter 7, p. 94.

NOTE:

Hiding a roto shape does not disable the shape layer unless you hide the shape overlay also. This enables you to turn off display of the shape to get a better view of the image under it while still being able to use the overlay to adjust the outline of the shape.

When a shape entry is expanded, you have access to additional parameters that control shape characteristics such as opacity, edge width, and blend operation, as well as which image channels will be affected by the shape.

Overlay Color

This parameter enables you to change the color of the shape overlay; that is, of the splines with their points and handles. You can use this feature to color code related shapes or make the overlay more visible over the image.

To change the overlay color, click and hold the Color button to access a menu of color swatches and select a different color.

Operation

The Operation menu specifies how the roto shape will be blended with the underlying image and any other overlapping roto shapes. The default blend mode is Mix, however you can also choose Add, Subtract, Minimum, Maximum, or Multiply.

Edge Width

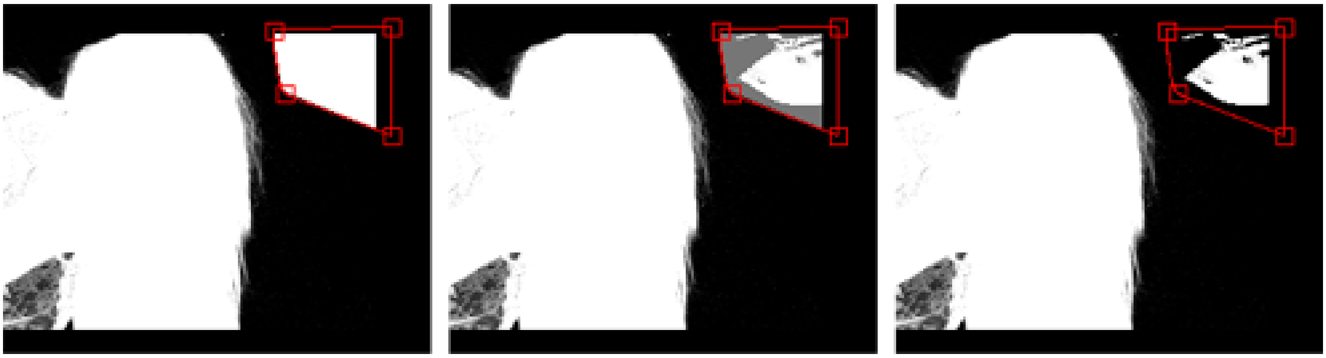
The Edge Width parameter is used to feather the edges of a shape by creating a gradual falloff either inward or outward from the edges. The default value is 0, which creates a hard edge that follows the spline outlines. Negative values create an inward falloff and positive values create an outward falloff.

Opacity

The Opacity parameter specifies how much of the underlying image shows through the shape; that is, it controls the opacity of the shape and not the image channel.

This means that the effect of the opacity setting depends on the color of the shape, the image channel values under the shape, and whether there is any overlap with other shapes.

By default a shape is fully opaque. If you draw a roto shape over an area of the alpha channel that has a mixture of values, for example, you can use the Opacity parameter to control how much the alpha channel values show through the roto shape.



15.12 The Opacity value controls how much of the underlying image shows through the shape. As this example illustrates, zero opacity does not make opaque pixels in the alpha channel transparent.

NOTE:

To change the underlying alpha channel value to black, so that the area defined by the shape will be transparent in the output, use the Alpha parameter in the Render Color parameters instead, as described next.

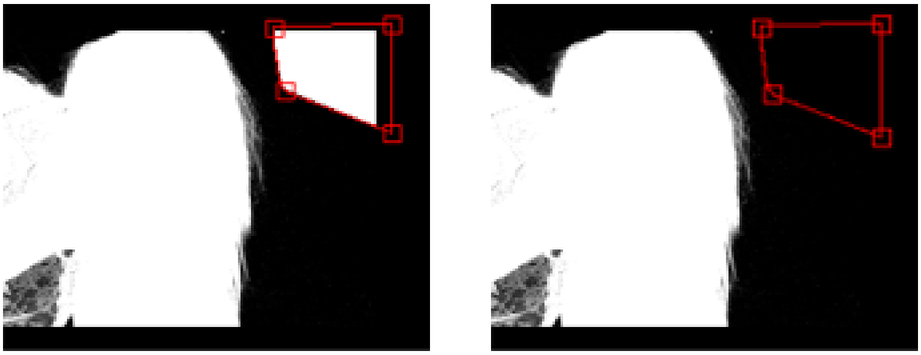
Render Color

This parameter group specifies the color values to apply to the channel data of the pixels delineated by the roto shape. Expand the parameter to access the standard RAYZ color parameters. (See also "Using the Color Parameters" in chapter 14, p. 168.)

Most commonly this parameter group is used to specify the alpha channel value for the shape. The default value, white, makes the area of the outgoing alpha channel under the shape fully opaque. Changing the value to 0 would make the area black, that is, completely transparent.

TIP:

You can matte out an unwanted element in an otherwise adequate matte by drawing a roto around problem area and setting the Alpha channel value in the Render Color parameters to 0.



15.13 In the image on the left, the Alpha channel value in the Render Color parameters is set to maximum (255 for an 8-bit image). The example on the right shows the effect of an Alpha value of 0.

TIP:

You can also use the Alpha parameter to invert a shape and use it as a cutout to punch a hole in another shape. For example, to create a donut-shaped matte, you could draw a large circular shape, and then a smaller circular shape centered over the large one. Finally, you would set the Alpha value of the smaller circle to 0 (leave the Alpha for the large shape at its default value of 1).

Channel Select

Use the Channel Select parameters to specify which channels of the image will be affected by the roto shape. By default, only the alpha channel is selected, but you can actually apply the roto shape to any or all channels in the image.

Overview of the Ultimatte Nodes

Ultimatte is bluescreen compositing software for film and video. It was designed by the Ultimatte Corporation, and its functionality is included in RAYZ. (You do not need a separate Ultimatte license to use it.)

Ultimatte is designed to optimize the bluescreen compositing process by seamlessly compositing imagery while retaining shadows, fine detail, and transparent objects in the foreground image.

As an example of the bluescreen process, a filmmaker might shoot an actor in front of a background that has been painted entirely blue, using paints formulated specifically for this purpose such as Rosco Labs Ultimatte Blue #5720, or Ultimatte Super Blue #5722, for wire removal. This bluescreen background area is referred to as the *backing area* or *screen area*.

Later, using the Ultimatte nodes, a digital artist can prepare a processed foreground image (an image of the actor, with the backing matted out) and seamlessly composite it over a background clip of a burning building, lunar landscape, or other dramatic scene.

NOTE:

In actuality, the screen may be blue, green, or red; in fact greenscreen is used quite commonly. However, this manual uses the term “bluescreen” generically.

Choosing the Right Ultimatte Node

RAYZ includes several Ultimatte matte nodes, each of which is optimized to solve a particular type of bluescreen image processing problem:

Ultimatte This node generates a processed foreground and matte; that is, an image of the foreground subject with any spill removed and the backing area suppressed to black, and the Ultimatte matte attached as the alpha channel. You may later composite this image using a node such as Multi-comp or Ultimatte AE.

Ultimatte CSC (Classic Screen Correction) If you have a reference clip (clean plate), this node can be used to correct uneven areas in the bluescreen backing behind the foreground subject. Use Ultimatte CSC before generating the matte in the Ultimatte node.

Ultimatte GK (Grain Killer) This node can be used to remove film grain or video noise from the bluescreen backing area of a foreground before generating the matte in the Ultimatte node.

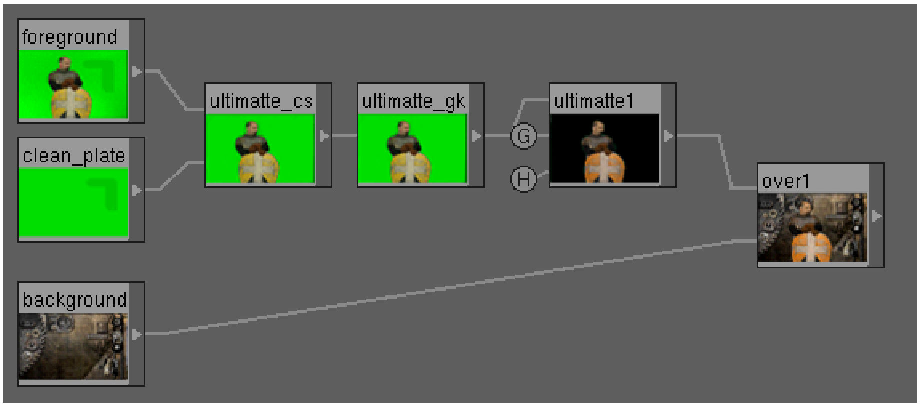
Ultimatte CC (Color Control) This node enables you to adjust the color balance of one film clip to make it appear natural or appropriate for the color values of a background clip when composited. Use Ultimatte CC after the other Ultimatte matte nodes.

NOTE:

RAYZ also includes an Ultimatte composite node. While any suitable composite node in RAYZ can be used to composite an Ultimatte processed foreground image, the Ultimatte AE node employs a method developed by Ultimatte to help fix any edge problems in the composite. For more information, see the description of the "Ultimatte AE (AdvantEdge) Node" in chapter 18, p. 342.

Overview of the Ultimatte Process

The following steps provide an overview of how the Ultimatte nodes are used in a typical compositing network. For detailed information on each node and how it is used, refer to the individual node descriptions that follow this overview.



15.14 Example of a compositing network using the Ultimatte matte nodes.

Step A: Screen Correction

The foreground image is connected to an Ultimatte CSC node to correct any flaws in the backing, or screen area. In an ideal situation, the screen area against which an actor is filmed would be of perfectly even tone. In reality, the backing almost always shows imperfections due to variations in ambient lighting and flaws in the surface. The Ultimatte CSC (Classic Screen Correction) node enables you to perfect the screen area while retaining shadow information.

A clean plate of the stage, shot at the same time as the foreground subject, is connected to the second input of the Ultimatte CSC node to be used as the reference screen for the correction process.

Step B: Grain Killing

For filmed imagery (or for a noisy video clip) the Ultimatte GK node is used to remove grain from the backing area of the foreground image. This will prevent the appearance of a double layer of grain when the foreground is composited over a filmed background.

The Ultimatte GK node should be used after the screen area has been corrected using the Ultimatte CSC node and before the final matte is created in the Ultimatte node.

Step C: Generate Processed Foreground with Matte

The output of the Ultimatte CSC node, or GK node, if used, is connected to the Ultimatte node to create a processed foreground that is ready to be composited. The Ultimatte node enables you to remove flare or spill on foreground objects and create a matte that is output as the alpha channel of the image.

Step D: Composite

The processed foreground image is composited with the background image using a composite node. If the foreground image is one of many foreground elements to be composited, you would use a Multi-comp node to composite all the layers.

Step E: Reviewing the Output

When you use the Ultimatte nodes to prepare bluescreen images, the best way to ensure the optimal result for any particular shot is to review the imagery as it looks when composited over the background in one of the RAYZ composite nodes. This is the true test of whether the various node parameter settings need further adjustment.

Step F: Color Control Option

In some cases you may find that the color balance of the foreground image does not match the background image perfectly in the composite. To correct this situation, you would insert an Ultimatte CC node into the network between the Ultimatte node and the composite node and use the Ultimatte CC controls to correct the color balance of the foreground image.

In Summary

The Ultimatte matte nodes enable you to create even bluescreen backings, remove film grain, process the foreground image, and correct the color balance of an image to be composited.

The output of the process will usually be a single processed foreground image suitable for seamless compositing in another RAYZ node.

How Ultimatte Generates Matte Data

The Ultimatte process is designed to create highly accurate matte data based on the color values of the foreground image. Most colors occurring in nature have at least some amount of red, green, and blue when represented in RGB colorspace. Given this fact:

- How does Ultimatte determine, based on the RGB value of a pixel, whether it is part of the backing or the foreground?
- And how does Ultimatte determine whether a foreground pixel should be opaque or partially transparent when composited over the background?

Using a blue backing screen as an example, the colors used for the bluescreen backing will have extreme differences between the blue channel value and the red and green channel values.

The Ultimatte process starts by examining the relationships among the values of the three channels of an RGB image. For each pixel, Ultimatte determines which has the greater value, the red channel or the green channel. This value is then subtracted from the blue channel value to obtain a difference value for the pixel.

Interpreting Difference Values

Areas where this difference value is pronounced are interpreted as backing. The greatest difference value is called the peak point, and is considered to be totally transparent for purposes of the final composite; that is, 100 percent of the background will show through the foreground wherever a pixel has this value.

In reality, unobstructed areas of the backing will display some variation in color. This means that the pixel with the highest peak point value might not be the most representative blue in the backing, so Ultimatte enables you to specify the ideal blue to use as the peak point.

Otherwise, unobstructed areas of the backing that have slightly lower difference values than the peak point could be interpreted as if they were not entirely transparent (as if they were in a faint shadow, perhaps).

Areas where the difference value is 0 or less are interpreted as foreground and considered to be totally opaque for purposes of the final composite. For example, a pure white pixel has equal values for R, G, and B, so its difference value is 0; and negative numbers will result for colors where B is less than the max of R or G.

Pixels with values that fall between the peak and minimum difference values are assigned varying levels of opacity in a linear distribution. This is what allows Ultimatte images to retain fine details, shadows in the backing area, and soft edges around foreground objects when composited over a background.

However, this also means that blue areas in the foreground subject may be treated as backing and result in print-through in the final composited image. “Print-through” refers to areas where the background is visible, or partially visible, through foreground objects that are supposed to be opaque.

Adjusting Matte Density

Ultimatte can correct print-through problems in a matte by altering the formula used to determine the difference value for each pixel: a matte density value is assigned for the image, and the max of R or G is first multiplied by this matte density value before it is subtracted from blue.

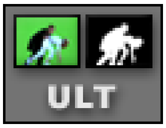
Boosting the value of the channel that is subtracted from blue results in a lower difference value. When the difference values for an image are decreased in this way, pixels which previously had some transparency can become entirely opaque, eliminating print-through.

Raising matte density too much, however, can harden matte edges and eliminate fine detail, with adverse affects on the realism of the composite.

The Ultimatte nodes include a number of controls that enable you to adjust these values as needed for any particular image.

In the Ultimatte node, for example, adjusting the Matte Controls affects the alpha channel that is output for subsequent compositing. In the CSC node, on the other hand, the matte density parameters affect the matte data that is used in the screen correction operation applied to the RGB channels.

Ultimatte Node



The Ultimatte node is used to create and output a clean, processed foreground image: that is, an image of the foreground subject with any flare removed, the backing area suppressed to black, and the Ultimatte matte attached as the alpha channel. You will then be able to composite this image using one of the RAYZ composite nodes.

You can specify whether the node will output only the matte channel or all four (RGBA) channels.

> **TIP:**
>
> Before you use the Ultimatte node, you should first use the Ultimatte CSC node to correct any flaws in the backing, or screen area. Screen correction is a critical step in the process of producing high-quality mattes. Also, if you plan to use the Ultimatte GK node, you should do so prior to processing the image in Ultimatte.

Matte density values can be adjusted to correct any holes in foreground areas of the matte caused by spill or flare that would result in print-through in the final composite. The Ultimatte node also provides flare controls to remove any discoloration caused by spill from foreground objects.

Refer to "Using Ultimatte" on p. 225 for general guidelines and to "Ultimatte Parameters" on p. 226 for descriptions of specific Node Panel parameters.

Garbage and Holdout Matte Inputs

The Ultimatte node accepts three inputs:

- foreground image
- garbage matte (optional)
- holdout matte (optional)

The optional inputs are designed to take a garbage or holdout matte. The Garbage matte input is labeled with a "G" and the holdout matte input with an "H."

Using a Garbage Matte

Usually a simple shape, a garbage matte is used to eliminate, or matte out, unwanted elements from an image that will be used as a foreground in a composite. Typically this "garbage" consists of rigging, microphones, or other equipment that showed up in frame, or a flaw in a bluescreen backdrop. A garbage matte input can also be used to extend the backing area in cases where the bluescreen was not large enough to fill the frame.

White areas in a garbage matte define the areas in the foreground image that should be matted out—that should become transparent in the matte channel that will be output from the node. Garbage matte pixels with a value of zero (that is, black) are simply ignored.

Using a Holdout Matte

A holdout matte, on the other hand, is used to specify that certain areas of the foreground do show up in the composited image; that is, to specify that these areas will be opaque in the foreground matte that is output from the Ultimatte node.

It may be necessary to use a holdout matte input to prevent print-through of blue or very shiny foreground elements—to fill in holes in the foreground matte—when the Matte Controls cannot be used for this purpose without adversely affecting the matte edges.

Unlike a garbage matte, the white areas in a holdout matte define areas in the foreground image that should be opaque when composited.

The Roto Node (p. 209) is often used to create garbage and holdout mattes.

Using Ultimatte

Step 1: Pick a Reference Color

Display the RGB image in an Image Viewer. Use the eyedropper tool in the Reference Color parameter in the Ultimatte Node Panel to pick a pixel in the backing area of the image. Ultimatte Intelligence will use the color values as a reference for setting color logic and generating matte data. See "How Ultimatte Generates Matte Data" on p. 221 for more information.

If the backing is a homogenous field of color (blue, green, or red), you can select any pixel in the backing that is totally unobstructed by shadows. If it is not, use an Ultimatte CSC node to correct the backing areas before using Ultimatte. Refer to the introduction, "Overview of the Ultimatte Nodes" (p. 219), for more information about when and in what order to use each Ultimatte node.

Step 2: Examine the Matte

Next switch from RGB to Alpha channel display in the Image Viewer to view the matte channel. This display represents the alpha channel that the node will output. Any holes in the matte will appear as gray or black areas within white foreground objects.

Step 3: Adjust Matte Density

Fix any matte density problems caused by spill to prevent print-through using the "Matte Controls" (p. 226).

STEP 4: CORRECT DISCOLORATION

After you are satisfied with the matte density values, you can use the "Flare Controls" (p. 229) to correct discoloration, especially of magentas, yellows, and browns, that can be caused by spill and flare. Compare the composited image with the original foreground image to identify these problem areas.

NOTE:

For the best result, be sure to refer to the Matte Controls and Flare Controls descriptions. Adjusting any of these parameters too dramatically can adversely affect the realism of your composite.

ULTIMATTE PARAMETERS

REFERENCE COLOR

Use the Reference Color eyedropper to click on the backing area of the image. This sets the reference value for the backing color that is used in the Ultimatte node operation.

CHANNELS

Use this menu to specify which channels the Ultimatte node will output:

- RGBA (the default)
- Matte Channel Only

INVERT MATTE

Check the Invert Matte box to invert the values of the outgoing alpha channel, so that transparent (black) areas become opaque (white) and vice versa.

MATTE CONTROLS

The Matte Controls are used to adjust matte density levels.

MATTE DENSITY & BLACK GLOSS

The Matte Density and Black Gloss parameters enable you to adjust matte density values to eliminate print-through, which occurs when pixels in a foreground object are interpreted as being part of the backing.

Both parameters function by minimizing patches of gray or black within the white foreground objects of the matte. If you set either parameter too high, however, the result may include hard, dark edges around foreground objects and darkening of shadows and backing areas.

Adjust Matte Density and Black Gloss interdependently: increasing Black Gloss may allow Matte Density to be set lower. Readjust both parameters to be as low as possible while still ameliorating the problem.

TIP:

The Black Gloss 2 and RGB Density parameters, which operate on the RGB channels rather than the matte, may help to minimize edge problems caused by Matte Density and Black Gloss.

MATTE DENSITY To control print-through in bright foreground objects, use Matte Density. This situation may be caused by flare or spill, or when a foreground color is too close to the backing color (a blue-eyed actor against a bluescreen, e.g.). The Matte Density default value is 50.

NOTE:

Matte Density will eliminate *print-through* problems caused by spill. However, to correct the *discoloration* spill causes, use the Flare Controls. (The Matte Controls should be adjusted before the Flare Controls.)

BLACK GLOSS To control print-through in black glossy or dark foreground objects, use Black Gloss. This situation can be caused when a black glossy object is reflecting color from the backing and is interpreted as being a dark area of the backing. The Black Gloss default value is 0.

BLACK GLOSS 2

The Black Gloss 2 parameter enables you to stop print-through on foreground objects that suffer from excessive spillage from the backing. Unlike Black Gloss, this parameter adjusts RGB values instead of the matte, and setting this parameter too high may alter the color of objects in the foreground. The default value is 0.

RGB DENSITY

The RGB Density parameters enable you to adjust RGB values to reduce hard, dark edges from objects with strong red (skin tones), green (flora), or blue components. Unlike Matte Density, this parameter adjusts the RGB channels instead of the matte.

If you set one of the RGB Density values too low, the result may be print-through in foreground objects of the respective color. (For example, a low blue value may result in print-through in blue foreground objects.) In that case, the Matte Density control may need to be readjusted.

You can use the master control to modify all three channels equally, but in most cases you will probably want to expand it to modify the individual color channels separately. The default value for all channels is 100 (full density).

CLEAN UP AND CLEAN UP BALANCE

The Clean Up and Clean Up Balance parameters are interdependent, with Clean Up Balance controlling how Clean Up affects the foreground relative to the screen area. In general, the Clean Up value should be as low as possible.

CLEAN UP The Clean Up parameter enables you to eliminate fine imperfections in the backing. However, it can also eliminate subtle details of the foreground, such as strands of hair, smoke, shadows, or reflections, and ruin the realism of the composite. The Clean Up default value is 0.

CLEAN UP BALANCE Use the Clean Up Balance parameter to assign the degree of influence that the Clean Up parameter has on the foreground relative to the backing area of the image. The default value is 50, which specifies that the Clean Up parameter will affect the foreground and backing equally. Try adjusting this parameter when the Clean Up parameter darkens edges or makes them glow.

LEVEL BALANCE

The Level Balance parameter will help you even out imperfections or textures in the backing that are noticeable in the composite.

Increasing this parameter will not brighten the background, but it will increase the level in the shadows and transparent areas of the composited image. On occasion, lowering this parameter will result in an enhancement to the fine edge detail in foreground objects. The default value is 50.

SHADOW NOISE

The Shadow Noise parameter enables you to reduce noise in shadows and glare areas. The default value is 50.

RGB VEIL

If there are slight variations in color in the backing area, a residue of the backing color may remain in some areas when Ultimatte suppresses the backing color to black. This veil of color can discolor the background image in the composite.

The RGB Veil parameters are used to adjust the RGB color components in the backing area to suppress this residue. Decreasing these values will remove veiling from the backing area, but may cause dark edges around foreground objects. Increasing the values will add a colorized tint or haze over the background.

You can use the master control, which has a default value of 50, to modify all three channels equally, or expand it to access the individual channel controls.

NOTE:

If possible, you should use the Ultimatte CSC node to even out the backing area before the image is input to the Ultimatte node.

FLARE CONTROLS

The Flare Controls are used to correct the discoloration of foreground objects caused by spills and flares. During shooting, light reflecting from the backing may "spill" onto the foreground subject, causing discoloration and print-through. This color contamination of the foreground can also occur from reflections within the camera lens, or lens flare.

NOTE:

To correct the matte density problems (print-through) caused by spills and flares, use the Matte Controls first. Then use the Flare Controls to correct the discoloration.

HOW FLARE SUPPRESSION WORKS

Ultimatte spill/flare suppression works by analyzing the color components of each foreground pixel. Using a blue backing as an example (this suppression works similarly for green and red screens), for most colors in nature, if the blue component exceeds green, it does so by less than green exceeds red. This is true even for many natural pastel shades of blue. The exceptions are deep blues, which have already been suppressed as part of the backing, and magentas.

Ultimatte assumes, therefore, that any time the blue component in a color exceeds green by more than green exceeds red, it is because of blue spill contaminating the color. Ultimatte adjusts the values so that blue only exceeds green by the amount green exceeds red (or by holding blue to the level of green if green is equal to or less than red), any unnatural bluish tints will be removed while natural blues will be unaffected.

As mentioned before, Ultimatte may interpret foreground objects with a magenta tint as being contaminated by blue spill and change shades of magenta to shades of red. In this case, you will need to adjust the Gate controls to compensate.

Another problem may arise with some shades of yellow and brown that are contaminated by blue spill. The blue component should be less than green, but Ultimatte suppression will allow blue to equal green, slightly desaturating these colors. In this case, you will need to adjust the Black or Gray Balance controls.

BALANCE PARAMETERS

The Balance parameters can be used to make foreground colors warmer or cooler by overriding the spill/flare suppression logic that ensures that whites in the foreground have equal amounts of red, green, and blue:

- Use the White Balance parameter for light colors.
- Use the Gray Balance parameter for midrange colors.
- Use the Black Balance parameter for dark colors.

The default value is 50 for all three parameters. Set the parameter by eye while viewing the composited image.

WHITE BALANCE: Use the White Balance parameter to adjust light foreground colors with minimal effect on darker colors. It can make foreground whites match the cool/warm tint of background whites.

For example, flare suppression logic can cause blond hair, shot against blue backing, to look white around the edges. Adjusting White Balance can restore a warmer, more natural color to the hair without significantly affecting other colors.

BLACK BALANCE: Use the Black Balance parameter to adjust dark foreground colors with minimal effect on lighter colors. It can make foreground blacks match the cool/warm tint of background blacks.

For example, adjusting Black Balance can eliminate flare from black and brown hair (and some blond hair) with minimal effect on light colors. You can also compensate for the slight desaturation of some colors that the suppression logic can cause.

GRAY BALANCE: Use the Gray Balance parameter to adjust midrange foreground colors with minimal effect on lighter and darker colors. It can make foreground grays match the cool/warm tint of background grays.

For example, used in conjunction with Black Balance, Gray Balance can help compensate for the desaturation of some colors.

GATE PARAMETERS

The Gate parameters, Gate 1 & 3 and Gate 2, enable you to adjust specific color ranges, depending on the backing color used to shoot the sequence: blue, green, or red. (The "gate" terminology, by the way, is a holdover from the original Ultimatte hardware, which included physical gate switches to control this function.)

GATE 1 & 3: The following table illustrates the spill suppression effect of the Gate 1&3 parameter on various color families:

	BLUE SCREEN	GREEN SCREEN	RED SCREEN
	BLUE-CYAN-GREEN	GREEN-CYAN-BLUE	RED-YELLOW-GREEN
100	Blue unaffected	Green unaffected	Red unaffected
50	Blue turns Cyan	Green turns Cyan	Red turns Yellow
0	Cyan turns Green	Cyan turns Blue	Yellow turns Green

Using blue backing as an example, compare the original foreground image and the composite. Look to see if any cyan objects have turned blue, or green objects cyan. If so, you can probably use this parameter to remove the excess blue. The default value is 100.

GATE 2: The following table illustrates the spill suppression effect of the Gate 2 parameter on various color families:

	BLUE SCREEN	GREEN SCREEN	RED SCREEN
	MAGENTA-RED	YELLOW-RED	MAGENTA-BLUE
0	Magenta turns Red	Yellow turns Red	Magenta turns Blue
100	Magenta unaffected	Yellow unaffected	Magenta unaffected

Again using blue backing as an example, compare the original foreground image and the composite. Look to see if any magenta or pink objects have turned red. If so, you can probably use the Gate 2 parameter to restore the true blue component.

The default value for Gate 2 depends on the backing color:

- For blue screens, the default value is 0.
- For green screens, however, the default value is 50. This will prevent hair and skin tones from losing their green component and taking on a purple or red cast.

G: MATTE CHANNEL

This menu becomes active when an optional garbage matte input is connected to the Ultimatte node. It is used to specify which channel of the input to use as the garbage matte, when the image has more than one channel. For more information, see also "Using a Garbage Matte" on p. 224.

H: MATTE CHANNEL

This menu becomes active when an optional holdout matte input is connected to the Ultimatte node. It is used to specify which channel of the input to use as the holdout matte, when the image has more than one channel. For more information, see also "Using a Holdout Matte" on p. 225.

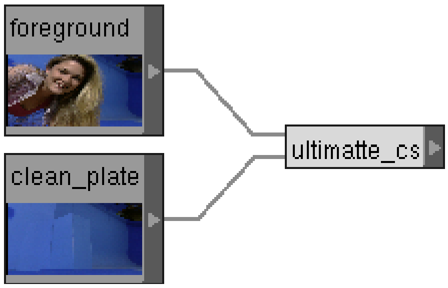
ULTIMATTE CSC (CLASSIC SCREEN CORRECTION) NODE

The Ultimatte CSC (Classic Screen Correction) node enables you to correct flaws in the screen area, or backing, of a foreground image. Flaws can be caused by uneven lighting or seams and patches on the surface of the backing. You can use Ultimatte CSC to transform imperfect bluescreen backings into homogenous fields of blue (or green or red) while retaining shadows and transparency.

You should process imagery with the Ultimatte CSC node before further processing with the Ultimatte GK, Ultimatte, or Ultimatte CC nodes.

The Ultimatte CSC node requires two inputs:

- foreground imagery (top input)
- screen correction frame or clip (bottom input)



15.15 The foreground image is connected to the top input of Ultimatte CSC and the clean plate is connected to the bottom input.

CLEAN PLATE

The screen correction image, commonly referred to as a clean plate, is a clip of the lit backing that has been shot in the absence of foreground objects, such as actors or props. The clean plate is used as the reference for the correction. This input may be a single frame, for locked-off shots, or a sequence of frames that matches the number of frames in the foreground image input.

LOCKED-OFF SHOT

For a locked-off shot (where the camera doesn't move, pan, or zoom during the scene), a single reference frame is adequate. The Ultimatte CSC node will use the single reference frame for comparison with each frame of the foreground image.

TIP:

If you do not have a screen correction clip, you may be able to create a reference frame by combining regions from different frames of the foreground sequence to create a complete reference frame without foreground objects or shadows.

Moving Camera Shot

In a scene where the camera does move, motion control equipment should be used to shoot a correction clip that matches the actual shot exactly. The Ultimatte CSC node will compare each frame of the foreground image to the corresponding reference frame.

How Classic Screen Correction Works

Essentially, to create a corrected foreground image for output, the Ultimatte CSC node must identify, for each pixel in the original foreground image, whether the pixel represents a totally opaque foreground object, a partially transparent foreground object (such as a shadow), or unobstructed backing. In the corrected image

- all pixels identified as unobstructed backing are assigned an identical RGB value,
- opaque foreground objects are assigned the same value that they had in the original foreground image, and
- semi-transparent foreground objects are assigned a value that represents how they would appear over the color to be used for the corrected backing.

To accomplish this screen correction, the Ultimatte CSC node generates temporary mattes for both the foreground image and the reference frame using the process explained in "How Ultimatte Generates Matte Data" on p. 221. Then the node generates a correction frame by subtracting the pixel values of the reference frame from the peak point value.

The value of each pixel in the foreground matte is divided by the corresponding pixel value of the reference matte to generate a correction matte. Each pixel of the correction frame is multiplied by the corresponding pixel of the correction matte. The result is added to the corresponding pixel of the foreground image to create a new, corrected foreground image.

Selecting a Sample Color

The first step to using Ultimatte CSC is to select a sample color Ultimatte will use as a reference in the screen correction process.

1. Display the node in an Image Viewer and select Input Image from the Viewer's Source menu to access the uncorrected image.

2. Use the Sample Color eyedropper in the Node Panel to sample an area that represents the desired color to use for the backing.

The sample color should be in an unobstructed area of backing (no shadows or strands of hair, e.g.) and should be a representative backing color.

The best blue (or green, or red) to select from an uneven backing will depend on the image:

- Choosing too bright a sample can cause darker areas of unobstructed backing to be interpreted as being in shadow.
- Choosing too dark a sample can cause glowing edges around foreground subjects that are against brighter backing areas, and faint shadows and thin transparencies can be lost.

It is usually best to select a bright, representative value that is close to important foreground information, such as near a person's face. Be sure to avoid clicking on fine foreground details, such as smoke, shadows, strands of hair, or mist. Zoom in if necessary before picking.

TIP:

Fill a swatch in the Image Viewer's Color Picker with the sample color and use it to fill the Sample Color parameters in any other Ultimatte nodes you may use to process the image further.

Ultimatte CSC Parameters

The Ultimatte CSC Node Panel parameters are used to select the backing color and adjust the screen correction.

Sample Color

The Sample Color parameter displays a swatch of the sample pixel you select by clicking on an area of backing in the RGB display of the Image Viewer. You can also expand the parameter to access the full color selection controls, which enable you to set RGB or HSV values numerically. (For a detailed description of these controls, refer to "Using the Color Parameters" in chapter 14, p. 168.)

See "Selecting a Sample Color" (p. 233) in the previous section of this node description for information about how to choose the best color.

Matte Density & Black Gloss

The Matte Density and Black Gloss parameters can be used, if necessary, to adjust how Ultimatte interprets what is foreground and what is backing during the screen correction process:

- To increase matte density in light or bright colored foreground objects, increase the Matte Density parameter (the default value is 50).
- To increase matte density in dark, reflective foreground objects, increase Black Gloss (the default value is 0).

If you set the Matte Density or Black Gloss parameters too high, the result may include enhanced visual noise or other unwanted effects. Be sure to check any adjustments you make to these parameters by viewing the result in a composite node.

ULTIMATTE GK (GRAIN KILLER) NODE

The Ultimatte GK (Grain Killer) node removes film grain from the screen area, or backing, of a foreground clip. It is also effective at reducing visual noise such as that introduced by video cameras under low-light conditions.

One of the greatest strengths of Ultimatte is its ability to retain fine details in a composite. However, this means that film grain (or video noise) in the backing area of the foreground image may be composited onto the background and result in a double-grain effect in those areas.

The Ultimatte GK node is designed to eliminate this problem by recognizing and filtering only the backing area of the foreground image. This enables you to avoid softening the foreground subject and eliminating detail. See "How Ultimatte Generates Matte Data" on p. 221 for an explanation of the process Ultimatte uses to recognize backing areas of a foreground image.

Ultimatte GK filters grain pixels in the screen area by averaging the values of surrounding pixels to create replacement values for the pixel currently being processed.

The Ultimatte GK node accepts one input.

USING GRAIN KILLER

The first step is to specify a sample color of the backing area that represents the ideal color for Ultimatte Intelligence to use as a reference. Then you can adjust the default grain filtering parameter values if necessary.

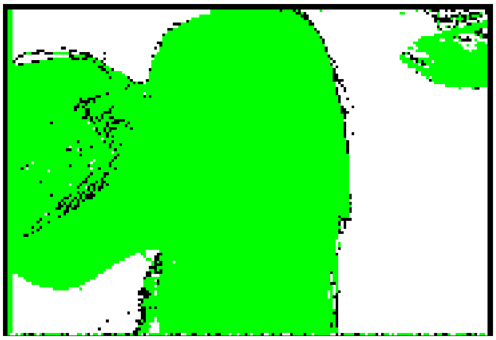
STEP 1: SETTING THE GK SAMPLE COLOR

If the input to the GK node is an Ultimatte CSC node—and this is highly advised, unless you have a foreground image with perfectly even backing—use the same sample color value in the GK node that you used for the Ultimatte CSC node.

If your shot does not require the use of the CSC node, use the eyedropper tool associated with the Sample Color parameter to click an appropriate pixel in the Viewer. See "Selecting a Sample Color" (p. 233) in the Ultimatte CSC node description for guidelines on selecting the best color.

STEP 2: ADJUSTING THE FILTER AREA

Select Filter Area from the Source menu of the Image Viewer to examine the Filter Area matte. This display does not represent a normal image channel to be output from the node. Instead, it shows which parts of the image will be grain filtered.



15.16 Select Filter Area display from the Viewer's Source menu to find out how the Ultimatte GK node will filter each pixel.

Areas defined as solid foreground subject are never filtered by the GK node. These areas appear green. To increase or decrease the subject area, adjust the Matte Density and Black Gloss parameters in the Advanced parameter group.

The backing area, as well as the transition areas between the subject and backing, can be filtered. These areas are black and white in the Filter Area display: white areas will be filtered and black areas will not. To increase the proportion of white pixels in the filter area, use the Screen Filter parameter in the Advanced parameter group.

Ultimatte GK Parameters

Sample Color

The Sample Color parameter displays a swatch of the sample pixel, or pixel area, you select in an area of backing in the RGB display of the Image Viewer. To select the color, click the eyedropper tool associated with the Sample Color parameter to click on an appropriate pixel in the image.

You can also expand the parameter to access the full color selection controls, which enable you to set RGB or HSV values numerically. For a detailed description of these controls, refer to "Using the Color Parameters" in chapter 14, p. 168.

See also "Using Grain Killer" (p. 235) in this node description for general guidelines.

Filter Orphans

Check this box to eliminate stray single pixels, or "orphans," from the screen area. These orphans are pixels that failed to be filtered by the node.

Advanced Parameters

The Advanced parameter group includes the following parameters, which are best used in conjunction with the Filter Area display (see *Fig. 15.16*).

Filter Passes

The Filter Passes parameter enables you to specify the number of times the image data will pass through the filter. The default value is 1; however, multiple passes may be necessary. If the backing still appears mottled after the initial pass, try increasing the setting.

The filtering process used by the node averages the color values of surrounding pixels to create replacement values for the grain pixels being filtered. Pixels are averaged 25 at a time, in a 5 x 5 matrix, as long as the matrix does not encroach on transition or foreground areas. For pixels too close to a transition edge for an unobstructed 5 x 5 square to be defined, 9 pixels (in a 3 x 3 matrix) are averaged.

MATTE DENSITY & BLACK GLOSS

The Matte Density and Black Gloss parameters enable you to control what portion of the image is defined as solid subject area. (The subject area is green in the Filter Area display in the Image Viewer.) This is important because the areas defined as totally opaque foreground objects will not be grain-filtered.

For example, if you boost the density values in GK Matte to eliminate holes in the matte, the portion of the image that is defined as solid subject, and will therefore not be filtered, is increased accordingly:

- Matte Density affects light or bright colored foreground objects more.
- Black Gloss affects dark, reflective foreground objects more.

Matte Density defaults to 50, while Black Gloss defaults to 0. Both parameter ranges are 0–100.

As the Ultimatte GK node does not generate an output matte (the Ultimatte node is used for that purpose), you can define transition areas as part of the solid subject area if you do not want them to be grain filtered. On the other hand, you can erode the subject area to ensure filtering of transition areas. It all depends on the nature of the image.

SCREEN FILTER

The Screen Filter parameter controls how much of the area defined as backing—the black and white pixels in the Filter Area display—will be grain filtered. White areas in the Filter Area display will be filtered; black areas will not.

If too much of the backing area is filtered, you will begin to filter the edges of the foreground subject, resulting in a loss of detail. To reduce the area of backing that will be filtered (to turn some of the white pixels black), start by expanding the Screen Filter parameter to access the individual color controls.

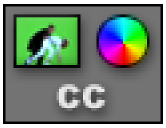
Decrease the Value parameter, and watch the change to the Filter Area display, which will update accordingly. The white pixels in the transition areas (next to the green subject areas) will turn black first. If you continue to decrease the Value parameter, more pixels in the rest of the backing will turn black.

To ensure that these values are set properly for your shot, be sure to review the result (especially in the final composite image) and judge for yourself.

NOTE:

You can use any of the individual color parameters in Screen Filter to adjust the Filter Area display. Experiment as much as you like; you can always use the Animation menu for the Screen Filter parameter to reset it to project defaults and start again.

ULTIMATTE CC (COLOR CONTROL) NODE



The Ultimatte CC (Color Control) node enables you to match the color levels of an image to a reference image. Typically, the Ultimatte CC node is used to adjust the output from an Ultimatte node to match the background image over which it will be composited.

NOTE:

The Ultimatte CC node only affects the input image; there is no change to the reference image. Additionally, Ultimatte CC does not affect spill or flare. (Any spill or flare problems should be corrected in the Ultimatte node before using Ultimatte CC.)

One example of how Ultimatte CC is used is the stereotypical "twins" scene, in which a single actor plays dual roles. In such a case, you may be faced with two clips: one of the actor against a background, and one of the actor turned in the opposite direction, speaking twin to self, filmed in front of a bluescreen.

It is likely that each clip was shot under different lighting conditions, and you may therefore need to adjust the flesh tones or clothing colors of one image to match the other. The Ultimatte CC node will enable you to make the colors of two such images compatible.

The Ultimatte CC accepts one input—it is not necessary to connect the reference image to the CC node.

USING COLOR CONTROL

Ultimatte CC is normally used to match the color of the input image to a reference image. However, it can be used to adjust color attributes of an image without using a reference image. To do so, skip step 1 and start with step 2.

STEP 1

The Color Control Node Panel includes parameters for a Sample Color value, which should be picked from the input image to be adjusted, and for a Reference Color value, which should be picked from the image you want to match.

Select representative areas from each image:

- To match highlights, for example, sample the pixels with the highest value in each image.
- To match flesh tones, on the other hand, you could sample an area in the same part of the face in each image.

Step 2

Then use the parameters in the Controls group to correct the output image:

- Adjust Level to match highlights.
- Adjust Gamma to match midtones.
- Adjust Black to match black levels.
- Adjust Saturation to match saturation levels.

TIP:

For the best result, fine-tune your color adjustments based on viewing the final composited image, rather than the foreground image alone.

Ultimatte CC Parameters

Sample & Reference Parameters

Both parameters work the same way: use the eyedropper tool to sample the color values in a representative area of an image, which will fill the parameter swatch with that color:

- Use the Sample parameter to sample a pixel area in the input image, which you want to correct to match the reference image.
- Use the Reference parameter to sample a pixel area in the reference image (the image that contains the color values you want to match).

Controls Parameter Group

Expand the Controls group to access the Level, Gamma, Black, and Saturation parameters. These parameters are master controls that adjust all three color channels equally. However, each of them can be expanded to reveal separate Red, Green, and Blue channel parameters if you need to adjust a channel individually. The range of all parameters in the Controls group is 0–100.

Level

The Level parameter is used to adjust brightness in the highlights; pure blacks are unaffected. This parameter is effectively a gain control on the color components because it sets a multiplier value for each pixel value. The default value is 80.

Gamma

The Gamma parameter is used to adjust contrast in the midrange tones; pure blacks and whites are unaffected:

- Increase Gamma to boost the contrast of darker colors.
- Decrease Gamma to boost the contrast of brighter colors.

When you increase the dynamic range of color values at one end of the range, colors at the opposite end are correspondingly compressed. The default value for the Gamma parameter is 50.

Black

The Black parameter is used to adjust black levels; pure whites are unaffected. If the Black parameter is set below 50 (which is the default value), the darkest color components in the image will clip at 0.

NOTE:

Although you can expand the Black parameter to adjust the Red, Green, or Blue channels individually, impure whites (less than 100 percent saturated) can be noticeably affected if all three channels are not adjusted equally.

Saturation

The Saturation parameter is used to control saturation by proportionally replacing each color component with its luminance value. If you set the value to zero, the result will be a grayscale image. The default value is 80.

chapter 16

COLOR NODES

The Color menu provides an array of color filters that give you the option to perform multiple operations on an image within a single node (Color Correct), or to select a single-purpose node for each type of color adjustment (Brightness, e.g.), based on your needs and preference.

GUIDELINES FOR CHOOSING THE RIGHT COLOR NODE

Some of the nodes in the Color menu perform a single, specific function. For example, the Video Safe node is used strictly to conform the colors of an image to television broadcast standards by converting “illegal” colors to video-safe colors. Similarly, the Monochrome node is used strictly to desaturate an image, Invert to invert color values, and so on.

The Color Curves node, on the other hand, which lets you manipulate color distribution curves directly, can be used to adjust brightness, contrast, color balance, and other image characteristics. When should you choose Color Curves rather than, say, a Brightness or Contrast node?

The answer will usually depend on the level of precision you need. Take, for example, an image that needs more contrast. The Contrast node is the fastest, easiest way to increase the contrast—just increase the parameter value. But if you need to increase contrast only in the shadows, and if you want the effect to fall off nonlinearly, use the Color Curves node instead, or the all-purpose Color Correct node, which is described next.

SWISS ARMY NODE

In many cases, you may need to perform several types of color adjustment on an image, or you may need to experiment with several techniques until you get the result you want. This is when the Color Correct node is the best choice.

The Color Correct node is the “Swiss army knife” of color correction nodes—the functionality of most of the other color nodes is included in it. In fact, Color Correct is the most comprehensive and powerful color adjustment node in RAYZ. It can be used to make a single adjustment or as many types of adjustment as may be necessary for a particular image.

The parameters relevant to each operation are adjusted in a separate, selectable layer in the Color Correct Node Panel. You can turn off any layer temporarily and you can reorder the layers to change the order in which each adjustment is applied to the image. This node is the best choice when multiple types of correction are needed, or when you want to experiment.

In addition, the Color Correct node offers the “BCG” layer, in which brightness, contrast, and gamma values can be adjusted separately in the shadows, midtones, and highlights. You can even change the definition of shadows, mids, and highs to suit any image.

COMPUTATIONAL EFFICIENCY

RAYZ concatenates node operations and calculates them at floating point accuracy regardless of the bit depth of the image. Values are not clamped on node output, which means that it is not necessary to use the Color Correct node strictly to optimize multiple color correction operations and prevent data loss.

NOTE:

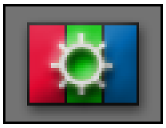
Certain operations, by their nature, are exceptions to the rule, such as Channel Swap and Monochrome. Whenever you add a node, or a Color Correct layer, of one of these types you could potentially lose some color data.

SELF-DOCUMENTING NODES

The single-purpose color nodes do offer the advantage of being "self-documenting." If you are examining a node network and you see a Brightness node, for example, you can immediately tell what type of operation is being performed.

With Color Correct, on the other hand, you would have to select the node and examine the layers in the Node Panel. However, you can always create an underlay for a Color Correct node and type a note that explains how it is being used. For more information about this feature of the Worksheet, see the section on "Adding Underlays to the Worksheet" in chapter 5 (p. 60).

BRIGHTNESS NODE



The Brightness node is used to adjust the brightness of an image, or individual image channel. The node includes an Offset parameter that can be used to shift the distribution curve that defines the brightness of the output.

The Brightness node performs the following computation, where "V" represents the input value:

(V - Offset) * Brightness

The Brightness node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

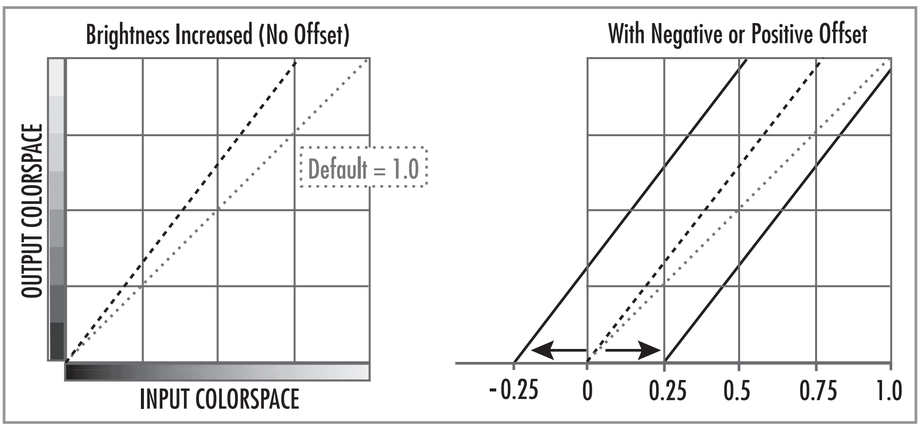
NOTE:

You can use the "Color Curves Node" (p. 257) when you need the increased precision afforded by manipulating brightness curves directly.

ADJUSTING BRIGHTNESS

The **Brightness parameter** can be thought of as specifying the angle of a distribution curve that controls the remapping of values from the input image to the output—the higher the parameter value, the steeper the curve, and the brighter the output (and vice versa). This means that the rate of change is not even across the colorspace. Pixels with higher RGB values are affected more than the darkest pixels.

The **Offset parameter**, on the other hand, can be thought of as shifting the distribution curve back and forth across the input (horizontal) axis without changing its angle. The offset has the effect of increasing or decreasing pixel values evenly across the tonal range. Positive offset values shift the brightness curve down; negative offset values shift it up.



16.1 Increasing the Brightness parameter affects the highest image values the most, while the Offset parameter affects the entire tonal range equally.

Brightness Parameters

Brightness

Each pixel in the input image is multiplied by the Brightness parameter value to get the corresponding output value, assuming an offset has not been specified.

The range of the Brightness slider is 0 to 2, although the upper end of the range is not constrained. The default value of 1.0 results in no change to the image, while values less than 1 decrease the brightness and values greater than 1 increase it.

If necessary, you can expand the parameter to access the channel controls, which work exactly like the master Brightness parameter but control each channel independently.

Offset

The Offset parameter value is subtracted from each pixel in the input image before it is multiplied by the Brightness parameter value. This means that a nonzero value in the Offset parameter will affect the output image even when the Brightness parameter is left at its default value.

Because the offset is subtracted from the pixel value,

- positive offset values darken the output and
- negative offset values brighten it.

Offset values are expressed by default in units based on the range of the input image colorspace: -255 to 255 for 8-bit; -65535 to 65535 for 16-bit; -1 to 1 for floating point. The default value is 0 (no offset), and the actual range of the parameter is unconstrained.

TIP:

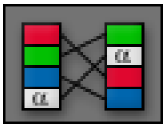
The Offset values can always be displayed in floating point units by checking the Float Display option in the parameter Animation menu, which is located on the right side of the parameter's slider bar.

If necessary, you can expand the parameter to access the channel controls, which work exactly like the master Offset parameter but control each channel independently.

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, the RGB channels of the input image are selected. To select or deselect a channel, press the button labeled with corresponding channel letter (such as A for Alpha) to toggle it to the opposite state.

Channel Swap Node

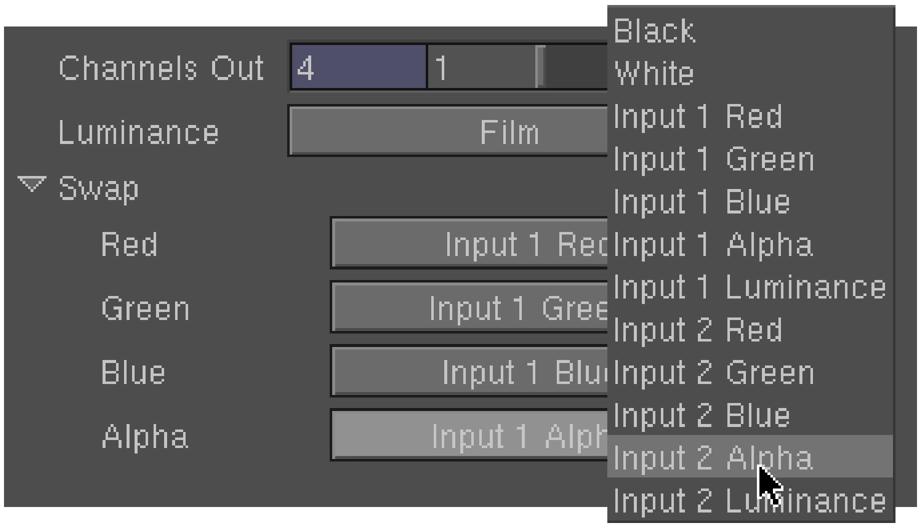


The Channel Swap node is used to redefine the channel assignments of the input(s). You can add channels, delete channels, swap channels, or fill a channel with black, white, or luminance.

The Channel Swap node accepts one or two inputs. The second, optional input enables you to combine channel data from two separate images in the output image:

- If the inputs are different bit depths, the node promotes the channels in the lower bit depth input to the higher.
- If the inputs are different sizes, the node uses the size of the top input for the output. This means that if the bottom input is larger, any of its channels that are used in the output will be cropped.

Channel Swap Parameters



16.2 Channel Swap node with two inputs: Input 1 alpha channel is being replaced by the Input 2 alpha.

Channels Out

Use the Channels Out parameter to specify the number of channels the output image will contain, from one to 10 (the default is four channels). A channel menu will appear in the Swap parameter group for each output channel specified.

Luminance

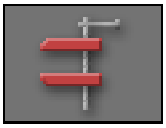
The Luminance menu is used to specify the type of luminance to be calculated if you fill an output channel with luminance: Film, NTSC Video, or PAL Video. For the formulas used in the calculation, refer to *Appendix A: How RAYZ Computes Luminance Values* (p. 439).

Swap

The Swap parameter group updates dynamically, based on the value you specify in the Channels Out parameter, to provide a menu for each channel of the output. Each Channel Swap menu lists all of the available input image channels, as well as Black, White, and Luminance. Just select the one you want to use for that output channel.

Most commonly, Channel Swap is used to add an alpha channel to an RGB image. You can select Black, White, or Luminance, or you can select any input channel you wish to use as the alpha. By connecting two nodes to Channel Swap, you can select the alpha channel from one input to use in the other.

CLAMP NODE



The Clamp node is used to clamp the colorspace of an input image to the range you specify. All pixel values higher than the specified Max Value or lower than the specified Min Value are clamped; pixel values within the Clamp range are not changed by the node.

CLAMP PARAMETERS

MIN VALUE

Use this parameter to set the minimum value of the output range. The default value is 0.

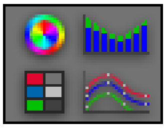
The master Min Value control adjusts all channels equally. To set the minimum value for a particular channel, expand the Min Value group to access the individual channel controls.

MAX VALUE

Use this parameter to set the maximum value of the output range. The default value is 1 (which is equivalent to 255 in 8-bit; 65535 in 16-bit).

The master Max Value control adjusts all channels equally. To set the maximum value for a particular channel, expand the Max Value group to access the individual channel controls.

Color Correct Node



The Color Correct node is a comprehensive tool for performing a wide variety of adjustments. It provides a number of common color manipulation tools in one layer-based interface. The Color Correct node enables you to do any or all of the following:

- adjust brightness, contrast, and gamma
- adjust shadows, midtones, and highlights
- perform hue-based color adjustments
- adjust tonal or color values by manipulating curves
- operate on individual channels
- swap channels
- invert colors
- desaturate colors
- simulate printer lights and f-stops

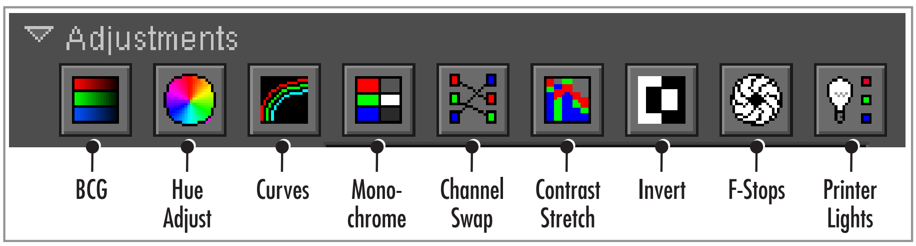
The Color Correct node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

Color Correct Parameters

In the Color Correct node, you create a separate layer for each type of adjustment you wish to make. Each layer features its own set of parameters, which you adjust to achieve the desired effect. You can add as many layers as you need, and any layer can be deleted, reordered, or temporarily disabled.

Adjustment Layer Buttons

Click any adjustment button in the Color Correct Node Panel to create a correction layer for that operation. You can expand any layer to access its parameters, which are described in the following sections covering each type of adjustment layer.



16.3 Click one of the layer buttons to create a correction layer of that type.

Each layer includes a common set of controls to disable the layer temporarily, delete the layer, and reorder the layer in the Node Panel. For more information, see also "Dynamic Parameter Groups" in chapter 7, p. 93.

CHANNEL SELECT

Use this parameter to specify which channels of the input image will be processed by the node. By default, the RGB channels of the input image are selected. To select or deselect a channel, press the button labeled with corresponding channel letter (such as A for Alpha) to toggle it to the opposite state.

BCG (BRIGHTNESS, CONTRAST, GAMMA) LAYER

The BCG layer enables you to adjust Brightness, Contrast, and Gamma values and to set a Brightness Offset or Contrast Pivot.

For basic information about these operations, such as the formulas used to compute them and graphs illustrating the effect of an offset, refer to the descriptions of the individual nodes: Brightness Node (p. 246), Contrast Node (p. 261), and Gamma Node (p. 264). The implementation of these operations in the BCG layer is described next.

16.4 The BCG menu is used to specify whether parameters apply to entire image (Master) or only to Shadows, Midtones, or Highlights.

BCG MENU

The Brightness, Contrast, Gamma, Offset, and Pivot parameters apply to whatever portion of the tonal range is currently selected in the BCG menu. By default, these parameters affect the entire tonal range of the image because the Master control is selected in the BCG menu; however, you can also select Shadows, Midtones, or Highlights.

This means, for example, that you can adjust the overall brightness level while Master is selected in the BCG menu, and then switch the menu to Shadows to further adjust the dark areas only.

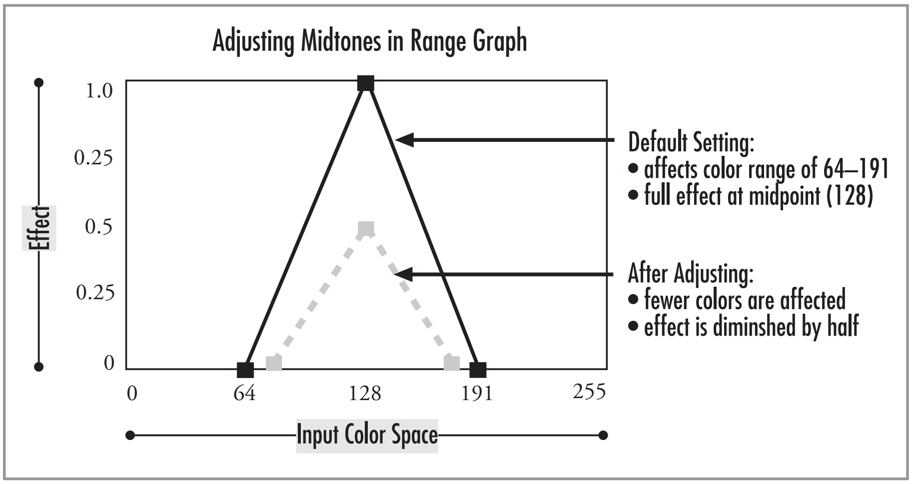
BCG PARAMETERS

All of the BCG parameter values are set by default at the value that represents no change from the input image, which for Brightness, Contrast, and Gamma is 1. Set any parameter by typing a value in the field or using the tuner to set the value. To adjust an individual image channel, expand the parameter to access the channel-level controls and adjust them in the same way as the image-level parameter.

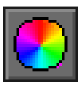
TONE RANGE

The BCG layer provides a Tone Range graph in which you can define what tonal values should be considered shadows, midtones, or highlights and control the magnitude of the effect. Any adjustments you make to the BCG parameters in the Shadows, Midtones, and Highlights will apply to the range of values specified by the corresponding curve in the graph.

To use Tone Range, check the Shadows, Midtones, and/or Highlights boxes to display the curves for in the graph. Each curve can be adjusted by dragging a control point to reposition it. You can also add and delete points and apply different functions to the distribution curves, in the same way you would manipulate animation curves, as described in *Chapter 8: Using the Curve Editor* (p. 105).



16.5 This example graphs the effect of reducing the range of values defined as Midtones and lowering the magnitude of any adjustment made to them.



HUE ADJUST LAYER

The Hue Adjust layer enables you to perform hue-based color adjustments using the HSV color model. This layer is identical to the "Hue Adjust Node" (p. 265).

HUE

The Hue parameter provides a color wheel to rotate the color distribution vectors of the image around the axis of the colorspace; that is, to shift the hue. The inner circle of the wheel represents the input data.

You can perform the hue shift visually by dragging the color wheel, or you can type a value representing the degree of color shift into the Hue field. A value of 0 (or 360) represents no shift, while a value of 90, for example, would shift pure red (0) to green (90) and green to cyan (180).

SATURATION

The Saturation parameter enables you to adjust the chrominance level in a range of 0 to 2, where the default value of 1 represents no change to the input image.

VALUE

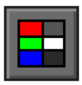
The Value parameter enables you to adjust relative intensity, or brightness, in a range of 0 to 2. The default value of 1 represents no change, 0 will result in a black image, and 2 will result in “blown out” colors.

CURVES LAYER

The Curves layer enables you to modify the color attributes of an image by editing curves in a graph display. The horizontal axis of the graph represents the input values, and the vertical axis represents the output values to which they are mapped.

You can adjust the entire image using the Master curve, or any individual channel. To display a curve for editing, check the associated box: Master, Red, Green, Blue, Alpha, and/or Other.

The Curves layer in Color Correct is identical to the Color Curves node interface. Refer to the description of the “Color Curves Node” on p. 257, which explains the use of color curves in more detail.



MONOCHROME LAYER

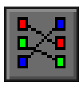
The Monochrome layer is used to desaturate an image.

AMOUNT

The Amount parameter specifies the level of desaturation, from 0 (no desaturation) to 1 (total). By default, the Monochrome layer totally desaturates the image.

LUMINANCE

The Luminance menu is used to specify whether Film, NTSC, or PAL luminance should be used in the calculation. The formulas used are given in *Appendix A: How RAYZ Computes Luminance Values* (p. 439).



CHANNEL SWAP LAYER

This layer enables you to add or delete channels, swap channel values, or fill any channel with black, white, or luminance. It works exactly like the “Channel Swap Node” (p. 248), with the exception that the Channel Swap node accepts a second input to be used when you want to swap channels between two different node images.

To use the Channel Swap layer, select the number of channels you want the output to have. Then use the menu that is automatically generated for each output channel to assign an input channel to it.

Contrast Stretch Layer

The Contrast Stretch layer is used to change the contrast of an image or image channel by adjusting the minimum and maximum values that define the tonal range of the input or output colorspace.

You can type new values into the parameter fields or use the sliders to adjust the default values. You can also specify RGB values by expanding each parameter to access the individual channel controls, or by using the eyedropper tool associated with the parameter to sample an image pixel or pixel area in the Viewer.

Increasing Contrast: White and Black In

To increase the contrast of the image, *decrease the range of the input* colorspace by adjusting the input black (minimum) and white (maximum) values. The resulting smaller range of input values will be remapped to the larger output range, effectively expanding, or stretching, the input.

Decreasing Contrast: White and Black Out

To decrease the contrast of the image, *decrease the range of the output* colorspace by adjusting the output black (minimum) and white (maximum) values. The input values will be remapped to the smaller output range, effectively compressing the input.

Using the Contrast Stretch Eyedroppers

Set the Image Viewer's Source menu to Input Image to pick input image values. Then click the eyedropper tool for a parameter and use it to sample the image (see the description of the "Color Picker Tools" in chapter 14, p. 168 if you need instructions):

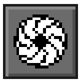
- The *maximum* value sampled per channel will used in the White In and White Out parameters.
- The *minimum* value sampled per channel will be used in the Black In and Black Out parameters.

Expand the parameter group to see the resulting values in the individual Red, Green, and Blue channel parameters, as the master parameter value is not modified when using the eyedropper.

Invert Layer

The Invert layer automatically inverts the color values of any or all channels in an image, effectively creating a negative of the image, by subtracting the current pixel value by the maximum pixel value.

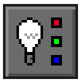
The only parameter in the Invert layer is **Channel Select**, which is used to specify which channels should be inverted. The Channel Select parameter in the Invert layer works exactly like the node-level Channel Select parameter, but its settings apply only to the invert operation.



F-STOPS LAYER

The F-Stops layer enables you to adjust the exposure of an image using units equivalent to photographic stops. This layer is equivalent to using the “F-Stops Node” (p. 263).

Increasing the **F-Stops** parameter value by one stop will double the values at each pixel. This effect is cumulative; bringing the value up two stops will increase the amount of light by a factor of four. Similarly, bringing it down one stop will reduce values by a factor of two.



PRINTER LIGHTS LAYER

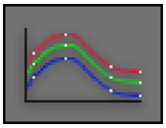
The Printer Lights layer enables you to control the exposure of an image using units that represent printer lights. This layer is equivalent to using the “Printer Lights Node” (p. 272). The node description contains more detailed information on the operation.

LIGHTS PER STOP: This parameter specifies how many Printer Lights are equivalent to one full stop. This parameter should be set to match the rate used by the color lab being referenced, usually 8 or 6.

LIGHTS: This parameter specifies the number of printer lights to use. The default value of 25 represents no change to the image.

The Lights and Lights per Stop parameters can be expanded to access parameters that adjust each channel individually.

Color Curves Node



The Color Curves Node Panel provides a graph in which you manipulate a curve that maps the color distribution from the input image to the output. This gives you more precise control than applying a single level of adjustment to all pixels equally.

The Color Curves node is commonly used to adjust contrast based on the tonal distribution of the input or to remove color casts by adjusting an individual color channel.

The Color Curves node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see “Using Mask Inputs” in chapter 7, p. 102.

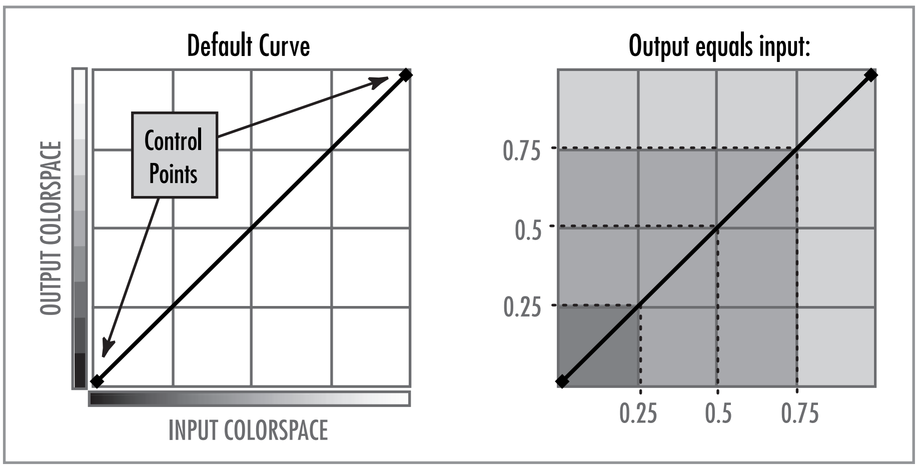
Manipulating Color Curves

The color curves are manipulated in the same way as animation curves in the RAYZ Curve Editor, except that the horizontal axis of the color curve graph represents the input colorspace instead of time. The vertical axis represents the output colorspace.

A row of checkboxes at the top of the graph in the Color Curves Node Panel controls which curves are displayed in the graph. The Master curve controls tonal values for the entire image, and the color channel curves control individual channel values.

16.6 Color Curve selectors in the Node Panel.

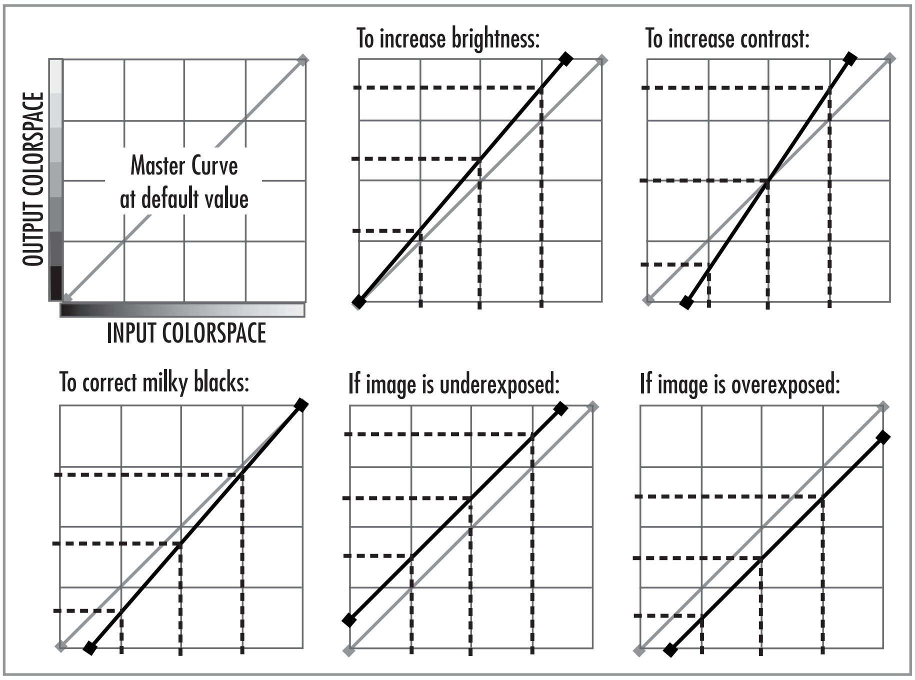
The default curve is set to evenly map each input image value to the output image, resulting in no change to the image, as shown in *Fig. 16.7*.



16.7 The default curve is linear, with a control point at the minimum and maximum value for the colorspace.

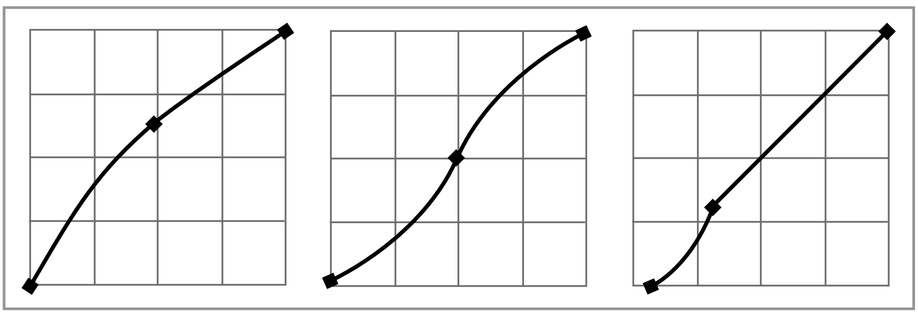
Making Tonal Adjustments

The angle, or slope, of the curve affects how tonal values are redistributed. Color corrections can be made by repositioning the control points representing maximum and minimum. The examples in *Fig. 16.8* provide some general guidelines.



16.8 Examples illustrating the effect of various modifications to color curves.

In many cases, however, you will want to take advantage of the additional precision offered by adding control points to a curve and by applying functions that specify a nonlinear distribution of values. You can add as many points as you want, and different functions can be applied to each curve segment.



16.9 Examples of color curves with control points and functions modifying the default (linear) curve.

Working with Points

- Ctrl-click on a line to add a new control point.
- Drag a point to reposition it.
- Select a point and press the Delete key to delete it.

Working with Curves

- Click anywhere on a line to select the entire curve.
- Drag one of the selected points to move the entire curve.
- Right-hold on a line to access a popup menu of interpolation functions. Select one from the menu list to apply it to the curve segment.

The functions in the popup menu are the same as those in the Curve Actions menu accessed within the RAYZ Curve Editor. For more information about each function, see the section on "Controlling Interpolation of Curve Values" in chapter 8 (p. 109).

Using Curves to Adjust Color Balance

The Color Curves node can also be used to adjust individual color channels. The procedure is the same as that described above for "Making Tonal Adjustments" (p. 258), except that you work on the Red, Green, and/or Blue curves instead of the master curve.

Most commonly the Color Curves node is used to correct an imbalance in one color channel. For example, if an image has a bluish color cast, you can adjust the curve for the Blue channel.

You can also adjust color channel curves to simulate the color temperature of another sequence that was shot under different lighting conditions.

Color Curves Parameters

In addition to the curve graph, which is described in the previous section on "Manipulating Color Curves" (p. 257), the Color Curves Node Panel includes the Curve Selector checkboxes and the Channel Select buttons.

Curve Selectors

The checkboxes at the top of the graph control which curve is visible in the graph. A curve must be visible to be edited. You can check any or all of the available curves, which include the master curve and the individual channel curves.

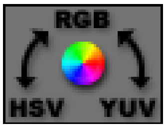
Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, the RGB channels of the input image are selected. To select or deselect a channel, press the button labeled with corresponding channel letter (such as A for Alpha) to toggle it to the opposite state.

NOTE:

You can display and adjust the curve for any channel in the input image, even when it is deselected in this parameter. As long as the channel is deselected in Channel Select, however, the adjustment made to the curve will not be processed by the node until the channel is selected again.

Colorspace Node



The Colorspace node is used to convert imagery from one colorspace to another: RGB, HSV, YIQ, YUV, or CMYK.

The Colorspace node accepts one input, which may be any number of channels. The Alpha channel, if any, is also converted by the node.

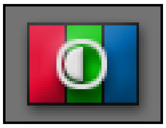
Colorspace Parameters

Conversion

Use this menu to specify the type of conversion:

- RGB to HSV
- HSV to RGB
- RGB to YIQ
- YIQ to RGB
- RGB to YUV
- YUV to RGB
- CMYK to RGB

CONTRAST NODE



The Contrast node is used to vary the contrast of an image, or image channel, effectively compressing or expanding the tonal range. You also have the option of adjusting the point around which the distribution curve pivots to define the contrast value.

The Contrast node performs the following computation:

V = ((V - Pivot) * Contrast) + Pivot

The Contrast node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see “Using Mask Inputs” in chapter 7, p. 102.

NOTE:

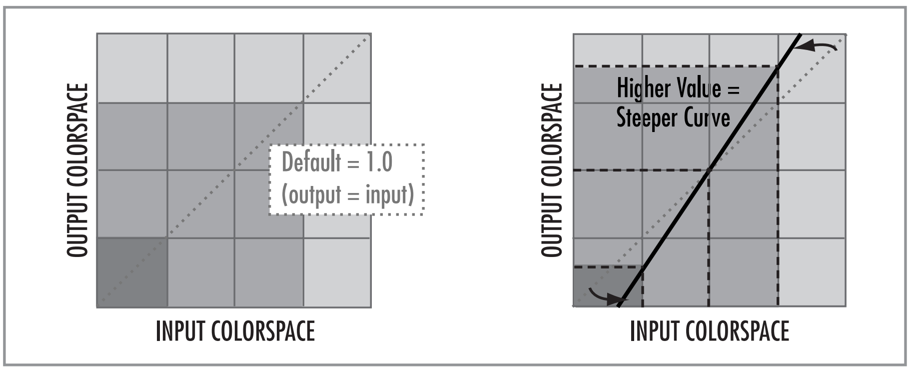
The “Color Curves Node” (p. 257) can be used to adjust image contrast when you need to manipulate the shape of the curve directly, as when you want to apply a nonlinear distribution function to the curve.

CONTRAST PARAMETERS

CONTRAST

The Contrast parameter range is unconstrained, with the default value of 1.0 resulting in no change to the image.

Changing the contrast value changes the slope of the distribution curve that plots how the input values are remapped to the output. Higher values, which steepen the curve, increase contrast; lower values, which produce a shallower curve, decrease it.



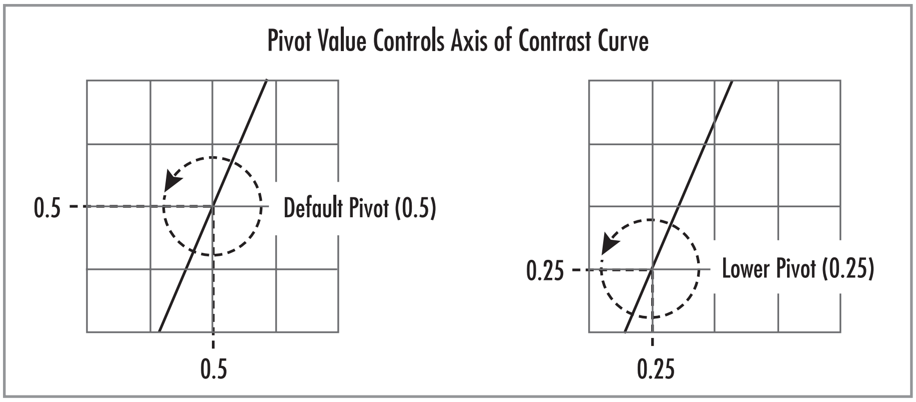
16.10 The Contrast value controls the angle of the distribution curve. Higher values create a steeper curve, which stretches the midtones while compressing the lows and highs.

If necessary, you can expand the parameter to access the channel controls, which work exactly like the master Contrast parameter but control each channel independently.

PIVOT

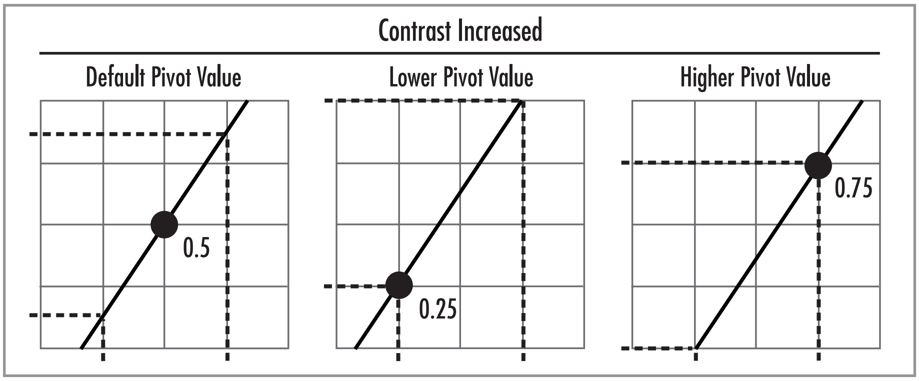
The Pivot parameter controls the position of the axis about which the contrast curve will pivot. The Pivot value only affects the output image when the Contrast value is adjusted to a non-default value.

16.11 The Pivot value controls the axis of the Contrast curve, as illustrated here using a very steep Contrast curve.



A lower or higher pivot value has the effect of shifting the tonal distribution up or down to an extent determined by the Contrast parameter value (see *Fig. 16.12*).

16.12 Lowering the pivot shifts the midtones up, at the expense of the highest image values; raising the pivot shifts the midtones down, at the expense of the darkest image values.



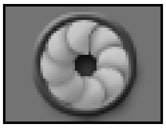
The Pivot value is expressed in units based on the range of the input image colorspace: 0–255 for 8-bit; 0–65535 for 16-bit; 0–1 for floating point. By default the pivot point is centered in the colorspace, which would be a value of 0.5 in floating point.

If necessary, you can expand the Pivot parameter to access the channel controls, which work exactly like the master Pivot but control each channel independently.

CHANNEL SELECT

Use this parameter to specify which channels of the input image will be processed by the node. By default, the RGB channels of the input image are selected. To select or deselect a channel, press the button labeled with corresponding channel letter (such as A for Alpha) to toggle it to the opposite state.

F-Stops Node



The F-Stops node simulates the process of using photographic stops to adjust exposure levels. When you need to adjust an image using f-stops as the unit of measure, perhaps because a DP has asked you to "bring it up a quarter stop," you can use this node without having to convert the requested adjustment into the equivalent parameter values used in other nodes.

The F-Stops node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

F-Stops Parameters

F-Stops

The F-Stops parameter has a range of 4 stops; that is, you can increase or decrease exposure by a maximum of 2 full stops. The range is -2 to 2, with the default value of 0 representing no change to the input image.

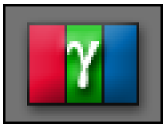
To decrease the image by a quarter stop, for example, enter -0.25 in the parameter field. To increase the image a half stop, enter 0.5.

If necessary, you can expand the parameter to access the individual channel parameters, which work exactly like the master F-Stops parameter but control each channel independently.

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, the RGB channels of the input image are selected. To select or deselect a channel, press the button labeled with corresponding channel letter (such as A for Alpha) to toggle it to the opposite state.

GAMMA NODE



The Gamma node enables you to correct the gamma of an image. Gamma is an objective measure of the contrast in an image, equal to the slope of the characteristic curve. The characteristic curve, or gamma curve, is a plotted curve that illustrates the change in density of a negative as exposure increases.

The Gamma node performs the following computation (where V is the color value of a pixel):

$$V = V^{1.0/Gamma}$$

The Gamma node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

GAMMA NODE PARAMETERS

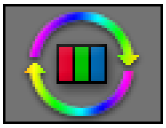
GAMMA

Set the Gamma parameter by typing a value into the field or using the slider to set the value in a range of 0–3 (the actual range is unconstrained). The default value is 1, which represents no change to the input image.

CHANNEL SELECT

Use this parameter to specify which channels of the input image will be processed by the node. By default, the RGB channels of the input image are selected. To select or deselect a channel, press the button labeled with corresponding channel letter (such as A for Alpha) to toggle it to the opposite state.

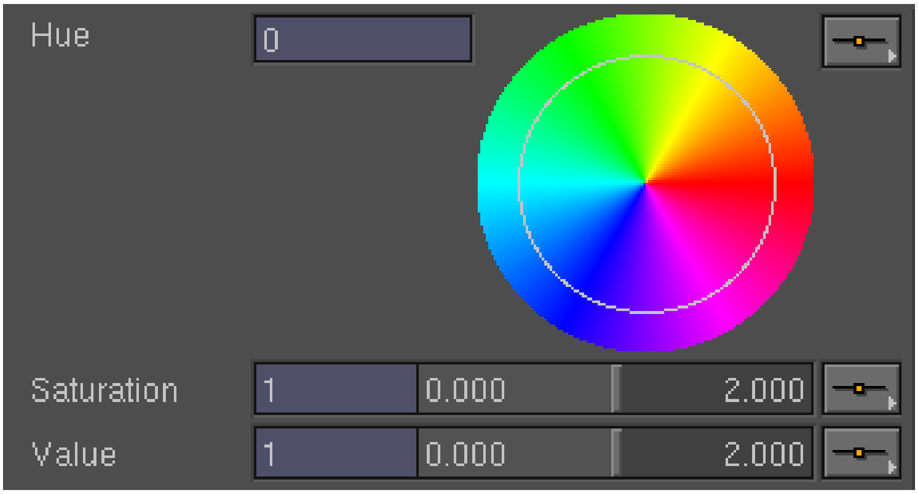
Hue Adjust Node



The Hue Adjust node enables you to perform hue-based color adjustments to an image using the HSV (Hue, Saturation, Value) color model.

The Hue Adjust node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

Hue Adjust Parameters



16.13 Hue Adjust parameters in the Node Panel.

Hue

The Hue parameter is used to specify the hue shift for the input by rotating the color distribution vectors of the image 360° around the axis of the colorspace. The default value of the Hue parameter is 0, which represents no change from the input data.

You can perform the hue shift visually by dragging the color wheel (the inner circle of the wheel represents the input data), or you can enter a value representing the degree of color shift into the Hue field. For example, a value of 90 would shift pure red (0) to green (90) and green (90) to cyan (180).

Saturation

The Saturation parameter is used to adjust the chrominance level of the image in a range of 0 (complete desaturation) to 2. The default value is 1, which represents no change to the saturation of the input image.

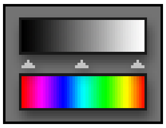
Value

The Value parameter represents the relative intensity, or brightness, of the color data. The range is 0 (which would result in a black image) to 2 (blown out colors), with the default value of 1 representing no change to the input image.

TIP:

It is helpful to remember that two different hues with the same numerical brightness setting may not have the same perceived brightness. Also, the eye is more sensitive to changes in brightness at lower levels; for example, the difference in perceived brightness may be as great between numerical values of .10 and .11 as between .50 and .55.

INDEXED COLOR NODE



The Indexed Color node is used to add a color wash, or tint, to a grayscale image. The color will be applied based on the distribution of luminance values in the input image to preserve the tonal gradations, textures, and shading of the original. You specify the color or colors to map to the luminance gradient, and those color values are interpolated linearly across the range.

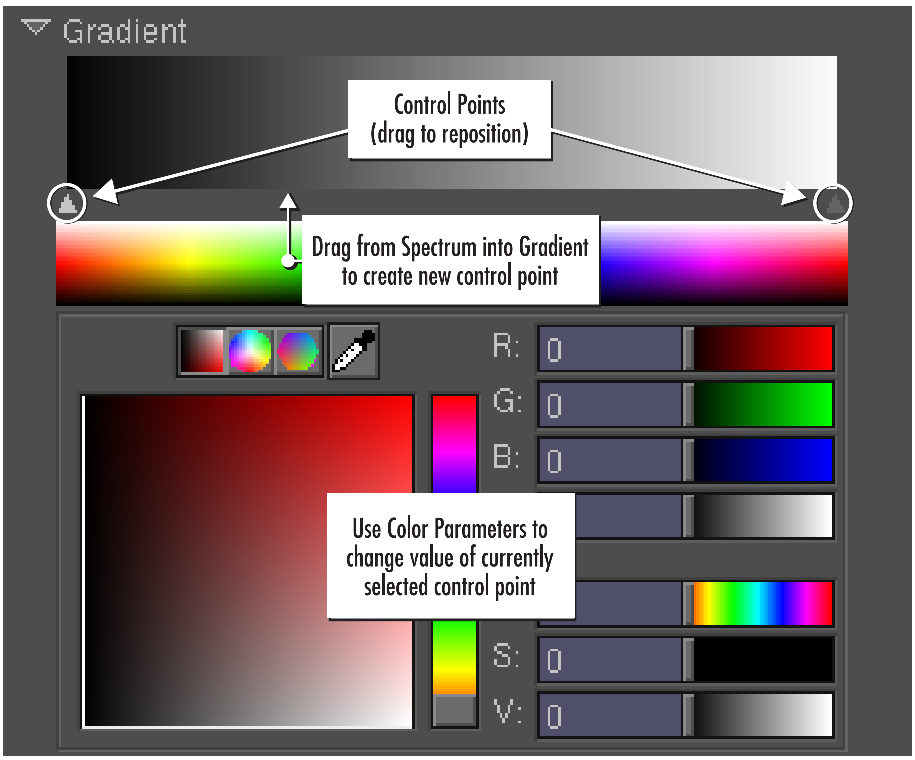
The input image may be monochromatic or color. For a color input, however, the node will generate a monochromatic image by calculating the luminance values of the RGB image.

TIP:

The Indexed Color node is often used with the optional mask input, which will limit the colorization to a specific area when you want to change the color of a particular object in the scene, rather than the entire scene itself. For more information, see "Using Mask Inputs" (ch. 7, p. 102).

USING INDEXED COLOR

The Gradient parameter includes the gradient strip, which represents the tonal range of the image, and the color selection tools, which are used to assign specific color values to control points in the luminance gradient.



16.14 Gradient color tools in the Indexed Color Node Panel.

GRADIENT CONTROL POINTS

Control points are indicated by triangle icons located along the bottom edge of the gradient. By default, the gradient has two control points, one located at the maximum luminance value (on the far right) and one at the minimum (on the far left).

You can add additional control points, and you can change the position of any control point along the gradient, that is, change the luminance value that a point controls:

- Add a control point to the gradient by dragging a color from the spectrum strip and dropping it over the gradient. A new triangle icon will appear at that position.
- Reposition a control point in the gradient by dragging it back and forth.

The gradient must have a minimum of two control points to define the range. However, you can delete any additional control point you have created by selecting it (click on the triangle to be sure it is selected) and pressing the Delete key.

ASSIGNING COLOR VALUES

To assign a color to a control point, you can drag a color from a swatch stored in the Image Viewer and drop it onto the control point. Alternatively, you can select a color from the spectrum strip located directly under the gradient and drag and drop it onto a control point.

In either case, when you release the mouse button over a triangle icon in the gradient, the color will be assigned to that control point and interpolated across the luminance range to the next control point.

You can also specify the color value for a control point numerically by selecting the point and using the color parameters underneath the spectrum strip. These color tools, which are common to a number of nodes in RAYZ, are described in detail in “Using the Color Parameters” in chapter 14 (p. 168).

INDEXED COLOR PARAMETERS

INDEX CHANNEL

This menu specifies which channel to use in the node operation. By default, a luminance channel is selected, which has been generated from the RGB data. However, you can use an individual image channel instead by selecting it from the menu.

LUMINANCE

This menu enables you to specify the type of luminance to be calculated when Luminance has been specified in the Index Channel menu. You can select Film, NTSC Video, or PAL Video. The formulas used by RAYZ to compute this data are in *Appendix A: How RAYZ Computes Luminance Values* (p. 439).

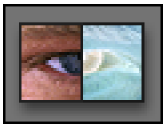
GRADIENT

The Gradient parameter is used to apply color to the tonal range of the image as described in the previous section on "Using Indexed Color" (p. 267).

CHANNEL SELECT

Use this parameter to specify which channels of the input image will be processed by the node. By default, the RGB channels of the input image are selected. To select or deselect a channel, press the button labeled with corresponding channel letter (such as A for Alpha) to toggle it to the opposite state.

INVERT NODE



The Invert node automatically inverts the color values in an image, effectively creating a negative of the image. The Invert node performs the following computation, where Z is the maximum color value and V is the color value of the pixel being processed:

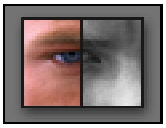
$$V = Z - V$$

The Invert node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

CHANNEL SELECT

Use the Channel Select parameter to specify which channels of the input image will be inverted. By default, the RGB channels of the input image are selected. To select or deselect a channel, press the button labeled with corresponding channel letter (such as A for Alpha) to toggle it to the opposite state.

MONOCHROME NODE



The Monochrome node desaturates the color component of an image to the extent that you specify. The Monochrome node performs the following computation on each image channel in turn:

$$C = mN + C(1-m)$$

In this formula, C represents the channel value of the pixel being processed (the red, green, or blue component of the RGB triplet); N is the result of the Luminance computation; and m is the Amount parameter value.

The Monochrome node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

MONOCHROME PARAMETERS

LUMINANCE

The Luminance parameter enables you to specify how the luminance value will be calculated. You can select Film, NTSC, or PAL luminance from the menu. For the formulas used by RAYZ to compute the three types of luminance, refer to *Appendix A: How RAYZ Computes Luminance Values* (p. 439).

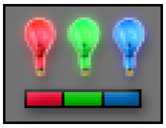
AMOUNT

This parameter enables you to control the degree of desaturation of the image by entering a value in the range of 0 to 1, where 0 represents no change to the input image and 1 represents complete desaturation. The default value is 1, producing a fully monochrome image.

CHANNEL SELECT

Use this parameter to specify which channels of the input image will be processed by the node. By default, the RGB channels of the input image are selected. To select or deselect a channel, press the button labeled with corresponding channel letter (such as A for Alpha) to toggle it to the opposite state.

Printer Lights Node



Printer lights are used by color timers in film labs to control the exposure, or density, of film prints. The Printer Lights node is designed to simulate the use of printer lights to control film density.

The Printer Lights node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

Using Printer Lights

The Printer Lights node might be used to compensate for color temperature or exposure problems in a filmed sequence or to match computer generated imagery to filmed imagery based on test prints of gray cards or other calibration material.

Parameter values in the Printer Lights node are in units that match the settings used to control printer lights in the lab, where each additional printer light increases log exposure by a specified fraction of a stop. The number of printer lights that is equivalent to a stop depends on the system used by the lab, typically 8 or 6 lights per stop. This value can also be adjusted in the node.

Any time that you need to make image adjustments in RAYZ based on reference data that has been specified in terms of printer lights, you can use this node without having to convert the values to work with the measurement units used in other nodes.

Printer Lights Parameters

The Lights parameter is used to adjust the image, and the Lights Per Stop parameter is used to adjust the unit scale of the Lights parameter to match the standard of the relevant color lab.

Lights

The Lights parameter is used to increase or decrease the number of printer lights. The master control affects the RGB channels equally, however you can expand the Lights parameter to adjust each channel individually.

The default value is 25, which is a standard printer setting for a normally exposed negative, and represents no change to the input image.

To increase exposure, increase the number of lights. Assuming that the Lights Per Stop parameter is set to 8, increasing the Lights parameter by 8 would double the exposure. Adding 16 lights to the default value would be equivalent to adding two stops, quadrupling the exposure of the input image.

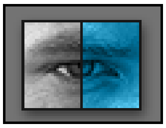
Lights Per Stop

This parameter specifies how many printer lights must be added to double the film density value, that is, to add a stop. The default value is 8, which is used by Technicolor labs, but to match a Foto-Kem lab, for example, you would set this value to 6.

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, the RGB channels of the input image are selected. To select or deselect a channel, press the button labeled with corresponding channel letter (such as A for Alpha) to toggle it to the opposite state.

TINT NODE



The Tint node is used to adjust the color values of the input image to match the colorspace of another image. The Tint node remaps the color values based on high and low values you sample in the other image.

The Tint node accepts a single RGB or RGBA input.

TINT PARAMETERS

Use the Tint parameters in the Node Panel to specify how the maximum and minimum values of the input image should be remapped (Map White To and Map Black To), and how much the image will change (Amount).

The "Map To" parameters use the standard RAYZ color controls, which are described in detail in "Using the Color Parameters" (ch. 14, p. 168). Although you can expand the parameters to access numeric controls, the Tint node is designed to take values sampled from another image—the image you want to match—by using the eyedropper tool.

TIP:

You can pin the Image Viewer to the reference image to make it easier to sample. Alternatively, you can pin the Node Panel to the Tint node and select the reference image node in the Worksheet to display it in the Viewer.

MAP WHITE TO

Click the eyedropper and sample a representative "white" area in the image you want to match. The whites in the Tint input image will be adjusted to match; that is, they will become bluer or warmer, brighter or darker, based on the new Map White To values.

MAP BLACK TO

Click the eyedropper and sample a representative "black" area in the image you want to match. The blacks in the Tint input image will be adjusted to match, based on the new Map Black To values.

AMOUNT

Use the Amount parameter to decrease the effect of the "Map To" parameters, if desired, by decreasing the default value of 1 (full effect). The range is 0–1.

Video Safe Node

The Video Safe node modifies the color information in an image if needed to conform to broadcast standards for NTSC and PAL video transmission. You choose which standard to conform to in the Video Type menu and the correction method to use in the Fix Method menu.

The Video Safe node accepts on input. Unlike most other color nodes, a mask input is not applicable to the operation performed by this node.

Because of the limited bandwidth of the broadcast video signal, certain high-intensity, highly saturated colors (pure red, for example) must be modified to reduce their amplitude to an acceptable level.

A given color is considered "unsafe" for transmission if the amplitude of its chrominance vector exceeds a predefined limit, or if the amplitude of the composite video signal exceeds a (different) predefined limit.

	NTSC Video	PAL Video
Analog Colorspace Referenced	YIQ	YUV
Gamma Correction Value Applied	2.2	2.8
Max Chrominance Amplitude	50 IRE *	50 IRE
Max Composite Signal Amplitude	110 IRE *	110 IRE

* For NTSC video, the reference black level is offset by a pedestal of 7.5 IRE; therefore, the maximum composite signal amplitude is actually 102.5 IRE (110 - 7.5) and the maximum chrominance amplitude is actually 42.5 IRE (50 - 7.5).

The Video Safe node evaluates each pixel in the input image using standard conversion factors for RGB-to-YIQ (if NTSC video is specified) or RGB-to-YUV (if PAL video is specified).

If the color of a pixel exceeds the boundaries of the colorspace used by the selected standard, then that color is corrected by reducing either the intensity or saturation, as specified in the Fix Method menu.

Video Safe Parameters

Video Type

Use the Video Type menu to select the broadcast standard to which the output image should conform:

- **NTSC Video.** The television signal standard used for broadcast in North America and Japan. NTSC uses the YIQ colorspace.
- **PAL Video.** The television signal standard used for broadcast in most of Europe and in many other countries. PAL uses the YUV colorspace.

Fix Method

Use the Fix Method menu to select the method to use to modify any unsafe pixels in an image:

- **Reduce Intensity.** Select this method to reduce the intensity of the pixel; that is, to reduce both the luminance and chrominance values.
- **Reduce Saturation.** Select this method to reduce only the chrominance values of the pixel; the luminance value is unchanged.

The best method to use will depend on the nature of the image being processed. As a general guideline, it is useful to remember that the luminance and chrominance values are encoded separately in broadcast video signals.

The chroma signal is derived by subtracting the luminance value of the color from the blue and red components: (B - Y) and (R - Y). The green component value is reconstructed from both luma and chroma signal information when the composite video signal is decoded by the receiving device.

This implies that the Reduce Saturation method, which reduces only the chrominance values—(R - Y) and (B - Y)—could increase the relative contribution of the green component to the output image and thereby alter the hue of the affected pixels.

The Reduce Intensity method, on the other hand, is more likely to darken the affected pixels.

chapter 17

TRANSFORM NODES

The Transform menu contains nodes used for spatial transformations such as cropping, scaling, and rotating: Crop, Flip Flop, Resize, Skew, and Transform. The Transform node is the most versatile—it translates, rotates, and scales.

The Morph node can be used to distort or morph images, and the Track node tracks movement through time. Tracking data can be used throughout RAYZ to control the animation of various operations across a range of frames. The Match Move and Stabilize nodes require tracking data to control their operations.

IN THIS CHAPTER

Filtering Transformations

The Resize, Transform, Match Move, and Stabilize nodes all include a Filtering parameter group to specify the type of filtering to apply to scaling and other transformations to remove aliasing, ringing, and similar artifacts that may be introduced.

The Filtering menu provides the following options:

- Best for natural scenes (the default)
- Best for CG scenes
- Bilinear interpolation
- No filtering
- Advanced - User Set

Select the option that fits your imagery. For the best looking result, choose the natural scene option for film footage or the CG scene option for computer generated images, as from a 3D program. RAYZ assumes that CG imagery is likely to have harder edges than filmed imagery and will dynamically choose the best filter for the type of transformation operation specified.

Bilinear interpolation is fast and often produces an adequate result. The fastest option is no filtering at all, which may be adequate when image quality is not a top concern, as when reviewing your initial settings.

If you want to specify individual filtering parameters yourself, select the Advanced option, which activates the parameters within the Filtering group.

Advanced Filtering Parameters

Expand the Filtering group to access the Advanced parameters, which become active whenever “Advanced - User Set” is selecting in the Filtering menu. Use the Advanced parameters to select a specific type of image filter and then set any other filter-specific parameters that are activated by your selection.

Filter Type

The Filter Type menu includes Box, Triangle, Quadratic, Cubic, and Gaussian; as well as Catmull-Rom/Overhauser Spline, Mitchell, Sinc (Windowed), Bessel (Windowed), Lanczos 2-lobe Sinc, and Lanczos 3-lobe Sinc.

Box, Triangle, Quadratic, Cubic and Gaussian refer to the shape of the distribution function that defines the filter. Box equals constant, triangle is linear; and so on in increasing precision and processing time.

The other filter options are named for the individuals who developed them, and the best choice will depend on the individual characteristics of the image as well as on the type of operations the node is performing. For example, since scaling an image down can introduce high frequency artifacts, a low-pass filter would be a good choice.

The Mitchell and Lanczos filters are among the most popular filter options. Mitchell filtering is often preferred for images with a lot of fine lines (and for JPEGs). Lanczos filtering generally provides a good compromise between sharpness, ringing and aliasing reduction, especially for images without a lot of hard edges. It is best used on images which will be scaled down.

TIP:

To experiment with different filter options when working with large images, use ROI to confine processing to a smaller area. For more information about using ROI, see "Defining a Region of Interest" in chapter 6 (p. 84).

Blur Factor

This parameter can be used with any filter option when heavy blurring is needed. The minimum value of 1, which is the default, adds no blurring to the image; values greater than 1 represent increased blurring, up to the maximum of 4.

Mitchell B and C

The Mitchell B and C parameters are activated when Mitchell filtering is selected in the Filter Type menu to enable you to make adjustments to the Mitchell filtering process.

The default values, 0.333 for both parameters, fall within a recommended range of roughly 0.25 to 0.5. Since the quality of the result must be judged based on your imagery and intentions, the best course may be to experiment with these settings.

As a guideline, note that values above 0.5 for the Mitchell B parameter may produce unnecessary blurring. Values above 0.5 for the Mitchell C parameter may result in excessive ringing (rippling patterns). When both parameters are set at values above 0.6 or below 0.2, anisotropic artifacts may appear.

Window Type

The Window Type parameter is activated only when the Sinc or Bessel filter type is selected from the Filter Type menu. Choose Hann, Hamming, Blackman, or Kaiser.

Kaiser A

When Kaiser is chosen in the Window Type parameter, this additional control parameter is activated, which modifies the level of filtering.

Adding Motion Blur to Transformations

The Transform, Match Move and Stabilize nodes all include a Motion Blur parameter group, which is used to add motion blur to the movement generated by the node. The RAYZ motion blur algorithm works by sampling and accumulating, just as film or any other light-gathering source would.

Motion Blur

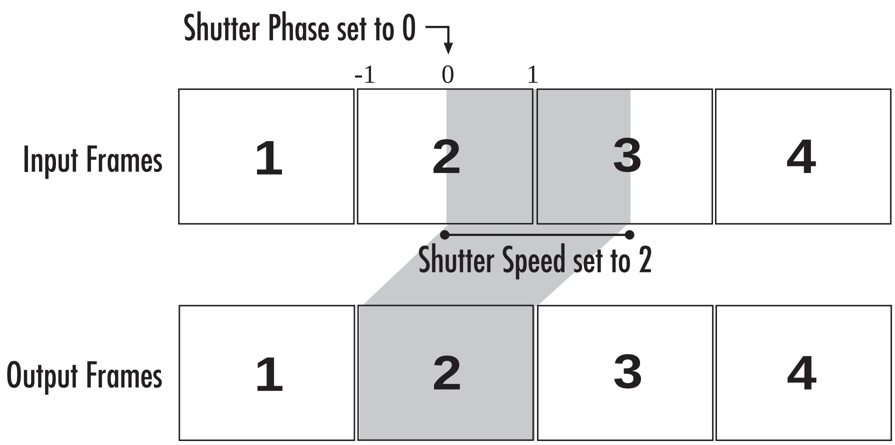
Check the Motion Blur box to turn on motion blur for the node. You can expand the Motion Blur group to access the Shutter Phase and Shutter Speed parameters:

Shutter Phase

The Shutter Phase parameter enables you to specify where in a frame to "open the shutter," in a range of -1 to 1. The default is 0, which represents the beginning of the frame.

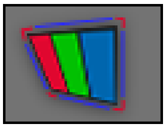
Shutter Speed

The Shutter Speed parameter enables you to specify the duration of the effect for each frame in a range of 0 to 2, where 2 represents the duration of one complete frame, or 360 degrees rotation of the iris (so that a Shutter Speed value of 1 represents 180 degrees, or a half-frame duration).



17.1 This diagram illustrates the effect of the Shutter Phase and Shutter Speed settings. Frame 2 of the output sequence is currently being processed.

CORNER PIN NODE



The Corner Pin node is used to animate the distortion of an image by controlling the position of the image corners through a series of frames. It can be used to stretch or twist an image, especially to simulate perspective.

The Corner Pin node is most often used to match the changing perspective of a moving element in a background image. For example, Corner Pin could be used to add a sign to the side of a truck driving along a winding road.

The Corner Pin node accepts one or two inputs: the image to be pinned and an optional reference image input. The reference image establishes the frame size of the output image. In addition, the primary input can be viewed in a temporary composite over the reference input by selecting Temporary Pre-Comp from the Source menu. (In fact, this view is selected by default when you connect a reference image.) However, the Corner Pin node only outputs the modified primary input (the pinned image), which can then be composited in an appropriate composite node.

USING CORNER PIN

The corner positions can be adjusted interactively using an overlay widget, similar to a cropping box, in the Image Viewer. The numerical values representing the *x,y* coordinates for each corner can be viewed and further adjusted in the Node Panel.

If you connect the background image to the reference input of Corner Pin, you will have the option of setting the corner positions while viewing the foreground, the reference image, or a temporary composite of the two.

All four corners can be animated, and the Corner Pin node often uses position data from a Track node to control the corner values.

USING TRACK DATA TO CONTROL A CORNER PIN

To use data generated in a Track node to control the position of each corner, follow these steps:

1. Connect the background image with the feature to be tracked to a Track node.
2. Create a point for each corner of the feature to be tracked and generate the tracking data. You can give each point a descriptive name, such as "top_left," to make it easy to identify which corner the track point is associated with. (See the description of the "Track Node" on p. 309 if you need more information.)
3. Connect the image to be pinned to the top input of a Corner Pin node and the background image (the one that you tracked) to the reference

input. For the corner pin operation to scale properly, the reference input, which establishes the output frame size, must be connected.

4. View the Corner Pin node in a Curve Editor. In the Curve Browser, check the box next to the *x* and *y* parameters for each corner to display them in the graph.
5. Animate the *x* and *y* curves for each corner by Ctrl-clicking on the dotted line at the first and last frame of the sequence to add two keyframes. This creates a constant curve the length of the sequence.
6. Right-hold on each curve to access the Curve Actions menu and select "Expression" from the menu list.
7. Open the Keypoint Editor panel in the Curve Editor to access the expression field for each curve (make sure the curve is visible in the graph or it will not show up in the editor).
8. Type the name of the appropriate track point to reference into each expression field using the following syntax, replacing "track1" with the actual name of the track node and "point_name" with the actual name of the track point:

   `/track1:points.point_name.position.x` (for the *x* curve)
   `/track1:points.point_name.position.y` (for the *y* curve)

EXPRESSION EDITOR OPTION

As an alternative to typing an expression into the Keypoint Editor as described in steps 7 and 8 above, you can right-hold on a curve in the graph and select Edit Expression.

In the Expression Editor, select Node Parameters and a list of all nodes in the Worksheet will be displayed. Select the Track node from this list to display all of the Track node's parameters in another list.

Then you can double-click the *x* or *y* position parameter for the appropriate track point and it will be entered in the expression field. This way you do not need to remember the proper syntax to use.

You can repeat the process with each curve, or you can then open the Keypoint Editor to access all of the expression fields in one list. You can copy and paste the first track point expression into the other fields, modifying only the point name and coordinate designation.

REVIEWING THE RESULTS

To view the results, select Temporary Pre-Comp in the Viewer's Source menu and use the flipbook controls to play the sequence in a temporary composite of the pinned foreground image over the background.

TIP:

Once you have set up a Corner Pin to use Track node data, you can save the nodes to the Custom menu to use as a template for future operations. (See also “Adding Nodes to the Custom Menu” in chapter 5, p. 59.)

In this way, any time you need to do a corner pin based on track data, select the custom nodes to add them to the worksheet and connect them to the specific images to be tracked and pinned. Reposition the four track points to the new image features and retrack. Be sure to match the track point name (“top_left,” e.g.) to the appropriate track feature.

Corner Pin Parameters

The Corner Pin Node Panel includes parameters that specify the position of the four corners as well as a Filtering menu.

Corner Positions

Each of the four corner position parameters provides a pair of data entry fields representing the *x,y* coordinate values for that corner. By default the parameter values create a four-corner pin area the size of the input image—the Bottom Left parameter value is 0, 0 and the Top Right parameter value is the maximum for the image. The corner parameters are listed in clockwise order starting at the bottom left corner:

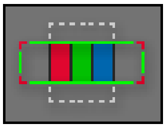
- Bottom Left
- Top Left
- Top Right
- Bottom Right

The values in the fields update automatically when the Corner Pin widget in the Viewer is adjusted, and vice versa.

Filtering

Use this menu to specify whether to use a bilinear interpolation filter to reduce aliasing and other image artifacts that may be introduced by the Corner Pin operation. Bilinear Interpolation is selected by default, as it is fairly fast and efficient, however, you can select No Filtering for the fastest processing.

CROP NODE



The Crop node enables you to crop, or trim, an image to a specified area. You can choose to output only the cropped area, that is, to change the size of the output frame; or you can output the same size frame as the input image, with the pixels outside the crop area set to the backing color specified for the source image.

DEFINING THE CROP AREA

The crop area can be defined interactively or numerically. When the Crop node is displayed in a Viewer and Input Image is selected in the Source menu, a crop box appears as an overlay on the image. You can drag the crop box in the image to interactively define the crop area, or you can adjust the parameter values in the Crop Node Panel—when one is changed, the other updates automatically to reflect the changes.

RESIZING

To resize the crop box,

- drag the edges or corners of the crop box in the Viewer or
- use the Size parameter in the Node Panel to specify the width and height of the crop box.

REPOSITIONING

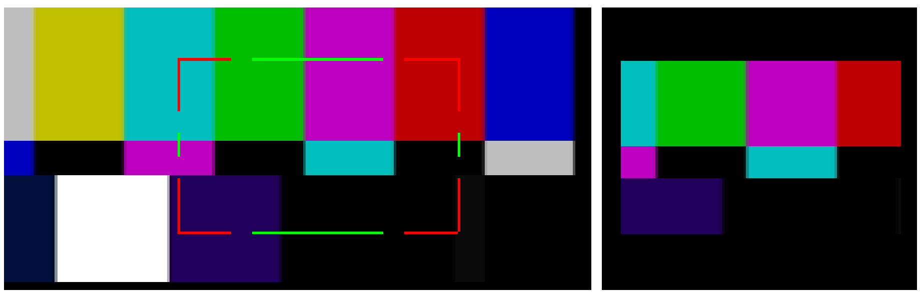
To reposition the crop box,

- drag the crop box across the image by clicking anywhere *inside the borders* of the box or
- use the Corner parameter in the Node Panel to position the bottom left corner of the crop box at a specific *x,y* coordinate.

VIEWING THE RESULT

By default, the input image is displayed in the Viewer to enable you crop to the desired area interactively. To view the cropped output image, switch the Source menu in the Viewer from Input Image to Output Image.

17.2 View the Input Image (left) to define crop area and switch the view to Output Image (right) to see the result.



TIP:

Use the **s hotkey** (with the cursor over the Image Viewer) to toggle back and forth between the input and output image views.

CROP PARAMETERS

Both the Corner and Size parameters display their values in pixel units by default. To view parameter values in floating point units, check the Float Display box in the Parameter menu (located on the right side of the parameter's slider control).

NOTE:

However you choose to display these parameter values, RAYZ actually stores them in floating point—that is, as percentages of total image size—rather than as specific pixel coordinates, to accommodate the use of proxies and clones.

CORNER

Use this parameter to position the cropping box over the input image by specifying the *x,y* coordinates of the bottom left corner of the box.

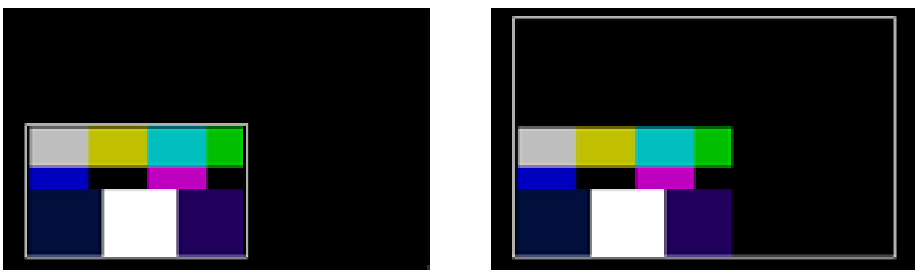
Unlike the Size parameter fields, the Corner fields accept negative values so that you can position the corner outside the bottom or right border of the image (the spatial coordinates of the bottom left corner of the input frame are 0,0).

SIZE

Use this parameter to specify the dimensions of the crop area. You can type positive pixel values into the Width and Height fields or use the Size menu to select a preset size from a list of film and video resolutions.

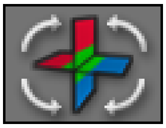
CHANGE OUTPUT SIZE

By default, the node resizes the output frame to the crop area. However, you can uncheck the Change Output Size box if you want the output frame to be the same size as the input, with the pixels outside of the crop area set to the Backing Color value of the source image (which is black, or 0, by default). Refer to the section on the "Backing Color Parameters" in chapter 14 (p. 164) if you need more information.



17.3 The frame boundaries are outlined in white in these examples. At left, output size changes to match the crop area; at right, output size matches input, with pixels outside crop area set to black (0).

Flip Flop Node



The Flip Flop node enables you to flip an image horizontally and/or vertically as well as to rotate it in 90-degree increments.

NOTE:

For finer control of rotation, or to animate rotation values across time, use the Rotate parameter in the Transform node instead.

The Flip Flop node provides three checkbox parameters, which can be used singly or in combination. When the associated box is checked,

- the **Rotate** parameter rotates the image 90 degrees counterclockwise,
- the **Flip H** parameter flips the image horizontally, and
- the **Flip V** parameter flips the image vertically.

The Flip Flop parameters can be used in the following combinations to rotate the input image:

	ROTATE	FLIP H	FLIP V
90 Degrees Counterclockwise	On	Off	Off
180 Degrees	Off	On	On
90 Degrees Clockwise	On	On	On

MATCH MOVE NODE

The Match Move node is used to animate a foreground element to match the movement in a background scene.

Match Move accepts one or two inputs, with the second optional input being used for a reference image that establishes the frame size of the output. This means that the reference image must be connected to Match Move whenever it differs in size from the primary input image, because the track point values are scaled in relation to the output size.

Another reason to use a reference input is that the primary input can be viewed in a temporary composite over the reference image by selecting Temporary Pre-Comp from the Source menu. (In fact, Temporary Pre-Comp is selected by default when you connect a reference image.)

NOTE:

Match Move uses tracking data to control the translation, rotation, and scaling of the foreground over time. The Match Move node does not create tracking data. Refer to the description of the Track Node (p. 309) later in this chapter for instructions on generating track data.

USING THE MATCH MOVE NODE

A match move is performed to position a foreground element at each frame to match the movement of an object in a background image. The Match Move node uses data previously created in a Track node by tracking an appropriate area of the background.

For match moves, follow these steps:

1. Be sure that Match Move is selected in the Translation menu.

   The Translation menu enables you to choose Match Move (the default) or No Translation. (The No Translation option is provided for matching scaling or rotation only.)

2. In the Translate Point menu, specify the Track node and point from which the Match Move node should access *x,y* position data.

   The Translate Point menu automatically lists all the track points in every existing Track node in the file.

CREATING AN OFFSET

Optionally, you can create an Offset for the position data by dragging the Offset overlay in the image viewer to the desired location, or by entering *x,y* coordinate values into the Offset parameter fields.

17.4 The Offset overlay is a simple crosshairs device located by default at the 0,0 coordinates of the image; that is, the lower left corner.

You can also match scaling and rotation, as explained below.

MATCHING SCALING AND ROTATION

You can match scaling and rotation in conjunction with the match move. Additionally, you can use the “No Translation” option when you want to use tracking data to match scaling and rotation alone.

NOTE:

The scale and rotate operations in the Match Move node are specifically designed to use track data to adjust scaling and rotation over time. If the image needs preprocessing on another basis—for example, if the image needs to be resized to fit the scale of other objects in the background scene—use the appropriate node (Resize or Transform) between the input image and this node.

1. Start by selecting an option in the Rotation and/or Scale menus for how the track data should be applied to the operation. You can choose to match the rotation in the original track and to match size or distance scaling.
2. Then expand the Rotation and/or Scale groups and select a pair of tracking points to use in the associated menus (Pivot and Reference in the Rotation group, and Reference A and B in the Scale group).

Refer to the “Rotation and Scale Parameters” (p. 289) description for more detailed information about these parameters.

NOTE:

Unlike the Translation parameter, which references a single track point, Rotation and Scale each require two track points because the scale and rotation values are controlled by changes in the length or relative position of the vector between the two points selected.

MATCH MOVE PARAMETERS

TRANSLATION

The Translation menu can be used to change the default selection from Match Move to “No Translation.” This option should be selected only if you are using the node strictly to match rotation or scaling, without any other translation.

Expand the Translation parameter to access the Translate Point and Offset parameters:

Translate Point

This parameter enables you to specify which track point to reference for the Match Move operation. The Translate Point menu will automatically list all track points in all existing Track nodes.

Offset

The Offset parameter enables you to offset the position data by keying specific *x,y* values into the fields. The Offset parameter is tied to the corresponding overlay in the Image Viewer, which can also be used to specify the Offset value by dragging the crosshairs device over the desired location in the image.

Rotation and Scale Parameters

The Rotation and Scale parameters enable you to match rotation and scaling. Unlike the Translation operation, the Scale and Rotation operations each require two sets of track data, as the changes in length or position of the vector between two track points are used to calculate the values.

Rotation

The Rotation menu enables you to select No Rotation (the default) or Match Rotation.

The associated Pivot and Reference menus become active whenever Match Rotation is selected (expand the Rotation parameter to access). Use the menus to select the track point data to use for each parameter.

Pivot Menu For the Pivot, use the track point that represents the pivot of the rotation.

Reference Menu For the Reference, use the point that represents the position of the moving end of the rotation vector.

Scale

The Scale menu enables you to select No Scale (the default), Match Size Scale, or Match Distance Scale:

- Match Size Scale enables you to match a scale up or down in size, such as a zoom.
- Match Distance Scale enables you to match relative changes in distance between objects.

Reference A and B The associated Reference parameters become active whenever a match option is selected (expand the Scale parameter to access). Use the Reference A and Reference B menus to select the track point to use to represent each end of the vector defining the scale value.

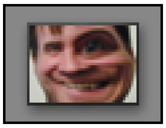
Filtering

Use this menu to select the filtering option to use to reduce aliasing and other artifacts that may be introduced by the node operations. The “Best for natural scenes” and “Best for CG scenes” options are recommended choices; however, you can have complete control over the options by selecting “Advanced - User Set” from the menu. This is explained in detail in “Filtering Transformations” on p. 278.

Motion Blur

Check this box to add motion blur. You can expand the Motion Blur group to access parameters that control the phase and duration of the effect, as explained in “Adding Motion Blur to Transformations” on p. 280.

MORPH NODE



The Morph node is used to distort or warp an image, or to morph one image into another. It can be used to correct a mismatch in size and shape between two image elements, as when an actor's face is to be composited over the face of a stunt double, where the final effect should be invisible to the audience. It can also be used for flashy effects such as morphing an actor into a monster.

The Morph node is spline-based; that is, you draw splines in the Image Viewer to designate the image areas to be distorted. You can draw open-ended splines or closed shapes.

The Morph node accepts one or two inputs. The second input is used when morphing between two separate image sequences. The top input is for the source image and the bottom is for the target image, which means that the node will morph from the source image to the target input.

The output will be the same number of channels as the source input unless you elect to output a shape matte with an RGB source image, as explained in the Alpha menu description in "Morph Parameters" (p. 299).

USING THE MORPH NODE

The basic steps to create a warp effect in an image are as follows:

1. Define the source spline—the pixel area that will be the focus of the distortion. (See "Creating Splines" on p. 292.)
2. Define the target spline—the size, shape, and location to which the source area will be distorted. (See "Creating Splines" on p. 292.)
3. Connect the source spline to the target spline. (See "Making and Adjusting Connections" on p. 294.)
4. Set the distortion level from source to target. (See "Animating Distortion Levels" on p. 297.)

When warping a single input, the source and target splines represent areas in the same image. When morphing between two image inputs, however, the source spline represents an area in the source image and the target spline, an area in the target image.

To morph between two images:

1. Perform steps 1–4 above for warping an image, drawing the target spline (step 2) over an image area in the target input. (See "Selecting a View" on p. 297.)
2. Then set the image dissolve level from source to target. (See "Animating Dissolve Levels" on p. 297.)

You can draw as many pairs of splines as necessary, as described in "Creating Splines" below. For example, you might draw source shapes outlining the eyes and mouth of a head, and then draw exaggerated versions of them to use as the targets. Different distortion and dissolve values can be set for each connected pair.

You can also protect an area from being distorted, as described in "Boundary Shapes: Preventing Pixels from Distorting" on p. 298.

Creating Splines

Typically, you would start by drawing a spline around an image feature that you want to distort, animating the spline, if necessary, to follow the feature throughout the sequence. Then you would draw another spline that represents the distortion, also animating it if necessary. (See also "Animating the Effect" on p. 296.)

You can create as many splines as you need, in any order, and any spline can become a source or target spline. It is the order in which two splines are connected that defines them as source and target: the source spline is selected first, as described in "Making and Adjusting Connections" on p. 294.

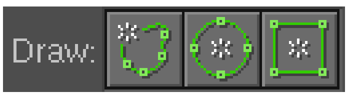
Splines are drawn over the image, using the Morph node controls in the Image Viewer to select drawing, editing, and connection modes. The image to view while drawing a spline is selected in the Viewer's Source menu, as described in "Selecting a View" on p. 297.

NOTE:

In general, you draw and edit a spline in the Morph node in the same way, using the same drawing tools, as in the Roto node. The difference is that the Morph node enables you to draw open-ended splines or closed shapes (in Roto you have to close a shape), and that additional tools for connecting splines are included in the Morph control strip.

For more information about drawing and editing splines, see also "Using the Roto Node" in chapter 15, p. 210.

Drawing a Spline



17.5 Draw mode buttons in the node control strip are, from left to right, Freehand, Ellipse, and Rectangle.

Select a drawing mode from the node control strip. The Ellipse and Rectangle draw modes are used to draw a prescribed shape in one dragging motion. Freehand mode is used to draw an open spline or a closed shape of any configuration.

The node controls should be displayed in the Viewer; if not, right-hold anywhere in the image to access the Viewer Actions menu and select the Node Controls item.

Freehand Drawing

Click in the image to create the starting point, move the cursor to a new location, and click again. A line will be drawn between the two points. You can continue to move and click, creating as many spline segments as you need:

- To create a **closed shape**, return to the starting point and click it again.
- To create an **open spline**, click the last point again to signal that the spline is complete.

17.6 Examples of open and closed splines. An open spline has been drawn across the top of the mouth, and a pair of closed spline shapes have been drawn around the eyes.

You can create linear (straight) or curved spline segments. A curved segment is controlled by a point with handles you can use to adjust the slope and acceleration of the curve.

The click-and-move-and-click method creates linear segments; to create a curved segment as you draw, click and drag briefly before releasing the mouse button and moving to a new click-spot. (This process is more complicated to describe than to perform.) In addition, you can always change the state of any existing point after the spline is drawn: Ctrl-click a linear point to convert it to a curve and vice versa.

For each spline you draw, an entry is created in the Splines list in the Node Panel. You can use this entry to give the spline a distinctive name, which is especially helpful when working with many splines in a complex morph. See also "Splines List" on p. 300.

Editing Splines

To edit a spline, click an Edit or Transform mode button.

17.7 The Edit and Transform mode buttons.

Edit Mode Buttons The Edit buttons are used to edit points in existing splines. In both editing modes, any spline point can be selected and manipulated. The choice of mode is only relevant when manipulating points with handles (points that control curved segments).

Selecting the Edit button on the left ensures that both ends of a point handle move when you drag one end. The other Edit mode enables you to adjust each end of a handle separately.

Transform Mode Buttons In Rotate or Scale mode, dragging selected points or shapes will rotate them or scale them, based on the mode selected.

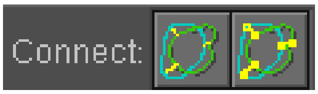
Spline manipulation is covered in more detail in the Roto node description. Refer to the sections on "Editing a Shape" (ch. 15, p. 212) and "Transform Modes" (ch. 15, p. 214).

TIP:

To duplicate a spline, copy and paste the corresponding entry in the Node Panel. See "Copying, Pasting, and Deleting Layers" in chapter 7, p. 94 if you need instructions.

Making and Adjusting Connections

To make and edit connections, press the appropriate Connect mode button in the Morph node controls in the Viewer.



17.8 Connect Mode buttons in the node control strip: Make Connection mode (left) and Edit Connection mode (right).

Connecting Splines

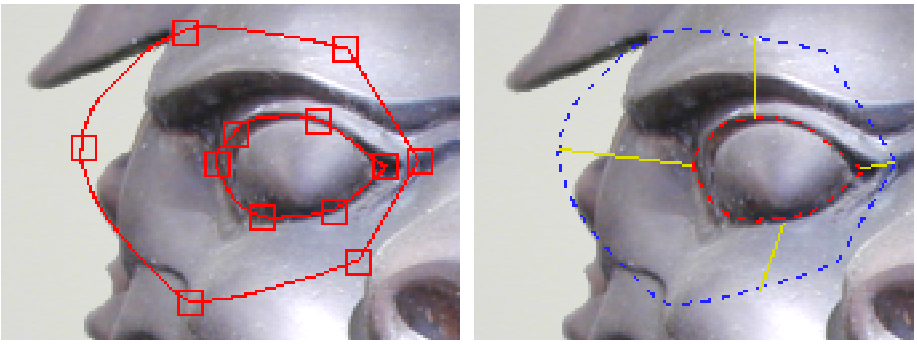
To create a connection between two splines, press the Make Connection mode button and select the splines:

- First click the spline that should become the source.
- Then click the spline that should become the target.

When two splines are connected, their overlays are color coded by the node: red represents the source and blue, the target. Several yellow connection lines representing correspondence points will link specific locations on the source spline to corresponding locations on the target.

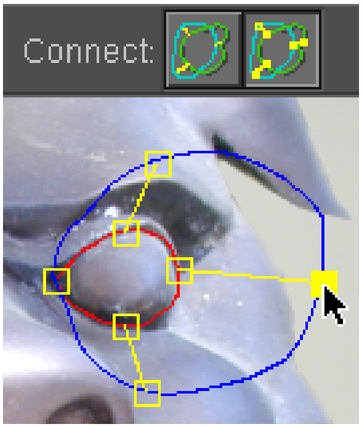
TIP:

You can change the colors used to code the splines and connections in Edit > Project Settings > Current Colors > Overlay Gadget Color Palette. Click on Source Color, Target Color, Connection Color, or Boundary Color to redefine the color used to draw the corresponding overlay.



17.9 The example on the left shows two unconnected splines; the one on the right shows the same splines connected to each other. The inner spline (red) is the source and the outer spline (blue) is the target.

ADJUSTING A CONNECTION



17.10 In Edit Connection mode you can manipulate the connection lines, as illustrated here.

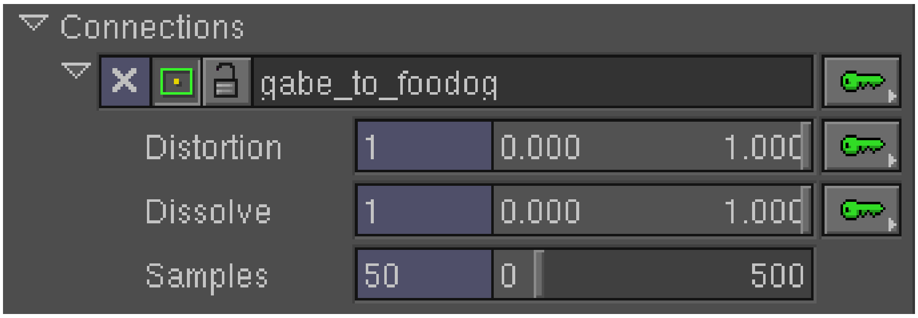
You can adjust the default connections made by the node if necessary. Press the Connect button on the right to enter Edit Connection mode and then drag the connection point at either end of a yellow line.

Say, for example, that you are morphing from an eye in one image to an eye in the other image. You might want to reposition the end-points of a connection line so that the spline point over the inner corner of the source eye is connected to the spline point over the inner corner of the target eye.

You can also delete a connection line by selecting one of its points and pressing the Delete key. Be careful not to select the connection line itself as this may select all connection lines. If you do delete all of them, you will disconnect the two splines altogether and the corresponding connection entry in the Node Panel will be deleted too. The Undo command is Ctrl-z.

CONNECTION ENTRIES

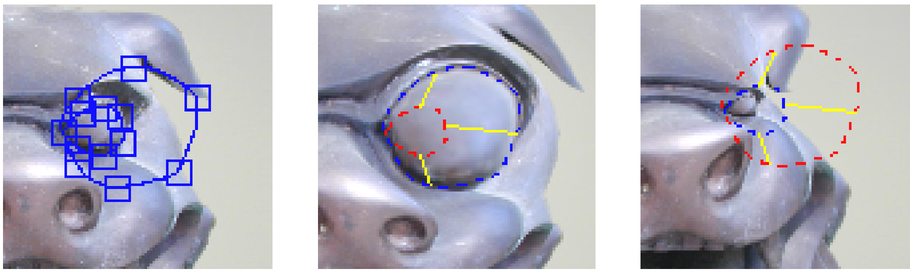
A new entry will appear in the Node Panel Connections group to represent each connected pair of splines. The connection entry name automatically includes the names of both of the splines, to helps distinguish among multiple connections.



17.11 The connection entry in the Morph Node Panel is where you access the Distortion and Dissolve parameters for the spline-pair.

It also includes the parameters used to control the distortion and dissolve levels, as described in "Animating the Effect" (p. 296). When working with multiple pairs of splines, you may want to use the Dissolve parameters for each connected pair in conjunction with the Overall Dissolve parameter (located at the top of the Node Panel), as described in "Morph Parameters" on p. 299.

DELETING CONNECTIONS: You can delete a connection by deleting the entry. Click once on the entry label to select it and then right-hold on the entry and select Delete from the actions menu. This does not delete the shapes, but they are no longer defined as source or target until you reconnect them (to each other or to different shapes).



17.12 The order in which two splines are connected affects the outcome, as these examples illustrate: The smaller shape has been designated as the source in the middle image, and as the target in the image on the right.

ANIMATING THE EFFECT

Typically you will be distorting a sequence rather than a single frame, and you can animate the splines as well as the distortion and dissolve levels. In fact, the distortion and dissolve levels are animated by default.

ANIMATING SPLINES

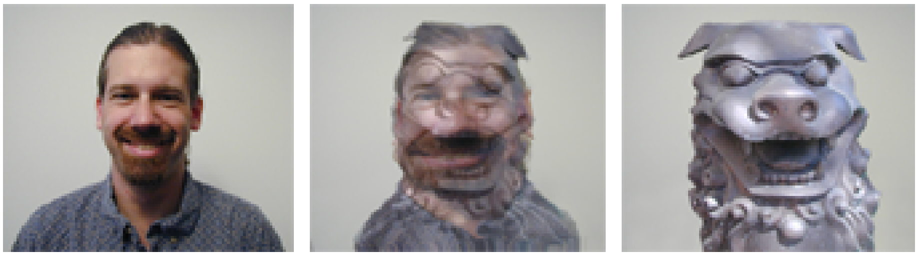
Always start by navigating to the first frame, or whichever frame the animation will start on, and turning on Autokey mode (press the Autokey button in the Viewer or the Node Panel). If you need more information, refer to "Using Autokey Mode" in chapter 7, p. 99.

You animate a Morph spline exactly the way you animate a spline shape in the Roto node: in Autokey mode, draw the spline at the first frame and then modify its shape and position at key frames. The node will interpolate the in-between frames.

Animating Distortion Levels

The distortion level for each connected pair of shapes is controlled by the Distortion parameter, which is located in the corresponding connection entry in the Node Panel (see *Fig. 17.11*).

Distortion is animated by default, using linear interpolation from 0 (no effect) at the first frame to 1 (maximum distortion) at the last frame, in the global range defined for the project.



17.13 First, middle, and last frames of a morph sequence from programmer to foo dog.

To use the same value for every frame, select “Delete All Keys” or “Change Interpolation > To Constant” from the parameter Animation menu. For general information about animating node parameters, refer to “Animating Parameter Values” in chapter 7, p. 99.

Animating Dissolve Levels

The Dissolve parameter becomes active when morphing between two image inputs to control the level of dissolve from the source input to the target input.

Dissolve is animated by default in the same way as Distortion, using linear interpolation from 0 to 1, where a value of 0 returns only the source input and a value of 1, only the target input.

To use the target input only as a reference to draw the target spline, change the Dissolve interpolation to Constant (so that the value will not change across the frame range) and set the Dissolve value to 0.

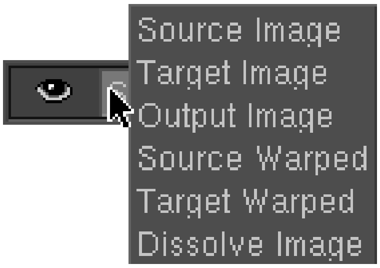
17.14 The target input was used as a reference to draw the target spline (middle), and the Dissolve value was set to 0 to warp the source image (left) to fit the target shape. The result is shown at right.

Selecting a View

Depending on the number of inputs, there may be up to six different views available in the Image Viewer when displaying a Morph node. The available views are always listed in the Viewer’s Source menu. You can cycle through all the available views by repeatedly pressing the **Source menu hotkey (s)**.

At minimum, you can choose between viewing the Source Image and the Output Image. When a target input is connected, you can also view the Target Image.

When morphing between two inputs you can also view the warped source or target image in isolation by selecting Source Warped or Target Warped. You can even view the dissolve matte used by the node to blend the warped source and target images by selecting Dissolve Image.



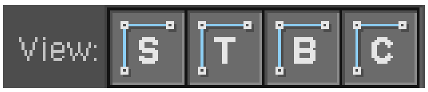
17.15 Source menu views available when morphing between two inputs.

View the Source or Target Image when you want to draw splines that define features in one of these images.

Then switch to Output Image to see the image that will be output from the node at the current frame. Always double-check that the Source menu is set to Output Image before running a flipbook of the morphed sequence.

VIEWING SPLINE OVERLAYS

The Morph node controls in the Image Viewer include a group of View buttons used to control the display of spline overlays.



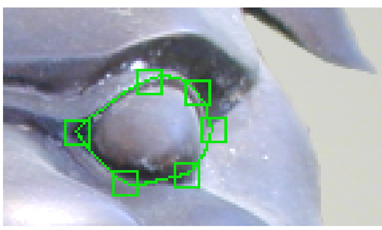
17.16 View buttons in the node control strip.

Unlike the overlay display buttons associated with each spline entry in the Node Panel, which affect only one spline, each View button toggles the display of all overlays of a specific type on or off in the Image Viewer:

- S controls display of Source splines.
- T controls display of Target splines.
- B controls display of Boundary splines.
- C controls display of Connection splines.

BOUNDARY SHAPES: PREVENTING PIXELS FROM DISTORTING

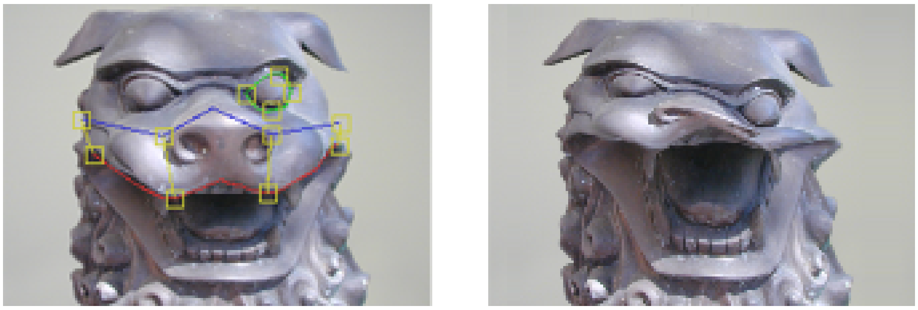
During the morph operation, when the pixels defined by the splines are warped, the surrounding pixels will also be distorted according to how they are “pushed and pulled” by the modification of the warped pixels.



17.17 Boundary shape drawn to prevent pixels inside the boundary from being distorted by the node operation.

To protect an image area from change, draw a boundary around the area and then connect the shape to itself by clicking the same shape twice while in Make Connection mode.

Whenever you connect a shape to itself, it becomes a boundary shape, which is color-coded in green. A corresponding entry is created in the Connections list in the Node Panel for each boundary shape. When morphing between two images, a boundary shape protects the pixels it bounds in both the source and the target images.



17.18 A pair of open splines are used to warp the mouth, with a boundary shape used to protect one eye from modification (left). The result (right) is a lopsided roar.

Morph Parameters

Overall Dissolve

The Overall Dissolve parameter can be used as a master dissolve level control for all of the spline-pairs by referencing Overall Dissolve in the individual Dissolve parameters for each connection entry. Unless you specifically reference the Overall Dissolve parameter, however, it has no effect on the image.

To reference the Overall Dissolve value, use the Animation menu of the individual Dissolve parameters to change the interpolation type to Expression and then select Edit Expression from the same menu.

In the Expression Editor, enter the same expression in all of the Dissolve parameters: `/morph1:dissolve` (replacing "morph1" with the actual name of the Morph node). Then when you adjust the Overall Dissolve parameter, the individual dissolve parameter values are reset.

If you forget the syntax, just use the Expression Browser at the bottom of the editor panel to select the Morph node. All of its parameters, including the Overall Dissolve, will be listed in the pane on the right. Double-click on the Overall Dissolve entry and the correct expression will be entered in the field automatically. See also *Appendix C: Using Expressions in RAYZ* (p. 445).

Alpha

By default, the Morph node does not affect the input alpha channel, if any, as reflected in the default Alpha menu setting: No Changes.

However, if you select the other Alpha menu option, Closed Shape Matte, the node will generate an alpha channel for output with opacity values based on the closed spline shapes you have drawn and animated according to the distortion level settings.

Splines List

An entry is created in the Splines list for each spline you draw in the image. The procedure for drawing splines was described above in "Creating Splines" (p. 292).

You can give each spline a unique name; in fact, this is recommended when working with numerous splines to help identify them easily. You can disable a spline temporarily, turn off the display of its overlay, or lock it to keep from modifying it accidentally.

You can also duplicate a spline (by copying and pasting its entry in the Node Panel) or delete it altogether if you wish. All of these actions are described in the section on "Dynamic Parameter Groups" in chapter 7, p. 93.

Connections List

An entry is created in the Connections list for each connected pair of splines. The procedure for connecting two splines was described previously in "Making and Adjusting Connections" (p. 294). The connection entry name automatically includes the names of both of the splines, to help distinguish among multiple connections when you have drawn many splines.

The connection entries have the same controls as the spline entries, which means that you can delete a connection. This does not delete the splines, but they are no longer defined as source or target until you reconnect them (to each other or to different splines).

Each connection entry can be expanded to access Distortion, Dissolve, and Samples parameters, which are specific to the spline-pair.

Distortion

This parameter controls the amount and rate of distortion from the source spline to the target spline. By default the Distortion value is animated linearly from 0 (no distortion) to 1 (maximum distortion) using the global range specified in the Time Scooter when the connection was created. See also "Animating Distortion Levels" on p. 297.

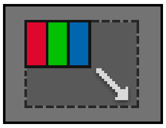
Dissolve

This parameter controls the amount and rate of the dissolve from the source image to the target image. By default the Dissolve value is animated linearly from 0 (all source image) to 1 (all target image) using the global range specified in the Time Scooter when the connection was created. See also "Animating Dissolve Levels" on p. 297 and "Overall Dissolve" on p. 299.

SAMPLES

This parameter specifies the number of point samples to use. If you think of a spline as a string of pearls, the Samples value represents how many pearls are in the string. In most cases the default value of 50 will be satisfactory, however, you may want to increase it for especially long spline segments if the distortion effect doesn't seem smooth enough in those areas.

RESIZE NODE



The Resize node changes the spatial resolution of an image by scaling it up or down to fit the dimensions you specify.

The Resize node accepts one or two inputs. The top input is the image to be resized, and the optional bottom input is used as a reference for the resizing operation.

TIP:

If you are resizing an input to match the size of another image in the network, the fastest method will probably be to connect the other image to the bottom input of the Resize node. This will automatically scale the primary input image to match.

RESIZE PARAMETERS

SIZE

Use the Size parameter to specify the size to which the input image should be scaled. You can enter specific pixel dimensions by typing a positive integer into each field, or you can use the Size menu to select a size from a list of common film and video resolutions.

If you would prefer to work in scale factors (fractional units) rather than pixels, check Float Display in the Animation menu, which is located on the right side of the Size parameter slider. Then, for example, to scale the image by half, you would enter 0.5 in each field.

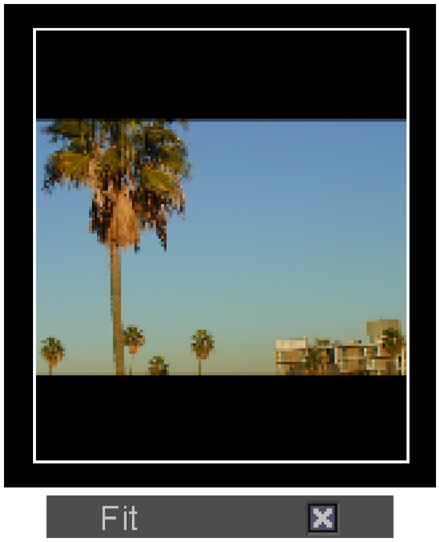
NOTE:

No matter which display units you use, RAYZ actually stores the Size value as floating point data rather than specific pixel values to accommodate the use of proxies and clones.

The Size parameter is deactivated when a reference input is used to resize the image.

FIT

Check the Fit box to preserve the aspect ratio of the input image when resizing.

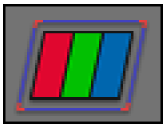


17.19 Example of how the Fit setting affects the output of the Resize node: when checked (at right), this parameter constrains the proportions of the output to match the aspect ratio of the input.

FILTERING

Use this menu to select the filtering option to use to reduce aliasing and other artifacts that may be introduced by scaling. The "Best for natural scenes" and "Best for CG scenes" options are recommended choices; however, you can have complete control over the options by selecting "Advanced - User Set" from the menu. This is explained in detail in "Filtering Transformations" on p. 278.

SKEW NODE



The Skew node slants the input image in the direction you specify. One edge is effectively translated relative to its parallel edge, with the intervening pixels on the axis lined up along the diagonal created by the skew.

SKEW PARAMETERS

DIRECTION

Use the Direction menu to select the orientation of the slant:

- Horizontal skew slants the upper edge of the image left or right.
- Vertical skew slants the right edge of the image up or down.

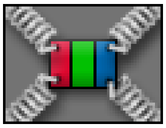
AMOUNT

Use this parameter to specify how much to skew in the selected direction. The range is -1 to 1, with the default value of 0 representing no change:

- For a horizontal skew, negative values will slant the image to the left, and positive values will slant it to the right.
- For a vertical skew, negative values slant the image down and positive values slant it up.

17.20 Examples of Skew settings, clockwise from upper left: horizontal, -1; horizontal,1; vertical, -1; vertical,1.

STABILIZE NODE



The Stabilize node is used eliminate the appearance of overall motion, as when the input sequence suffers from camera jitter.

Stabilize accepts one or two inputs, with the second optional input being used for a reference image that establishes the frame size of the output. This means that the reference image must be connected to Stabilize whenever it differs in size from the primary input image, because the track point values are scaled in relation to the output size.

Another reason to use a reference input is that the primary input can be viewed in a temporary composite over the reference image by selecting Temporary Pre-Comp from the Source menu. (In fact, Temporary Pre-Comp is selected by default when you connect a reference image.)

NOTE:

Stabilize uses tracking data to control the stabilization operations. The Stabilize node does not create tracking data. Refer to the description of the Track Node (p. 309) later in this chapter for instructions on generating track data.

USING THE STABILIZE NODE

To stabilize imagery, the Track node that is accessed must contain a track point that was placed to follow a background element that is meant to remain stationary throughout the scene.

For image stabilization, follow these steps:

1. Be sure that Stabilize is selected in the Translation menu.

   The Translation menu enables you to choose Stabilize (the default) or No Translation. (The No Translation option is provided for undoing scaling or rotation without translating.)

2. In the Translate Point menu, specify the Track node and point from which the Stabilize node should access *x,y* position data.

   The Translate Point menu automatically lists all the track points in every existing Track node in the file.

CREATING AN OFFSET

Optionally, you can create an Offset for the position data by dragging the Offset overlay in the image viewer to the desired location, or by entering *x,y* coordinate values into the Offset parameter fields. The Offset overlay is illustrated in *Fig. 17.4* (p. 288) of the Match Move node description.

You can also undo scaling and rotation, as explained below.

Undoing Scaling and Rotation

You can undo scaling and rotation in conjunction with the stabilization operation. Additionally, you can use the “No Translation” option to use tracking data to undo scaling and rotation alone.

NOTE:

The scale and rotate operations in the Stabilize node are specifically designed to use track data to undo scaling and rotation over time. If the image needs preprocessing on another basis—for example, if the image needs to be resized to fit the scale of other objects in the background scene—use the appropriate node (Resize or Transform) between the input image and this node.

1. Start by selecting an option in the Rotation and/or Scale menus for how the track data should be applied to the operation. You can choose to undo the rotation in the original track and to undo size or distance scaling.
2. Then expand the Rotation and/or Scale groups and select a pair of tracking points to use in the associated menus (Pivot and Reference in the Rotation group, and Reference A and B in the Scale group).

Refer to the “Rotation and Scale Parameters” (p. 307) description for more detailed information about using these parameters.

NOTE:

Unlike the Translation parameter, which references a single track point, Rotation and Scale each require two track points because the scale and rotation values are controlled by changes in the length or relative position of the vector between the two points selected.

Stabilize Parameters

Translation

The Translation menu can be used to change the default selection from Stabilize to “No Translation.” This option should be used only if you are using the node strictly to undo rotation or scaling, without any other translation.

Expand the Translation parameter to access the Translate Point and Offset parameters:

Translate Point

This parameter enables you to specify which track point to reference for the Stabilize operation. The Translate Point menu will automatically list all track points in all existing Track nodes.

Offset

The Offset parameter enables you to offset the position data by typing specific *x,y* values into the fields. The Offset parameter is tied to the corresponding overlay in the Image Viewer, which can also be used to specify the Offset value by dragging the crosshairs device over the desired location in the image.

Rotation and Scale Parameters

The Rotation and Scale parameters enable you to undo rotation and scaling. Unlike the Translation operation, the Scale and Rotation operations each require two sets of track data, as the changes in length or position of the vector between two track points are used to calculate the values.

Rotation

The Rotation menu enables you to select No Rotation (the default) or Undo Rotation.

The associated Pivot and Reference menus become active whenever Undo Rotation is selected (expand the Rotation parameter to access). Use the menus to select the track point data to use for each parameter.

Pivot Menu For the Pivot, use the track point that represents the pivot of the rotation.

Reference Menu For the Reference, use the point that represents the position of the moving end of the rotation vector.

Scale

The Scale menu enables you to select No Scale (the default), Undo Size Scale, or Undo Distance Scale:

- Undo Size Scale enables you to undo a scaling up or down in size, such as a zoom.
- Undo Distance Scale enables you to undo relative changes in distance between objects.

Reference A and B The associated Reference parameters become active whenever an undo option is selected (expand the Scale parameter to access). Use the Reference A and Reference B menus to select the track point to use to represent each end of the vector defining the scale value.

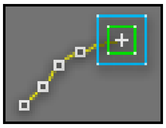
Filtering

Use this menu to select the filtering option to use to reduce aliasing and other artifacts that may be introduced by the node operations. The “Best for natural scenes” and “Best for CG scenes” options are recommended choices; however, you can have complete control over the options by selecting “Advanced - User Set” from the menu. This is explained in detail in “Filtering Transformations” on p. 278.

Motion Blur

Check this box to add motion blur. You can expand the Motion Blur group to access parameters that control the phase and duration of the effect, as explained in "Adding Motion Blur to Transformations" on p. 280.

Track Node



The Track node enables you to identify one or more features in an image, which the Track node then follows throughout the sequence, creating *x,y* coordinate values representing its position at every frame. The position data created by the Track node can be accessed from other nodes to perform various operations.

There is almost no limit to what you can do with this tracking data. Two common applications are to stabilize an image and to match the motion of a foreground element to an object in the background plate (i.e., match move). See also the descriptions of the "Match Move Node" (p. 287) and the "Stabilize Node" (p. 305). The "Corner Pin Node" (p. 281) can access tracking data to control the position of each corner, and complex animation effects can be created by using tracking data in conjunction with the "Transform Node" (p. 319).

The Track node accepts one input. The image data output from the Track node is the same as the input, and it is not necessary to connect the output of Track to another node to use the tracking data generated by the node.

Any node parameter that can use expressions can reference tracking data, and some nodes that rely on tracking data (Match Move, for example) provide convenient menus for selecting which Track point values to use.

Generating Track Data

Start by connecting the image sequence to be tracked to the input of a Track node. Make sure that

- the Track node image is displayed in an Image Viewer and that the Node Controls item is checked in the Viewer Actions menu (right-click and hold in Viewer to access menu).
- Autokey mode is on (it is by default).
- all frames within the range you want to track are sequential; that is, that there are no gaps between frames.

You can track a sequence that contains gaps, however the Track node will stop processing when it reaches a non-existent frame.

TIP:

If there are gaps in the sequence to be tracked, or you need to perform other re-sequencing tasks before tracking, you can insert a "Sequence Node" (ch. 21, p. 415) between the input node and the Track node. You can use the Sequence node to close gaps, track on twos, and so forth.

Follow steps A through D below for each image area you need to track. Then continue on to steps E and F. If applicable, see also “Tracking Features That Leave Frame” on p. 312.

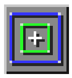
TIP:

When tracking multiple points in the same image, it may be helpful to give each point a descriptive name that makes it easier to identify in the Image Viewer and the Curve Editor. (The name field is located in the top line of each point entry in the Node Panel.)

STEP A: CREATE A TRACK POINT

1. Navigate to the first frame that you want to track and make sure that Autokey is on.

2. Click the “Add Point” button in the Track Node Panel.



17.21 Each time you press the Add Point button, a new track point is created.

A track point entry will be created in the Points list of the Node Panel, and a corresponding overlay for the track point will appear in the Image Viewer. By default, the track point is positioned in the center of the image.

STEP B: POSITION POINT IN IMAGE

In the Viewer, position the point over the image feature to be tracked by dragging the crosshairs in the center of the point overlay. The *x,y* values of the Position parameter in the Node Panel will update accordingly. (For detailed descriptions of the individual Viewer tools, refer to “Track Tools in the Image Viewer” on p. 312).

The more detail present in the area you select, and the more distinct the differences in contrast or color, the better RAYZ will be able to track the feature.

TIP:

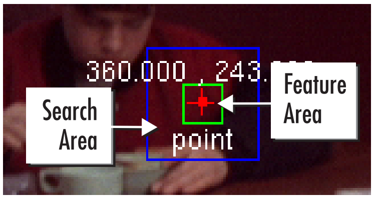
One method you can use to improve a track is to run the image through a filter node, such as Contrast, and adjust it until it is most suitable for tracking, regardless of whether it looks natural or suitable for final composition. You can then input the altered image to the Track node and send the original image through the rest of the network.

STEP C: ADJUST THE TRACK AREA

The track point overlay includes boxes that define the image feature to be tracked (Feature Area) and the surrounding area to search (Search Area):

1. Resize the **Feature Area** box, if necessary, to fit the feature being tracked. The corresponding Size parameter in the Node Panel will update accordingly.

2. Resize the **Search Area** box, if necessary, to suit the image being tracked. The corresponding Search Size parameter in the Node Panel will update accordingly.



17.22 Track Point: Drag crosshairs (red) in center of point to reposition it; drag Feature Area box (green) to resize it; drag Search Area box (blue) to resize it.

The search area must be large enough to cover the distance that the tracked feature will travel from one frame to the next. The larger the search area, on the other hand, the longer the track will take to process.

TIP:

Review subsequent frames to evaluate the extent of movement from frame to frame and set the Search area to accommodate it.

STEP D: SET THE TRACKING PARAMETERS

Each track point entry in the Node Panel provides numerous parameters for adjusting how RAYZ performs the tracking function. In most cases you can use the default settings, adjusting them only if you encounter a problem with the initial track. (For detailed descriptions of the individual Node Panel parameters, refer to "Track Parameters" on p. 315.)

By default, RAYZ will track every frame in the sequence (assuming you start tracking on the first frame). Press the Fit button in the Image Viewer's Flipbook controls to set the range to match the sequence. To track a subset of the total range, use the Range fields in the Image Viewer's Flipbook controls to specify the range of frames to track.

TIP:

You can change the default settings of the parameters in the first track point you create and then copy and paste that point to create other track points with the same settings. (See "Copying, Pasting, and Deleting Layers" in chapter 7, p. 94 if you need more information.)

STEP E: START TRACKING

Actuate the track function by clicking the "play" button in the Tracking Controls in the Image Viewer. The Track buttons enable you to track forward or backward. To stop tracking, press the Stop button. For more information, see "Track Tools in the Image Viewer" on p. 312.

STEP F: REVIEW THE TRACKING DATA

When the node has finished tracking, you can review the track path in the Image Viewer and make any necessary adjustments. Use the Flipbook controls to play the tracked sequence and watch the track point overlays change position from frame to frame.

Track points can be repositioned manually at any frame, and you can re-track if necessary. The tracker will use the keyframes you set to guide it when tracking if User Keyframed is selected in the Prediction menu (as it is by default).

You may want to change the default settings in the Node Panel for the Method, Channel, or Prediction parameters. The descriptions in the "Track Parameters" (p. 315) section explain which settings are best for track features that change shape significantly, move smoothly or jerkily, and so on.

Any track point can be disabled or locked before re-tracking, so that other points can be re-tracked without affecting accurate track data. See also "Locked Point Overlays" on p. 314.

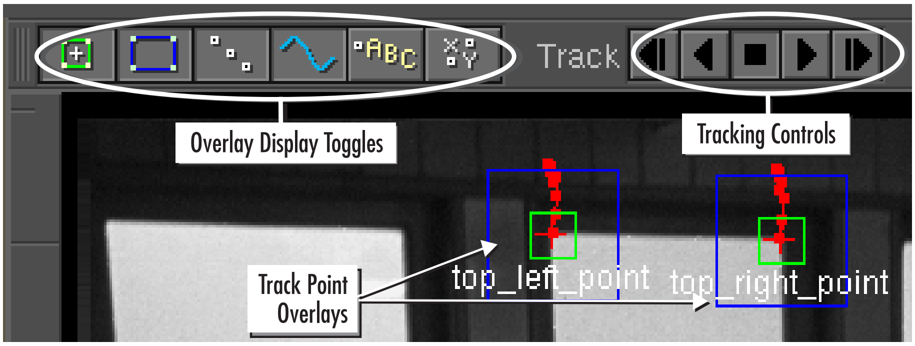
Tracking Features That Leave Frame

In some cases no single feature can be tracked for the entire length of the sequence. If so, you can track a feature until it moves out of frame (or cannot be distinguished for some other reason) and switch the focus of the track point to a different object, using an offset value to compensate for the change. That way you can still generate a single set of track data for the entire sequence.

1. Start by tracking the feature as described in the previous section, setting the frame range in the Flipbook to track only the frames in which you know the object is recognizable. (You can also just track the entire sequence until the node loses track to find out what frame that is.)
2. In the first frame at which the Track node will lose track, Ctrl-drag the Feature box (the green box) of the track point to position it over the new object to be tracked. (Ctrl-dragging the point creates the offset.)
3. If necessary, adjust the size of the Feature and Search area boxes to fit the new feature.
4. Resume tracking to the end of the sequence.

Track Tools in the Image Viewer

The Image Viewer for the Track node provides a tool strip, along with an overlay for each track point in the image. (The Node Controls item must be checked in the Viewer Actions menu to use these Track node tools.)



17.23 When displaying a Track node image, the Image Viewer provides tracking controls and overlays.

Track Point Overlays

Each time you create a new point (using the Points button in the Track Node Panel), a corresponding overlay is added to the Image Viewer. The overlay shows the current settings for the track point parameters that control the size and position of the feature area and the search area.

You can manipulate the overlays directly in the Viewer or use the corresponding parameters for the point in the Node Panel:

- Drag the crosshairs in the center of the point to position it over the feature to be tracked.
- Drag the Feature box to resize it, if necessary, to encompass the image area to be tracked.
- Drag the Search box, if necessary, to change the size of the surrounding area in which RAYZ will search for the track feature.

You can also create offsets using the track point overlays:

- Ctrl-drag the Feature box to adjust the feature offset.
- Ctrl-drag the Search box to adjust the search offset.

Once a feature has been tracked, the overlay can also display the tracking path for the point and the location of the point at each frame.

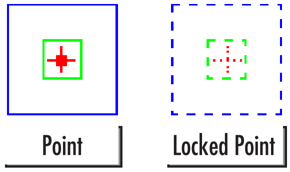
Reference Points

Whenever you reposition a track point manually, the point indicator in the center of the overlay changes to a larger dot at that frame.

This indicates that the point is a reference point at that frame, which is relevant whenever “User Reference Points” is selected in the Reference menu of the point entry in the Node Panel (this option is selected by default). The description of the “Reference” (p. 316) menu explains the significance of reference points.

To change a reference point to a “regular” point, Ctrl-click on the point marker in the center of the point overlay. Ctrl-clicking a regular point, on the other hand, turns it into a reference point.

LOCKED POINT OVERLAYS



17.24 The overlay for a locked point is drawn in the Image Viewer using a dotted lines.

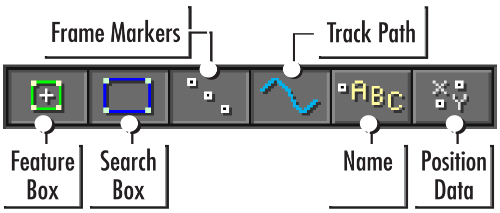
You can lock any point using the Lock button (padlock icon) in the track point entry in the Node Panel. While a point is locked, it cannot be modified, which means that you won't accidentally drag an accurately tracked point out of position in the Viewer.

OFFSET LINES

When you Ctrl-drag a point by its Feature box to create an offset (or change the default value in the Offset parameter for the point), a dotted indicator line is drawn from the original position to the offset position.

OVERLAY DISPLAY TOGGLES

The tracking controls in the Viewer tool strip include a group of buttons that control the display of individual elements in the track point overlays.

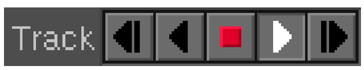


17.25 Overlay Display Toggles: These buttons control the display of individual components of the track point overlays.

Display of the Feature boundary box, the Search Area boundary box, the Track Path, and the Frame Markers can be turned on and off individually. The same is true for display of the track point Name and the Position data for the current frame.

TRACKING CONTROLS

The tool strip also provides the tracking controls, which are used to start and stop the RAYZ tracking function.



17.26 Tracking Controls in Play Forward Mode.

You can track forward or backward from the current frame to the end (or to the start, if tracking backward), or you can track one frame forward or back at a time. Press the Stop button (red) to stop tracking.

FRAME RANGE TO TRACK

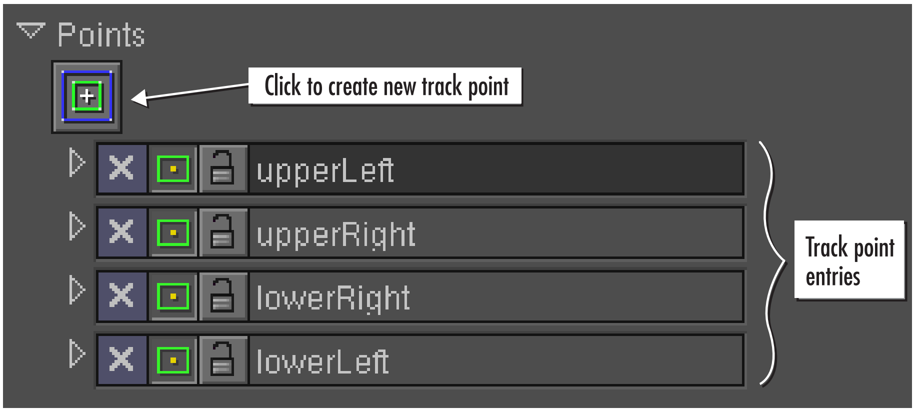
RAYZ will track forward from the current frame to the last frame specified in the Flipbook Range fields. When tracking backward, RAYZ tracks from the current frame back to the first frame specified in the Flipbook Range fields.

If most of the frames in a sequence tracked well, but you need to re-track a subset of frames, use the Range fields in the Flipbook to specify the new frame range to track.

TIP:
To avoid re-tracking points that do not need revision—especially if you have tweaked any of the frame positions by hand—lock those points before re-tracking other points.

TRACK PARAMETERS

The Track Node Panel generates a new track point entry each time you click the Add Point button.



17.27 Track point entries in the Points list of the Node Panel.

Each track point entry can be temporarily disabled or permanently deleted. You can rename and lock a track point and control whether the overlay for a track point is displayed in the Image Viewer, as illustrated below in *Fig. 17.28*.

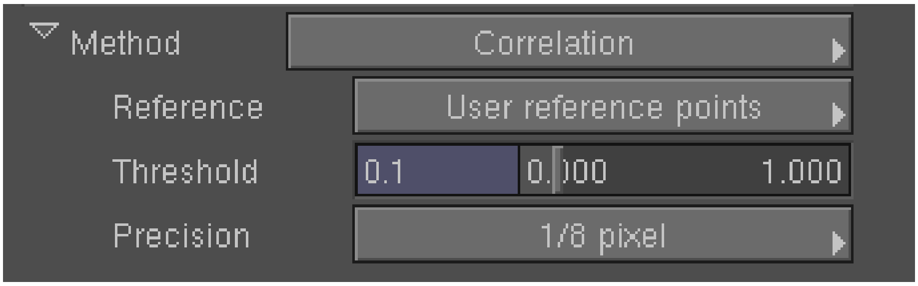
17.28 Top-level track point entry controls. (The dark shading of the entry label indicates that it is selected.)

METHOD

The Method menu is used to select the tracking method RAYZ will use: Correlation or Centroid. The default method, Correlation, is appropriate for most situations. It examines the difference between two frames to find the best correlation with the feature being tracked.

The Centroid method, on the other hand, is useful for tracking a simple shape on a plain, contrasting background such as a track ball that was placed on a bluescreen backing for the express purpose of tracking it in post-production.

The Method parameter can be expanded to access additional controls for tracking.



17.29 Expand the Method group to access parameters that modify the selected tracking method.

REFERENCE

The Reference menu tells the tracker where (from which frame) to get the track feature to use as the reference when searching other frames for a match. The best choice depends on how much the feature being tracked changes across the frame range.

The default selection, **User Reference Points**, searches each frame for a match with the track area in the previous reference frame. (User reference points are created at any frame in which you manually adjust the track point position. For more information about how to designate a reference point, or remove this designation, see "Reference Points" on p. 313.) This works well if the feature being tracked doesn't change shape significantly over time, just its location in the frame.

If the feature being tracked is rotating or zooming, or otherwise changing size and shape, however, **Previous Frame** is a better option. Instead of searching each frame for a match with a reference frame, Previous Frame searches each frame for a match with the previous frame. Although this makes it easier to track an object that changes shape, it may also introduce a little drift.

The other Reference menu options, **Match Above/Below Threshold**, can be a good compromise between User Reference Points and Previous Frame. Threshold matching uses the user reference frame as the reference, until it reaches a frame for which the error value goes above or below a specified amount, or threshold. Then the previous frame is used as the new reference frame until the error value again exceeds the threshold, and so on.

THRESHOLD

The Threshold parameter becomes relevant when a Match Threshold option is selected from the Reference menu. The tracker automatically calculates the amount of error in the correlation between the track feature in the reference frame and the current frame being tracked, and the Threshold parameter specifies a threshold level for this error value. The default Threshold level is 0.1, in a range of 0–1.

Precision

The Precision menu is used to specify the level of precision RAYZ should use to calculate the correlation during tracking, expressed in terms of sub-pixel accuracy, from 1 pixel (the lowest level of precision) to 1/64 pixel (the highest).

Higher precision levels track more accurately, and lower levels track more quickly. The default Precision level, 1/8 pixel, is a good compromise.

Channel

The Channel menu is used to specify which channel of the input image should be used in the track operation. The default is Luminance, however you can select any image channel that will provide the most detail and contrast in the image feature you are tracking:

- If your tracking point is in an area with high contrast and detail, luminance should provide good tracking.
- Alternatively, a point surrounded by pixels with considerably different color values may track better using the Red, Green, or Blue channels.

Prediction

The Prediction menu specifies the method the tracker should use to search each frame; that is, where in the frame it should start looking. The best method to use depends on the type of motion in the sequence.

For naturally moving objects in a filmed scene, Velocity or Acceleration are probably the best options. For jerky motion, such as that caused by camera shake or certain kinds of computer animation, Last Position or User Keyframed should work better.

The default prediction method, **User Keyframed**, means that the tracker will start looking in a frame at the interpolated position of the point, based on any existing keyframes you have set (by positioning a point manually). If no user keyframes are detected, the tracker uses the Last Position method. The **Last Position** method means that the tracker will start looking at the track position of the previous frame.

Velocity prediction uses the speed at which the point was moving in the previous frame to extrapolate its position in the current frame and start searching there. **Acceleration** uses both the velocity and the acceleration rate to predict where the point should be in the current frame.

Overlay Color

This menu is used to specify the color of the track point (and track path, once the point has been tracked) that is displayed in the Image Viewer. This enables you to specify different colors for different track points if necessary to help distinguish them from one another, or simply to pick a color that shows up better against the image being tracked.

Position

The Position parameter specifies the *x,y* coordinate location of the track point at each frame. This data is generated automatically when the point is tracked, however, you can also type values into the fields. (RAYZ will update the position of the track point overlay in the Image Viewer accordingly.)

Offset

The Offset parameter enables you to specify an offset to the Position parameter values by typing *x,y* coordinates into the fields.

You can also create an offset in the Image Viewer by Ctrl-dragging a point by the Feature box (the green box in the point overlay), and the Offset parameter values will change accordingly.

Error

The Error parameter value displays the correlation error of the point at the current frame. It indicates the extent to which the pixel area determined in the Size parameter correlates with the pixel area in the previous frame. The greater the error value, the less accurate the match.

Size

The Size parameter specifies the size of the track area, in pixels. The Size rectangle (the middle rectangle of the track point) should fit as closely as possible the feature to be tracked. The default size is 16 x 16 pixels.

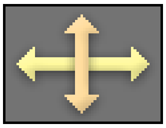
Search Offset

This parameter enables you to offset the position of the search area, as defined in the Search Size parameter.

Search Size

The Search Size parameter specifies the size of the surrounding area to search, in pixels. The search size is represented by the outermost rectangle of the track point. The default size is 48 x 48 pixels. The search area needs to be large enough to get a good correlation from frame to frame; however, the larger the search area the longer the track will take to process.

Transform Node

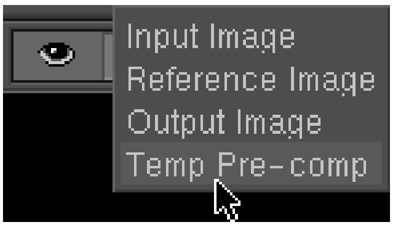


The Transform node enables you to rotate, translate, or scale an image within the frame area. You can use Transform to animate the motion of one image relative to another or to simulate camera moves such as pans or zooms or complex shots that integrate pans, zooms, translations, and rotations.

The Transform node accepts one or two inputs: the foreground image and an optional reference image input.

Using the Reference Input

The reference image establishes the frame size of the output image and acts as a visual reference for the transform operation when Temp Pre-Comp view is selected from the Source menu. This view shows the primary input in a temporary composite over the reference input.



17.30 Temp Pre-Comp is selected by default in the Image Viewer's Source menu when an image is connected to the reference input of the Transform node.

Use the Source menu to view the output image or the temporary pre-comp (or use the Source menu hotkey, which is S, to cycle through all available views).

Interactive V. Numeric Controls

It is important to note that Transform is different from other nodes that offer interactive overlays in the Image Viewer. In other nodes, the overlay in the Image Viewer and the corresponding parameters in the Node Panel are completely interdependent, in that any change in one is reflected exactly in the other.

In Transform, however, the overlay behavior is optimized for performing transforms interactively, where each change updates the image in order, while the Node Panel parameters are optimized for complex animations of the rotation and scale parameters over time.

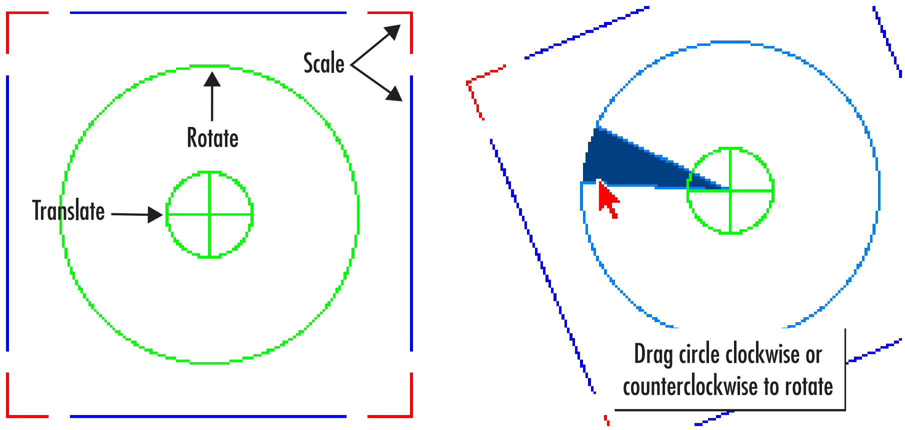
For example, the Pivot parameter in the Transform Node Panel applies to both Scaling and Rotation. This enables the pivot value to use track point data and rotate and scale around the same position, producing the desired result in almost all cases.

When manipulating the pivot interactively in the overlay, however, this would often result in unexpected behavior—the entire image would move when the scale was recalculated to account for the pivot value change. RAYZ prevents this by actually offsetting the current translation value (this is evident in the Translate parameter in the Node Panel), with the result that the image doesn't "jump" in the Viewer.

The bottom line is this: RAYZ assumes that you don't want the pivot value to control the scale operation when using the interactive tools and it manipulates the actual Node Panel parameters accordingly to produce the equivalent result. And if, on the other hand, you use the Transform node to animate a complex acrobatic maneuver using tracking data or expressions, it will be much easier to use the Node Panel parameters to do so.

USING THE TRANSFORM OVERLAY

When the Transform node is displayed in a Viewer, a transform tool appears as an overlay on the image. You can use the mouse to manipulate the transform overlay in the image.



17.31 Transform Overlay: an interactive overlay in the Image Viewer that is manipulated by dragging.

TRANSLATE To translate the image, drag the center of the transform tool to a new position.

TIP:

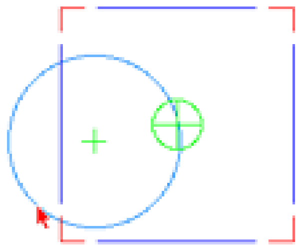
Use the arrow keys on your keyboard to **nudge** the image in one-pixel increments in the direction indicated by the arrow. Hold down the Shift key as you press an arrow key to move in larger increments (4 pixels).

To change the default increment values, go to Edit > Preferences > Settings and select Nudge Size (Small) or Nudge Size (Large).

SCALE Drag inward to scale down; outward to scale up. To scale the image

- in X (width), drag the edge on the left or right.
- in Y (height), drag the edge on the top or bottom.
- in X and Y, drag any of the corners. (To scale proportionally, hold down the Shift key as you drag a corner.)

ROTATE To rotate the image, drag the circle in the transform tool clockwise or counterclockwise. The rotation circle will animate to show you the degree of rotation.

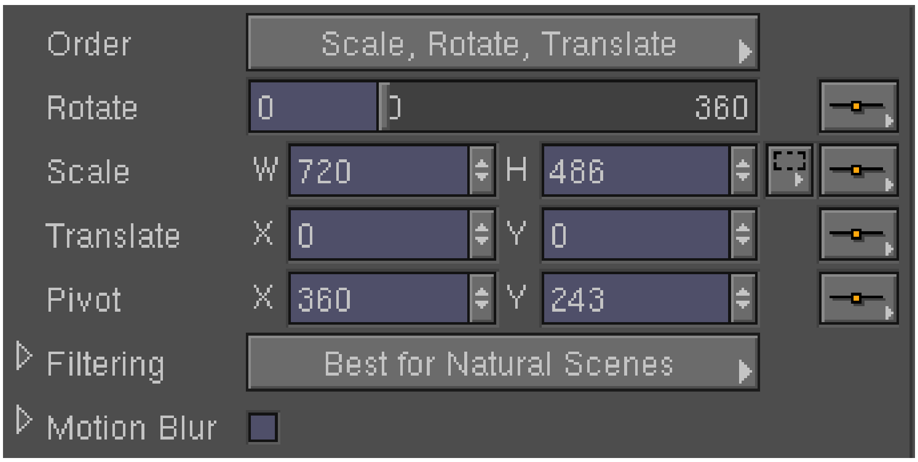


17.32 Ctrl-drag circle to move pivot point.

Pivot By default, the pivot point for the rotation is in the center of the image. To change the pivot, hold down the Control key and drag the rotation circle to a new position. The "x" in the center of the rotation circle marks the new pivot point.

Transform Parameters

The numerical parameters in the Transform Node Panel are Rotate, Scale, Translate, and Pivot.



17.33 Transform parameters in the Node Panel.

The Rotate parameter value is expressed in degrees and the others are expressed, by default, in pixel units.

If you prefer, you can change the display units for a parameter from pixels to fractional units by checking the Float Display item in the Parameter menu (located at the right end of the parameter). For example, you might prefer to use pixel values for the Translate parameter to move the image 10 pixels horizontally, and float display for the Scale parameter to scale the image down to 80 percent (0.8) of full size.

However the parameter values are displayed, RAYZ always stores them at floating point precision to accommodate the use of proxies and clones.

Order

This menu enables you to specify the order in which the transformation operations will be computed. The default order—scale first, then rotate, then translate—is likely to be the best choice in the majority of cases, where the pivot point will be at the center of the image being transformed.

TIP:

For transformations where the pivot will be located at a corner (such as a swinging arm motion), try using the "Translate, Rotate, Scale" order.

ROTATE

The Rotate parameter enables you to rotate the image by specifying a value which is expressed in degrees, with the default value of 0 representing no rotation. A positive value rotates the image in a counterclockwise direction; a negative value rotates the image in a clockwise direction. The rotation is centered around a pivot point that you specify by using the Pivot parameter.

NOTE:

The Rotate parameter range is unconstrained to accommodate animation of the rotation over time, as to create a spinning or spiraling effect.

SCALE

The Scale parameter enables you to scale the input image up or down by keying appropriate values into the Width and Height fields. By default these fields are set such that no change occurs to the input image.

The Width and Height values can be displayed as scale factors (using the Float Display option in the Animation menu) or as pixel values. Float display is the easiest way to scale proportionally, as you can enter the same value in both fields. For example, full size (no change to the input) is expressed by a float value of 1.0 in each field, so entering 0.5 in each field will result in an image that is half as high and half as wide as the input.

The Scale parameter also includes a Size menu, which you can use to select specific dimensions from a list of common film and video resolutions.

TRANSLATE

The Translate parameter is used to translate the image—to move it along the X and Y axes. By default the X and Y parameter values are set at 0, which represents no change to the input image.

A nonzero value in the first field shifts the image along the X axis (horizontally), and the second field shifts the image along the Y axis (vertically). Positive values shift the image to the right and upward; negative values shift the image to the left and downward. The values may be displayed in floating point (as a percentage of the total image width and height) or in pixels, and the parameter range is unconstrained.

For example (using float), values of 0.5 and 0 will shift the image to the right a distance equal to one-half the width of the image. To shift the image to the left by an equal amount, use values of -0.5 and 0.

Pivot

The Pivot parameter specifies *x,y* coordinates that represent the pivot point to be used in conjunction with the Rotate and Scale parameters (see above). The pivot point is the center of the rotation and/or scaling.

The Pivot parameter defaults to the center of the image (0.5 in both fields, assuming values are displayed in floating point). The lower-left corner of the image is the 0,0 point; the upper-right corner of the image is 1,1. The Pivot parameter range is unconstrained.

Filtering

Use this menu to select the filtering option to use to reduce aliasing and other artifacts that may be introduced by the node operations. The "Best for natural scenes" and "Best for CG scenes" options are recommended choices; however, you can have complete control over the options by selecting "Advanced - User Set" from the menu. This is explained in detail in "Filtering Transformations" on p. 278.

Motion Blur

Check this box to add motion blur. You can expand the Motion Blur group to access parameters that control the phase and duration of the effect, as explained in "Adding Motion Blur to Transformations" on p. 280.

chapter 18

COMPOSITE NODES

The Composite nodes enable you to combine two or more image sequences in a variety of ways.

IN THIS CHAPTER

Multilayer Composite Nodes

The most versatile and commonly used composite node is Multi-comp. The **Multi-comp** node is the "Swiss army knife" of composite nodes. You can composite an unlimited number of image layers in one node, using the operator you specify for each layer of the comp—over, multiply, screen, and so on. The section on "Choosing a Composite Operator" (p. 332) includes a comparison chart of all the Multi-comp composite operators. In addition, each layer can be transformed individually to position it in the context of the entire composite.

The other multi-input node, **Z-comp,** is used instead of Multi-comp when the image layers to be composited include z-depth channel data.

Binary Composite Nodes

You can't beat the convenience and flexibility of using Multi-comp for multilayer composites. For a simple two-layer composite, however, you also have the option of using a binary (two-input) composite node.

The **Over, Under, Atop, Inside,** and **Outside** nodes composite two image inputs based on the alpha coverage, as specified in each node description.

The **Add, Subtract, Multiply,** and **Difference** nodes, on the other hand, composite two images without regard to alpha coverage; in fact, the inputs to these nodes need not include alpha channels.

The **Dissolve** node performs a dissolve between two inputs, while the **MinMax** node compares two inputs and uses the minimum or maximum value in the output, as you specify.

Ultimate AE is designed to composite an Ultimatte foreground image over a background while controlling various edge characteristics.

Compositing Disparate Input Images

You can composite input images that differ in spatial resolution, frame range, and bit depth, and RAYZ will use the following criteria to determine the output:

The **bit depth** of the output is determined by the input with the highest bit depth. (The only exception would be the Ultimatte AE node, assuming that the optional garbage matte input is connected, as the bit depth of the garbage matte is irrelevant.) RAYZ actually converts all of the inputs to floating point, or fractional, values internally to perform the node operations (see "Composite Formulas" on p. 327). Then the result is converted back to 8- or 16-bit integer for output as needed to match the input data.

The **frame size** of the output is determined by the *bottom input* to the Over, Atop, Inside, Outside, and Ultimatte AE nodes, based on the assumption that the bottom input is the background image. For the same reason, the *top input* determines the output size for Multi-comp, Z-comp and Under. The top input also controls the output of the remaining composite nodes: Add, Difference, Dissolve, MinMax, Multiply, and Subtract.

The **frame range** of the output is determined by the same input that controls output size (see previous paragraph). This means that you will only generate an error if you navigate outside the range of the controlling input. If you navigate to a frame at which the other input is out of range, the composite node will use the controlling input image for the output. In the case of the multilayer composite nodes, Multi-comp and Z-comp, the node ignores any input layer that does not have image data at the current frame and comps the rest of the inputs.

Formulas and Terminology Used for Composite Nodes

Composite Formulas

In the descriptions of the various composite nodes, we resort to the following notation for composite formulas. "A" denotes the A image, the top layer, or foreground (the top input to a binary composite node); and "B" denotes the B image, the bottom layer, or background image. The lowercase letters *r, g, b,* and *a* represent the red, green, blue, and alpha channels. For example, "Aa" would refer to the alpha channel of the A image.

All multiplication in these formulas is fractional; that is, at floating point precision. For 8-bit and 16-bit per channel images, this means that the pixel values of each channel are converted into decimal fractions. For example, a 16-bit pixel with RGB values of [65535, 52428, 0] would be converted to [1.0, 0.8, 0].

Alpha Channel

The alpha channel contains the opacity information for each pixel of the RGB channels it accompanies. In RAYZ, a value of 0 represents total transparency (no coverage) while a value of 1 represents complete opacity (full coverage). The alpha channel is displayed in the Image Viewer as a monochrome image where white represents a value of 1 and black is 0.

NOTE:

When a composite node includes an Opacity parameter for an input, the value of each pixel in the alpha channel is multiplied by the Opacity parameter value. This means that such a parameter can be used to proportionally increase or decrease the overall opacity level of that image layer in the composite.

Alpha v. Matte

In reference to the composite nodes, the terms *alpha channel* and *matte* can be considered synonymous.

Alpha Coverage

The term *alpha coverage,* as in the phrase "wherever the image has alpha coverage," refers to any pixel in an image where the alpha channel value is greater than zero. The concept of coverage is central to composite operations where the alpha channel value determines the extent to which the RGB channels contribute to the result, such as the over, under, inside, outside, and atop operations.

ABOUT PREMULTIPLICATION

A premultiplied image is one in which, for every pixel, the value of each color component (RGB) has been premultiplied by the alpha component and stored in the color component. This transfers the opacity level represented by the alpha channel to the RGB components themselves before the RGB channels of two separate images are blended in a compositing operation.

As a general rule, computer-generated images, such as those from a 3D animation package, have been premultiplied while digitized frames of film footage, such as Cineon files, have not.

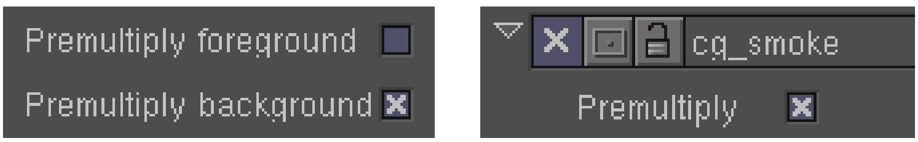
The premultiplication status of an image is important because premultiplication is an initial step in many compositing operations. An input image to a composite node will always be premultiplied by the node, unless it has already been premultiplied. Normally, a premultiplied image should *not* be premultiplied again, although it is certainly possible that this could create a desired effect in some cases.

NOTE:

RAYZ also provides nodes in the Conversion menu that are dedicated to the tasks of premultiplication and its reversal: the "Premultiply Node" (ch. 20, p. 410) and the "Unpremultiply Node" (ch. 20, p. 411).

PREMULTIPLY PARAMETERS

RAYZ automatically evaluates each input to determine its status and sets the Premultiply parameter in the Node Panel accordingly. For example, an RGBA image that was imported into the Image In node is assumed to be premultiplied, while an RGB image for which the alpha was generated in a matte node is assumed to be unpremultiplied. One exception is the output from an Ultimatte node, which is premultiplied.

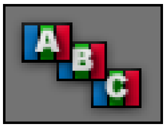


18.1 The Over node settings (left) indicate that the node will premultiply the background input but not the foreground. The layer shown in the Multicomp node example (right) will not be premultiplied.

TIP:

You can override the premultiplication setting for any input, if necessary, in the composite Node Panel. If you don't get the results you expect from a composite node, double-check the premultiplication settings.

MULTI-COMP NODE



The Multi-comp node enables you to composite a virtually unlimited number of image layers in one node.

You simply connect imagery from upstream nodes to the input connectors of the Multi-comp node. Each time you connect a new input to the Multi-comp node, another input connector is automatically created, and at the same time, a corresponding layer entry is created for that input in the Multi-comp Node Panel.

Each layer entry provides parameters that control how that layer is composited into the output image, including the type of operation and opacity level to use. You can even transform any layer using numeric or interactive overlay controls.

NOTE:

Because the Multi-comp node does not have a set number of inputs, the first, or top, input image is actually the bottom, or background, layer in the composite. Each subsequent input is layered on top of the previous layer in the composite image.

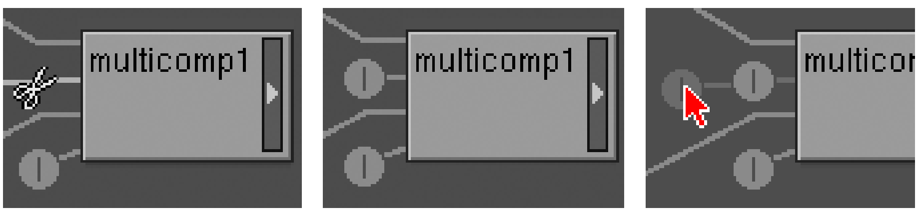
Each input can be a different size and bit depth. Lower bit depth inputs are promoted to the highest bit depth, and the output of the Multi-comp node will match the highest input bit depth. The output size is determined by the first (background) input.

CHANGING INPUTS

You can change the order of the existing inputs to a Multi-comp node, and you can also replace one input with a different input image, which will appear in the same layer of the composite as the old image.

To change the order of an input in the composite, drag the layer button with the double-arrow icon up or down in the layer list. See *Fig. 18.3*.

To replace the input to a layer with a different image, you can disconnect the input where it flows into the node (Ctrl-click on the connector line). This will leave a connector hanging off the node which you can use to connect a different input image.



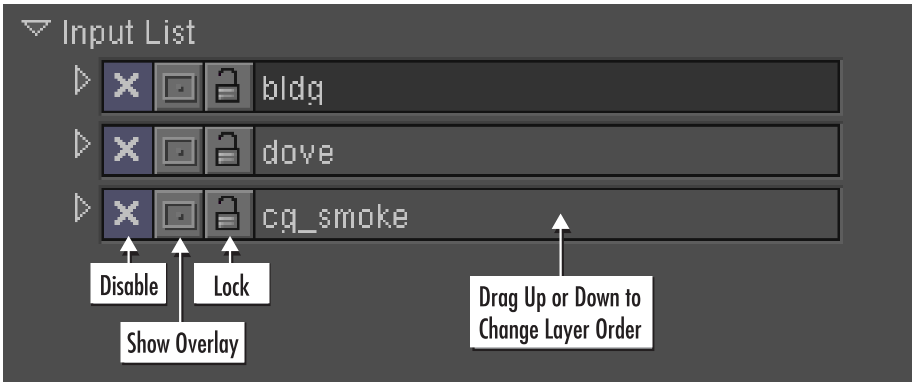
18.2 Ctrl-click connector line (left) to disconnect input (middle) and drag free connector to connect new image (right).

In this way you can replace the input image with another image at the same level in the stack of image layers without having to reorder the layers manually. This also enables you to keep the parameter values set for the old image so that, for example, the replacement image will be transformed in the same way as the old image.

Deleting Layers

To delete a layer altogether, select the layer and use the Layer Actions menu. (See "Copying, Pasting, and Deleting Layers" in chapter 7, p. 94 if you need more information.) This disconnects the corresponding input, deletes the layer entry, and moves the subsequent layers up.

Be aware, however, before you delete the background layer that the background image sets the frame size and range of the output. You may want to disable the background instead of deleting it, or replace it with another input image.



18.3 Top-level controls available in the Node Panel for each layer of the composite.

Disabling Layers

Any layer can be disabled temporarily without disconnecting its input from the node by clicking the checkbox in the top-level controls for the layer entry. When the box is checked, the layer is included in the composite, and when it is not, it is as if the layer did not exist (at least until you turn it back on again).

If you disable the background layer (the top layer in the Node Panel), the layer disappears, in that all pixel values become 0, but the input image to that layer still determines the size and range of the output.

How Multiple Layers Are Composited

Each successive layer becomes the A image in a composite, with the B image being the result of all the composite operations performed on preceding image layers in turn.

Assume, for example, that you have a five-layer composite in which the first entry in the Node Panel list (which is the bottom image layer in the composite) is layer 1, the next entry is layer 2, and so on to the last entry in the list (the top image layer in the composite), which is 5.

If you select the Over operator for layer 3, layer 3 becomes the A image in an "A over B" composite in which the B image is the result of the whatever type of composite was performed on layers 1 and 2. Then layer 4 is composited as the A image with the result of the A over B composite in layer 3, and so on.

Multi-comp Parameters

When you create a new Multi-comp node, the Node Panel is blank until you connect an input. A new layer entry is created in the Node Panel for each input you connect to the node.

Each layer can be reordered, temporarily disabled, or deleted using the top-level layer controls, as described previously in "Changing Inputs" (p. 329). And each layer can be expanded to access additional parameters: Premultiply, Opacity, Composite, and Transform, as described next.

18.4 Multi-comp layer expanded to access layer parameters.

Premultiply

RAYZ automatically sets this parameter based on the nature of the incoming image. This checkbox gives you the ability to override the setting if necessary. See also "About Premultiplication" on p. 328.

If the box is checked, RAYZ will premultiply the image; if the box is unchecked, RAYZ will not premultiply it. When an input does not have an alpha channel, the Premultiply parameter is grayed out.

Opacity

The Opacity parameter enables you to proportionally increase or decrease the opacity of the layer as a whole. The default value of 1 represents no change to the opacity of the input. Values greater than one will increase the opacity where the image is not already fully opaque, while values less than 1 will decrease the opacity where the image is not already fully transparent. For example, a value of 0.5 would reduce the opacity of the layer by 50 percent.

Composite Type

Use the Composite menu to select the type of composite operation to perform on the current layer. The default is the commonly used Over operator, however you can also choose Under, Atop, Inside, Outside, Add, Subtract, Multiply, and Difference, as well as Screen, Overlay, Soft Light, Hard light, Lighten, and Darken.

Some composite operators require an alpha channel and others do not. Be aware, however, that once an RGBA layer is added to the composite, all subsequent layers in the list will also require an alpha.

Choosing a Composite Operator

The following table lists and describes the available composite operations in Multi-comp. "A" refers to the current layer, the layer with the Composite menu you are currently setting, while "B" refers to the layer under it in the composite. The "B" image will itself be a composite of all the previous layers in the Multi-comp node. (See also "How Multiple Layers Are Composited" on p. 330.)

Layer Order The order of the layers affects the result for all of the alpha-centric operators—Over, Under, Atop, Inside, and Outside—as well as for Subtract, Hard Light, Soft Light, and Overlay.

The Screen, Overlay, Hard Light, Soft Light, Lighten, and Darken operators are analogous to blending modes in Gimp or Photoshop. They are frequently used to composite translucent objects such as smoke and fog into a scene (the Screen operator, e.g.) or to accentuate or de-emphasize some aspect of an image—often by layering the same image data over itself or by layering it with a grayscale ramp or similar input.

See also *Fig. 18.6* (p. 336) and *Fig. 18.7* (p. 338) for examples of the effect various operators have on the same pair of images.

OPERATION	DESCRIPTION	APPLICATION
Over	A + B * (1 - Aa) A over B wherever the A layer has alpha coverage	The most common composite operator; selected by default
Under	B + A * (1 - Ba) B over A wherever the B layer has alpha coverage	The same as Over with layer order reversed
Atop	A * Ba + B * (1 – Aa A over B wherever the A layer has alpha coverage, but only where A falls inside the alpha coverage of B	Restricts the layer to appearing only where the layer under it is also visible
Inside	A * Ba Only the A layer is visible in result; A appears wherever it falls inside the B alpha	Hides all lower layers, the combined alpha of which becomes the matte for the current layer

OPERATION	DESCRIPTION	APPLICATION
OUTSIDE	A * (1 – Ba) Only the A layer is visible in result; A appears wherever it falls outside the B alpha	Hides all lower layers, and the current layer only appears where there is no alpha coverage in the lower layers
ADD	A + B A and B layer values are added	The resulting image is lighter; can be used to emphasize highlights
SUBTRACT	A - B B layer values are subtracted from A	Darkens the resulting image most wherever the B layer is brightest
MULTIPLY	A * B A and B layer values are multiplied	Drops out white; can be used to emphasize shadows
DIFFERENCE	|A - B| Calculates the difference—the absolute value of the result of subtracting B from A	Creates a dramatic, “color negative” type of effect
LIGHTEN	max(A,B) Compares the A and B layer values and uses the higher value	Always uses the lighter value of the two layers for the result
DARKEN	min(A,B) Compares the A and B layer values and uses the lower value	Always uses the darker value of the two layers for the result
SCREEN	1 - (1 - A) * (1 - B) The A and B layers are inverted, multiplied, and their product is inverted	Drops out black; can be used to emphasize highlights
OVERLAY	Darkens pixels where B layer luminance is less than 0.5, and lightens pixels where B layer is greater than 0.5.	Uses highlights and shadows in the B layer to control brightness of the resulting image
HARD LIGHT	Darkens pixels where A layer luminance is less than 0.5, and lightens pixels where A layer is greater than 0.5.	Uses highlights and shadows in the A layer to control brightness of the resulting image; effect is similar to using multiply on dark areas and screen on light areas
SOFT LIGHT	Darkens pixels where A layer luminance is less than 0.5, and lightens pixels where A layer is greater than 0.5.	Uses highlights and shadows in the A layer to control brightness of the resulting image; effect is similar to using dodge and burn techniques

TRANSFORM

Expand the Transform parameters when you want to adjust the position of an element in the composite. The Multi-comp Transform parameters are identical to those in the Transform node. Refer to the description of the “Transform Node” in chapter 17 (p. 319) if you need more information about using them.

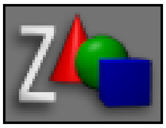
18.5 The interactive Transform overlay for this layer will be active in the Image Viewer.

You can also use the Transform overlay tool in the Viewer to transform the layer interactively, as described in "Using the Transform Overlay" in chapter 17 (p. 320). To display the overlay, press the Overlay toggle button in the layer button list (the button icon will turn green).

Locking Layer Transforms

Press the Lock button for any layer to lock the current Transform settings (the padlock icon on the button turns red when locked). This will prevent you from accidentally moving the Transform overlay when it is displayed in the Viewer.

Z-COMP NODE



The Z-comp node enables you to use both z-depth and alpha channel information to composite multiple layers of imagery. Z-comp requires two inputs per image layer: an RGBA input and a single-channel z-depth input.

NOTE: If your images do not contain z-depth data, use the "Multi-comp Node" (p. 329) to do multilayer composites.

The Z-comp node composites the input layers in a series of over operations, using the values contained in the z-depth channel of each input image to determine, for each pixel, the order in which that pixel will be layered in the composite. You can specify whether larger or smaller z-depth values are nearer to the camera.

Z-COMP PARAMETERS

A *single* layer entry is created in the Input List in the Z-comp Node Panel for each *pair* of inputs (one RGBA input and one z-channel input) you connect to the node.

You can change the order of a layer in the list, to change the corresponding order of the image layers in the composite. The order of the layers in the list is superseded by the z-depth channel values, however, so layer order is only relevant when z-depth values are equal.

INPUT LAYER PARAMETERS

Using the top-level layer controls, you can disable a layer temporarily to see what the composite would look like without the contribution of that pair of image inputs, or delete the layer altogether.

You can expand any layer to specify whether the RGBA input should be premultiplied and adjust the overall opacity of the layer.

PREMULTIPLY

When checked, the Premultiply parameter specifies that the RGB channels will be premultiplied by the alpha channel. This parameter should only be checked when the input has not already been premultiplied. See also "About Premultiplication" on p. 328.

OPACITY

This parameter can be used to proportionally increase or decrease the opacity of the matte as a whole. The alpha channel value of each pixel is multiplied by the value set in the Opacity parameter, which means that the default value of 1 makes no change to the opacity of the input layer.

LARGER Z-VALUES ARE CLOSER

Set this parameter, which applies to all layers, to match your z-depth data. If larger values in the z-depth channel represent pixels that are closer to the camera, leave the box checked, as it is by default. If smaller values represent pixels that are closer, uncheck the box.

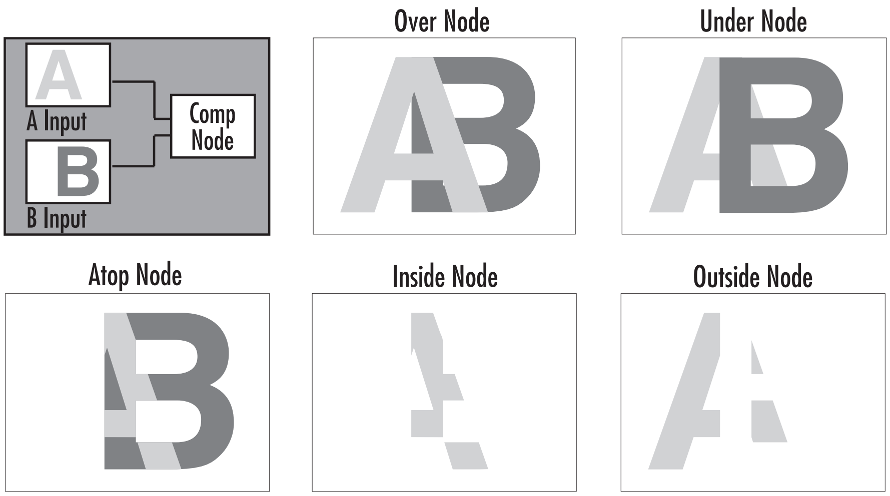
OVER, UNDER, ATOP, INSIDE, AND OUTSIDE NODES

These are binary composite nodes; that is, they take only two inputs each. Both inputs can be RGBA images, however, the alpha channel is only required for the B input, with two exceptions: the Over node, for which the A input requires an alpha channel, and the Atop node, for which both inputs require an alpha channel.

NOTE:

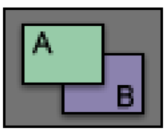
When the inputs are different sizes, the Over, Outside, Inside, and Atop nodes will use the spatial resolution of the B input as the output frame size, even if the B input is smaller than the A input. The exception is the Under node, which outputs an image the same size as the A input.

The computations performed by these composite operators will be preceded by a premultiplication step, unless the input is already premultiplied. The checkboxes in the Node Panel indicate whether each input will be premultiplied by the node, and you can change the setting by clicking the box. For more information, refer to "About Premultiplication" (p. 328) at the beginning of this chapter.



18.6 How the Over, Under, Atop, Inside, and Outside nodes composite the same pair of inputs.

OVER NODE

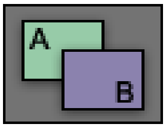


The Over node places a foreground image over a background image wherever the A image has alpha coverage, which means that only the A input requires an alpha channel.

The Over node performs the following computation:

A + B * (1 - Aa)

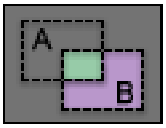
Under Node



The under operation is identical to the over operation, described above, with the inputs reversed (which means that only the background requires an alpha channel):

B + A * (1 - Ba)

Atop Node

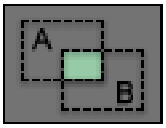


The Atop node places the A image over the B image wherever the A image has alpha coverage, but only where the A image falls inside the alpha coverage of the B image.

The Atop node accepts two RGBA inputs and performs the following computation:

A * Ba + B * (1 – Aa)

Inside Node



The Inside node places the A image wherever the A image falls inside the alpha coverage of the B image. However, none of the B image is visible in the result (even where the A image has no alpha coverage). Only the B input requires an alpha channel.

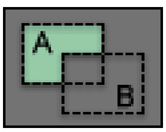
The Inside node performs the following computation:

A * Ba

TIP:

Effectively, the B image acts as a matte, and the Inside and Outside nodes can be used to create complementary mattes from the same B image.

Outside Node



The Outside node places the A image wherever the A image falls outside the alpha coverage of the B image. However, none of the B image is visible in the result, even where the A image has no alpha coverage. Only the B input requires an alpha channel.

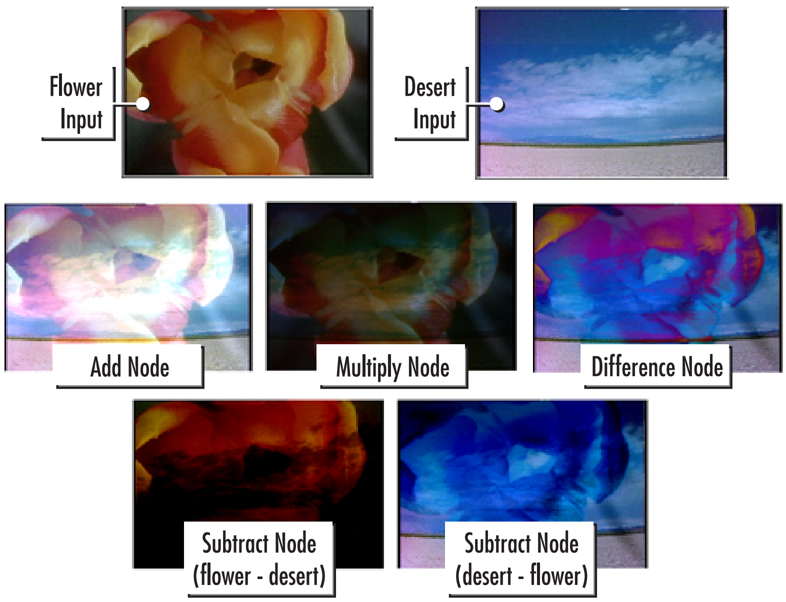
The Outside node performs the following computation:

A * (1 – Ba)

Add, Subtract, Multiply, and Difference Nodes

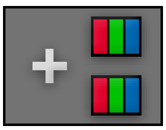
These four nodes composite two images based on the RGB values of two input images as specified in the following descriptions. Unlike the nodes discussed in the previous section (“Over, Under, Atop, Inside, and Outside Nodes” on p. 336), the composites performed by these nodes are not based on alpha coverage.

Due to the nature of the operations, the two required inputs must have the same number of channels. The order in which the inputs are connected to the nodes (i.e., to the top or bottom input connector) is irrelevant to the node operation, with the exception of the Subtract node, as described below. However, the input order does determine the size of the output image, when the inputs are different sizes: the A (top) input size is always used for the output.



18.7 How the Add, Subtract, Multiply, and Difference nodes composite the same pair of inputs.

Add Node

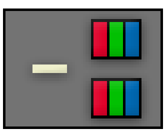


The Add node adds the values of the A image to the values of the B image:

A + B

The Add node results in higher RGB values, creating a lighter composite.

Subtract Node

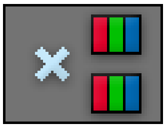


The Subtract node subtracts the values of the B image from the values of the A image:

A - B

Subtract darkens the output image most where the B input is brightest.

Multiply Node

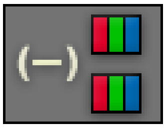


The Multiply node multiplies the values of the A image by the values of the B image:

A * B

Multiply accentuates the dark areas of the composited image, and can be used to emphasize shadows.

Difference Node



The Difference node calculates the difference between the A and B images. The difference is the absolute value of the result of subtracting image B from image A:

|A - B|

This means that Difference has the effect of subtracting the lower pixel value from the higher, whether the higher value is in the A image or the B image. For example:

[0.5, 0.25, 0.75] - [0.5, 0.5, 0.5] = [0.0, 0.25, 0.25]

The Difference node can result in a dramatic, colorful composite that is difficult to predict unless one of the inputs is a grayscale image.

TIP:

The Difference node may be most useful as a diagnostic tool to compare two similar images. You can easily pinpoint how much of a difference a particular operation made, and where, by inputting the same imagery from two different nodes to a Difference node.

DISSOLVE NODE

The Dissolve node performs a cross-dissolve from the A image (top input) to the B image (bottom input). The dissolve is calculated by multiplying the A image by the dissolve value (V) and the B image by the inverse of the dissolve value and then adding the results:

[A * V] + [B * (1 – V)]

DISSOLVE PARAMETERS

DISSOLVE

In the Dissolve parameter, the value you specify controls the transparency of the A image: a value of 0 represents a complete dissolve to the B image, a value of 0.5 is a 50 percent dissolve, and a value of 1 represents no dissolve from the A image.

MinMax Node

The MinMax node combines two inputs by comparing each channel on a pixel-by-pixel basis and selecting which values to use in the output based on the Node Panel settings. For each output channel, you can specify whether to use the lower (MINimum) or higher (MAXimum) values, or the A input or B input values.

The most common use for the MinMax node is to combine two mattes by selecting the maximum alpha channel values of the two inputs. In this case, the RGB channels are all set to one of the two inputs, either A or B.

MinMax Parameters

An entry is created in the MinMax Node Panel for every channel in the inputs. Each entry consists of a row of buttons labeled Input A, Input B, Minimum, and Maximum.

Input A

Depress the Input A button to use the channel data of the top input image as the output value for the referenced channel.

Input B

Depress the Input B button to use the channel data of the bottom input image as the output value for the referenced channel.

Minimum

Depress the Minimum button to use the minimum value of the two inputs as the output value for the referenced channel.

Maximum

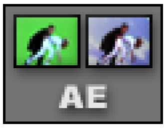
Depress the Maximum button to use the maximum value of the two inputs as the output value for the referenced channel.

Output Menu

Use this menu when the inputs to the MinMax node do not have the same number of channels. It specifies whether the output image should have the same number of channels as the input with fewer channels (Minimum Input Channels) or the input with more channels (Maximum Input Channels).

When Maximum Input Channels is selected, as it is by default, and the node tries to compare a channel that is missing in one of the inputs, the existing channel data will be used for the output regardless of the parameter setting for that channel.

ULTIMATTE AE (ADVANTEDGE) NODE



Ultimatte AdvantEdge is a composite node optimized for foreground imagery processed with the Ultimatte node. It can correct edge artifacts and matte lines created by a number of factors including lighting problems, unwanted shadows, and detail generator enhancement settings often used on video and telecine equipment.

The AdvantEdge node would replace the composite node used in the sample Ultimatte network illustrated in "Overview of the Ultimatte Process" in chapter 15, p. 220.

NOTE:

The Ultimatte AdvantEdge composite node does not have a premultiplication parameter because it is designed specifically for compositing the output of an Ultimatte matte node. Ultimatte AdvantEdge is not intended for use with an unpremultiplied foreground image.

HOW ADVANTEDGE WORKS

The AdvantEdge node works by identifying darker areas of the edges of a foreground matte that may represent undesirable edge artifacts. Because you may wish to preserve some of these darker areas while suppressing others, the node offers a wide range of controls to vary the strength, position, and direction of the filtering process that corrects edges of the matte.

ADVANTEDGE PARAMETERS

The AdvantEdge parameters allow you to specify how the edges of the foreground image will be adjusted in the composite.

HORIZONTAL & VERTICAL CHECKBOXES

These two checkboxes, which are on by default, specify the direction in which the node should search for edge-correction criteria. However, you can turn either parameter off to limit the search direction if you wish. And you can turn both parameters off to see what the composite would like without the edge correction.

PARAMETER	OPERATION PERFORMED
HORIZONTAL (SEARCH)	Detects and corrects edges that run *vertically*
VERTICAL (SEARCH)	Detects and corrects edges that run *horizontally*

HORIZONTAL

The Horizontal parameter specifies that the correction and smoothing operations search horizontally; that is, that the left or right inside edges be searched for edge-correction criteria. This means that Horizontal AdvantEdge will correct edges that run vertically in the image.

Vertical

The Vertical parameter specifies that the correction and smoothing operations search vertically; which means that the upper or lower inside edges are searched. Vertical AdvantEdge will correct edges that run horizontally.

Position

The Position parameter adjusts the center of the edge identified for correction by moving the edge away from or toward the outside of subject areas. This may help cover problem areas without increasing the overall thickness of the edge correction area.

Width

The Width parameter sets the width, or thickness, of edge correction. Increasing Width control too far may cause detail loss in the edge area.

Correction

The Correction parameter sets the strength, or amount, of gray/black suppression in the matte edges. Adjust this control until problem edges disappear. Advancing Correction too far may cause loss of detail within the edge area.

Smoothing

The Smoothing parameter smooths hard or jagged object edges in the direction specified by the Horizontal and Vertical parameters. Smoothing is not often recommended, as it can degrade fine detail.

Interlace

This parameter, when checked, configures AdvantEdge for interlaced video images that have been split into frames of even or odd field lines. Because the AdvantEdge filtering works over several field lines, you must specify interlaced input for images that contain the single-line gaps so that the node doesn't try to interpret the data in the irrelevant field.

G: Matte Channel

This menu becomes active when an optional garbage matte input is connected to the AdvantEdge node. It is used to specify which channel of this input to use as the garbage matte, when the image has more than one channel. See also "Using a Garbage Matte" (ch. 15, p. 224) in the Ultimatte node description.

chapter 19

FILTER NODES

The Filter nodes menu includes a variety of filters for blurring and sharpening, adding and removing grain, embossing, warping, and other effects.

IN THIS CHAPTER

A number of these nodes perform a convolution or other neighborhood operation on each pixel in the image. For the most detailed description of the convolution process, refer to the **Convolve** node, which is a comprehensive convolution filter featuring a configurable kernel as well as a library of kernel presets. The **Rank** node is also a multipurpose filter that can be configured for various tasks and offers a menu of preset operations.

Both Convolve and Rank examine pixels surrounding the pixel currently being operated on to obtain a value. However, Convolve averages the pixels in the sample area to obtain a value, whereas Rank sorts the pixels in the sample area and picks one to use for the value based on its ranking in the sort list.

The **Blur** node and the **Sharpen** node can be used by setting a single parameter, or advanced filtering options in each node can be accessed to precisely control a number of variables. **Time Blur** averages pixels from different frames rather than pixels in an area of the same frame. **Vector Blur**, which blurs directionally, is often used for motion blurs, while **BlurXY** lets you control horizontal and vertical blur levels separately.

The **Grain** node includes presets for various film stocks, and offers parameters for custom grain pattern matching, while **Degrain** offers comprehensive control over the process of removing grain.

Another special-purpose filter is **Unsharp Mask**, which sharpens an image using a specific photographic technique that involves boosting the contrast in areas with a lot of detail.

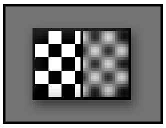
The **Bump Map** node creates a two-channel floating point output that can then be used as a mask input to other nodes, such as **Emboss** and **Vector Warp**, that create embossing, ripples and other distortions. The **Posterize** node, which reduces colors to an indexed palette, can be used to create broad regions of uniform color.

The output of the **Edge** node, which detects the edges of an image, is often used as a mask input to other nodes to confine processing to edge or non-edge areas. The **Text** node, which is used to add a layer of text over an image, can be used for titling, adding time code, and so forth.

The **Comment** node, unlike the other nodes in this group, does not process image data; rather, it is used to store remarks.

The **Xpresso** node is perhaps the most versatile node in RAYZ. To use Xpresso, you type expressions into a field, which enables you to essentially design your own filter, composite, or other operator. Xpresso is a multi-input node—every time you add an input to Xpresso a new connector is created.

BLUR NODE



The Blur node enables you to blur an image, or selected channels of an image.

The Blur node is a convolution filter that performs the equivalent of low-pass filtering in the frequency domain, where the high frequencies from a signal are filtered out and only the low frequencies are allowed to pass through. Because areas of greatest visual detail in an image are expressed as high frequencies, the low-pass filter effectively blurs the image.

The Blur node accepts one or two inputs, with the optional second input used as a mask to control the area of the primary input that is blurred by the node. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

BLUR PARAMETERS

All you have to do to use the Blur node is specify a value in the Kernel Size parameter—higher values blur more and lower values blur less. However, you also have the option to customize the kernel used in the convolution by modifying the other kernel parameters. (For more information about convolution kernels, see also "How Convolve Works" on p. 352.)

KERNEL SIZE

The Kernel Size parameter is used to control the amount of blurring, in a range that represents approximately 5 percent of the total width of the input image, although the upper end of the range is unconstrained. The default Kernel Size value is equivalent to 1 percent of the total width.

TIP:

To make extremely fine incremental adjustments using the slider, hold down the Shift key as you drag the slider back and forth.

KERNEL SHAPE

The Kernel Shape menu specifies the distribution function to use to weight the kernel values; that is, the extent to which surrounding pixels will contribute to the blur. The default is Gaussian, a bell-shaped distribution curve. The other choices are Box (constant), Linear (triangle), Quadratic, and Cubic.

The optimal choice will depend on the nature of the image and the effect you wish to achieve. The Blur node operation is so fast with any of them that it is easy to try all the different options on an image until you find the best one.

Border Style

The pixel currently being modified is in the center of a matrix of pixels that contribute to the effect. This means that when processing a pixel that borders the image, or when the operation area is sufficiently large, there will not be adjacent pixels on all sides to contribute to the calculation.

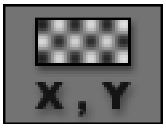
The Border Style menu specifies where the node will get the values it will use to fill the empty cells in the matrix as it processes each border pixel:

- Hold – Use edge value: This option assigns the value of the border pixels to the cells that fall outside the image border.
- Black – Blend with global space: This option assigns a value of 0 to cells that fall outside the image border.
- Mirror – Mirror edge values: This option assigns the values of the inner pixels in the matrix to the corresponding cells that fall outside the image border.

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, all channels of the input image are selected. To deselect a channel, press the button labeled with that channel letter (such as A for Alpha).

BLURXY NODE



The BlurXY node can be used instead of Blur when you need to specify the Blur level separately in X and Y.

The BlurXY node accepts one or two inputs, with the optional second input used as a mask to control the area of the primary input that is blurred by the node. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

BLURXY PARAMETERS

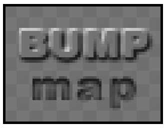
BLUR SIZE

Use this parameter to specify the amount of blurring in X (horizontally) and Y (vertically). The value is expressed in pixels, with the default value being equivalent to 0.01 percent of the total width for the X field, and of the total height for the Y field.

CHANNEL SELECT

Use this parameter to specify which channels of the input image will be processed by the node. By default, all channels of the input image are selected. To deselect a channel, press the button labeled with that channel letter (such as A for Alpha).

BUMP MAP NODE



The Bump Map node enables you to create a bump map that you can use as a mask input to other nodes to control embossing, rippling, or other effects that use pixel displacement, such as in the Vector Blur and Vector Warp nodes.

The Bump Map node uses a single, user-selectable channel of the input image. A surface normal, that is, a vector perpendicular to the surface, is calculated for each pixel of the selected channel by taking a weighted average of a sample of surrounding pixels. The value of the normal is based on the rate of change between the current pixel and neighboring pixels of the selected plane.

The data is output as a two-channel floating point image, with the value for the X component of the normal in the red channel, and the value for the Y component in the green.

BUMP MAP PARAMETERS

CHANNEL

Use the Channel menu to specify which channel of the input image should be used to calculate the surface normal for each pixel.

RADIUS

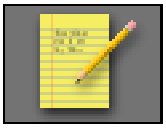
Pixels are sampled along the circumference of a circle surrounding the pixel currently being processed. The Radius parameter enables you to set the size of the radius that defines this circle in a range of 0 to 5.

You may wish to increase or decrease the radius of the sample area depending on the level of detail in the source image and the effect you wish to achieve. For example, a larger radius would result in a loss of local detail when used for an image with many fine lines.

SAMPLES

The Samples parameter enables you to specify the number of surrounding pixels to be sampled in a range of 3 to 10. The default value is 4. Larger sample values may take longer to process but produce a more accurate result.

Comment Node



The Comment node does not process image data. In fact, image data is always passed through the node without change. Instead, the Comment node is used to store notes and comments about any aspect of a shot.

You can use the Comment node to enter evaluation comments by a supervisor or to make notes about a particular effect for later reference. Such notes can be especially helpful when you have to come back days or weeks later and revise a shot, or reuse elements of it in a related shot or on a new project. Comments are also helpful when another compositor ends up working on the same file.

NOTE:

You may prefer to use Underlays in the Worksheet to annotate a network instead of or in addition to Comment nodes. For more information, refer to "Adding Underlays to the Worksheet" in chapter 5, p. 60.

The Comment node has a single input connector, which is usually connected to whichever node the comment is about. The Comment node may branch off on its own, or it may be inserted into a data stream. In any case, the way the Comment node is connected does not affect the shot.

Comment Parameters

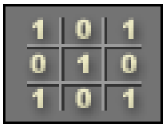
The Comment Node Panel offers a single parameter, a multi-line text field into which you enter your remarks. Click in the field and type your comment, pressing the Enter key when you are done.

To start a new line in the Comment field, hold down the Shift key as you press Enter.

TIP:

Use the Name field in the strip at the top of the Node Panel to give the comment a short, descriptive title that can be read on the Comment node in the Worksheet.

CONVOLVE NODE



The Convolve node is a convolution filter that can be used to sharpen or blur an image or detect its edges, depending on the type of filtering selected.

The Convolve node accepts one, two, or three inputs:

- The top input is the image to be convolved.
- The middle (optional) input is used as a convolution kernel.
- The bottom (optional) input is used as a mask image to control which pixels in the top input are convolved. See also "Using Mask Inputs" in chapter 7, p. 102.

HOW CONVOLVE WORKS

The Convolve node operates on each pixel in the input image in turn, modifying its value based on the values of surrounding pixels. The extent to which each adjacent pixel will affect the processing of the center pixel is determined by the convolution kernel.

A **convolution kernel** can be thought of as a square grid of cells placed over the group of pixels, with the pixel to be modified under the central cell of the grid. This grid, or matrix, of cells is referred to as the kernel.

Each cell in the kernel contains a value that determines how the corresponding pixel value under the cell will be used to modify the central pixel. The type of convolution filter you select will control the distribution of kernel cell values.

For example, to blur an image the values of the pixels under the grid would be averaged to generate the value to use for the central pixel. As the kernel operates on each pixel in the input image in turn, a blur effect is created in the output image.

If a simple box (constant) function is used to generate the cell values, each cell in the kernel will be equal, meaning that each pixel will contribute equally to the result. But if a nonlinear function such as Quadratic is used, the kernel will be weighted so that some cells will have greater values than others, meaning that some of the pixels will contribute more of their value to the result than others.

The Convolve node provides predefined convolution kernels for blurring, sharpening, and so forth that you select from the "Kernel Library" (p. 354). However you can also a create a custom kernel and use it instead of a library kernel.

You can create your own filter by entering values into the cells of a kernel. You can start from scratch or modify the values of one of the predefined kernels in the library to suit your needs. Another option is to connect a kernel input, as explained in "Using an Input as a Kernel" (p. 353).

Creating Custom Kernels

To create a custom kernel with the cell values you specify:

1. Select Custom from the Kernel Library menu.
2. Expand the Kernel Values parameter to access the cell grid.
3. Modify the default values entered in the Kernel Values grid.

By default, all 49 cells in the kernel have a value of 1. The larger the matrix, the greater the contribution of surrounding pixels to the pixel currently being convolved.

To create a smaller matrix (such as 3 x 3 instead of the default 7 x 7), enter a value of 0 into the non-applicable cells, or modify one of the preset kernels that is the size you want.

To modify an existing kernel in the library:

1. Select a specific kernel (such as 5 x 5 Gaussian) from the Kernel Library menu.
2. Then select Custom from the Kernel Library menu.
3. Expand the Kernel Values parameter to access the grid, which will display the cell values of the last selected preset (5 x 5 Gaussian, e.g.). All cells outside the 5 x 5 area will be set to 0.
4. Modify the preset values in the grid.

Using an Input as a Kernel

Any image connected to the middle, optional input to the Convolve node is treated as the kernel in the convolution operation. That is, the value of each pixel represents a cell value in the kernel matrix.

The kernel input must be a one-channel, floating point image. It can be square or rectangular, but it must be an odd number of pixels in both width and height. (The odd numbers are required because there must be a center cell in the matrix.)

Within these restrictions, there is no specific limit on the size of the kernel input. A relatively small kernel will still have a powerful effect, however, and the larger the kernel, the longer it will take to process the image.

TIP:

You can use a Color node to create a custom kernel input. Set the node to output a single-channel (select Alpha Only), floating point image.

Then, for example, you could set the size to 11 x 3 and the color to white to create a kernel that blurs the primary input to the Convolve node horizontally (vertical lines/edges would be affected most). A 3 x 11 image, on the other hand, would blur vertically, affecting horizontal edges most.

Convolve Parameters

At minimum, all you need to do is select a preset type of filtering from the Kernel Library menu and the other Convolve parameters are set accordingly. You can always adjust the settings if necessary to fine-tune the result.

You also have the option of creating your own kernel by selecting Custom from the Kernel Library menu and then typing cell values into the Kernel Values matrix (expand the Kernel Values parameter to access the grid).

Kernel Library

The Library menu provides kernel presets: predefined cell matrices with different functions to perform various types of filtering, including blurring, sharpening, and edge detection. Most of the kernels use floating point values; the exceptions are 3 x 3 Average, 3 x 3 Laplacian, and 7 x 7 Prewitt, which are integer.

TIP:

To examine, or modify, the individual cell values used for any preset kernel in the library, first select the kernel from the menu and then select the Custom kernel from the same menu. This activates the Kernel Values parameter matrix, which always contains the kernel values of the last selected preset in the Library.

Custom

Select Custom from the Library menu to activate the Kernel Values parameter and create your own convolution kernel.

3 x 3 Average – Mild Blur (Integer)

This filter averages cell values to create a mild blur.

3 x 3 Laplacian – Edges (Integer)

The Laplacian filter emphasizes edge detail in an image. (For more general sharpening, use High Pass instead. To generate an edge map, use the Canny or Prewitt filters.)

3 x 3 High Pass – Sharpen

The High Pass filter is used to sharpen the image by emphasizing the high frequency image data that is associated with fine detail.

5 x 5 Low Pass – Stronger Blur

The Low Pass filter is stronger than the Average blur. It will pass color frequencies in the lower ranges through to the output image; higher frequencies will be filtered out.

5 x 5 Gaussian – Blur

The Gaussian filter is an alternative to the Low Pass filter for a strong blur that uses a Gaussian distribution function to define the kernel.

5 x 5 Quadratic – Smooth

The Quadratic filter usually produces a subtle blur that removes noise to effectively smooth the image.

7 x 7 Prewitt – Vertical Edges (Integer)

The Prewitt filter is an alternative edge-detection tool to the Canny filter. The Prewitt filter finds edges that run vertically in the image.

7 x 7 Canny – Edges

The Canny filter is a general edge-detection tool that creates an edge map of the input image, which is most often used as a mask input in other node operations.

Border Style

The pixel currently being modified is in the center of a grid of pixels that define the operation area (the kernel). This means that when processing a pixel that borders the image, or when the operation area is sufficiently large, there will not be adjacent pixels on all sides to contribute to the calculation.

The Border Style parameter enables you to specify where the node will obtain the values it will use to fill the empty cells in the kernel as it processes each border pixel. Select one of the following options from the menu:

- Hold – Use edge value: This option assigns the value of the border pixel to the cells that fall outside the image border.
- Black – Blend with global space: This option assigns a value of 0 to cells that fall outside the image border.
- Mirror – Mirrors edges: This option assigns the values of the inner pixels in the grid to the corresponding cells in the grid that fall outside the image border.
- None – Do not process edges: When this option is selected, border pixels are not processed by the node.

Normalize Kernel

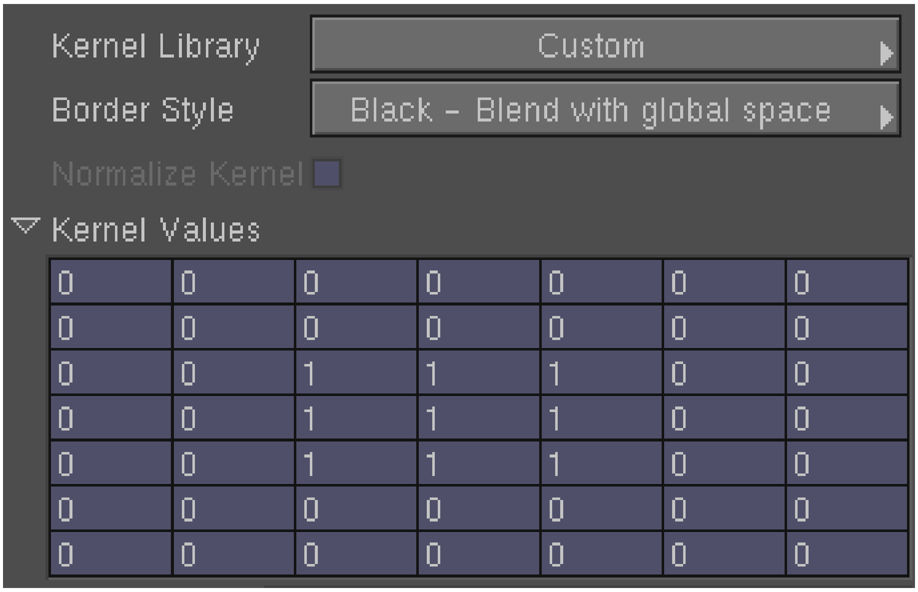
This parameter becomes active whenever a kernel that uses floating point (rather than integer) cell values is specified in the Kernel Library menu.

It is selected (on) by default whenever available, because it ensures that the sum of the kernel is 1 by dividing each cell in the kernel by the sum of all the cells in the kernel. Otherwise, the overall brightness of the image might be affected.

Kernel Values

The Kernel Values parameter becomes available whenever Custom is selected in the Kernel Library menu. Expand the parameter to access a grid of editable cells in which you can create your own convolution kernel.

The custom kernel matrix can be any size from 3 x 3 (9 cells) up to 7 x 7 (49 cells). It need not be square, but it must have an odd number of rows and columns in order to define the central cell of the kernel.

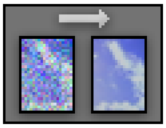


19.1 Expand the Kernel Values parameter to access the editable grid of kernel cells.

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, all channels of the input image are selected. To deselect a channel, press the button labeled with that channel letter (such as A for Alpha).

Degrain Node



The Degrain node is used to help remove film grain and other noise from imagery. You can elect to perform a rank operation (a type of blur) on the input, or the more complex frequency analysis method, optimized for film or video imagery, by selecting an option from the Filter menu.

When you select a method, other parameters are set to default values based on the selection. You can start by using the defaults and examining the results. If necessary, you can then modify parameter values using the guidelines described in the next section.

The frequency analysis parameters require more care to use properly, but can result in decreasing the appearance of grain while preserving image detail and sharpness that would be lost otherwise.

The Degrain node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

Choosing a Degrain Method

Choose the method to use in the Filter Type menu. The best choice will depend on the specific imagery being degrained. Median Rank is the easiest to use, but the Frequency Analysis methods, one optimized for film grain and the other for video noise, may preserve more detail.

The Median Rank operation is similar to area operators used for some types of blurs. It will reduce the appearance of grain, but it can also eliminate fine image detail. This means that Median Rank may make some imagery unacceptably soft.

How Frequency Analysis Works

Grain and noise are related image phenomena involving high frequency changes in the image content, which tend to be more or less random. Pure noise is completely random, like the snow you get with bad TV reception. Grain is bit more structured, being somewhere between pure noise and structured high-frequency content. (A good example of structured high-frequency content would be a long shot of a waving crowd in a stadium.)

The frequency analysis method examines the image data as a signal. It performs what is known as a coring operation to identify and decrease high frequencies, where both noise and image detail reside. This can also produce an overly soft image, so the Degrain node performs additional operations that compensate for this problem by essentially adding detail and sharpness back into the degrained image.

The Degrain methods and parameters are described in detail below.

DEGRAIN PARAMETERS

FILTER TYPE

The Filter Type menu enables you to specify the degrain method to use: Median Rank, Film Frequency Analysis (the default), or Video Frequency Analysis.

MEDIAN RANK FILTER

This method uses a median rank operation, which sorts the value of the pixel currently being processed with those of its neighbors and uses the median (middle) value in the ranking. For more detailed information about this operator, refer to the description of the "Rank Node" on p. 371.

FREQUENCY ANALYSIS FILTERS

Frequency analysis treats the image as a continuous signal of changing intensity (from one pixel to the next and then from one scan line to the next), as it would be represented in the frequency domain. This signal is analyzed to determine which parts of the signal are original, and which are noise or grain.

The part determined to be noise, based on the parameters set in the Advanced parameter group, is removed from the signal. (This implies that the success of frequency analysis for any particular image may depend on the Advanced parameter values.)

The "cleaned up" signal is then added back into the original in order to get some or all of the detail back into the image.

As the names imply, you should select Film Frequency Analysis to remove film grain and Video Frequency Analysis to remove video noise.

KERNEL SIZE

The Kernel Size menu is used to specify the size of the kernel used by the Median Rank operation. As the menu options indicate, the smaller 3 x 3 kernel is faster, while the 5 x 5 kernel may take a little longer to process but produce better results.

NOTE:

The frequency analysis operations also reference the Kernel Size parameter for a blur operation that is performed initially to help identify the high frequency component, which is the difference between the original image and the blurred image.

ADVANCED PARAMETERS

Expand the group to access the Advanced parameters, which become active whenever a frequency analysis filter is selected.

TIP:

If you are unfamiliar with frequency analysis, read the guidelines in the parameter descriptions below and then experiment with the settings as you view the result in the Image Viewer. Your eye will tell you when you have a satisfactory result.

NOISE LEVEL

The Noise Level parameter enables you to set a threshold that determines how much of the signal should be subject to filtering:

- Larger Noise Level values will remove more of the original signal—they will remove more noise, but also more picture detail.
- Smaller values will preserve detail, but allow more noise through.

The master Noise Level parameter can be expanded to access individual channel controls.

For video imagery, it is only necessary to set the master Noise Level value, because video signals do not exhibit noise differences between the red, green and blue components.

For film, however, you may need to set the threshold values for each channel individually because each film layer responds differently to light. In particular, the blue layer typically has much more noise than the green layer. The default channel values are a good starting point, as they reflect a ratio typical for many film stocks: Red, 1.5; Green, 1; Blue, 2.

GAIN AND EXPONENT

These parameters help control how much of the new, degrained signal is added back into the original image.

GAIN: Higher values will enhance detail, up to a point; after that, the image will begin to look harsh, or over-sharpened.

EXPONENT: The exponent parameter controls the slope of the nonlinear curve that represents the frequency distribution. In other words, the exponent controls how steep the climb is between low and high:

- Larger exponent values will tend to concentrate the degrain effect in areas of greatest frequency change in the image, typically edges.
- Smaller exponent values will tend to spread the effect throughout the image, making it appear soft.

Typically, more filtering should be done at higher frequencies, which tend to encompass most of both noise and image detail, and less filtering should be done as the frequencies become lower. This relationship is not linear, however, as more of the high frequency material is added than the low frequency information.

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, all channels of the input image are selected. To deselect a channel, press the button labeled with that channel letter (such as A for Alpha).

EDGE NODE

The Edge node is used to perform edge-detection on an image. The output of the Edge node is an edge map of the input, and it is often used as a mask image in blurs, dissolves, and other operations to improve the realism of composites by softening or otherwise adjusting foreground edges.

The Edge node works by comparing each pixel with surrounding pixels to find adjacent pixels with abrupt changes in value, which are identified as edges. The pixel currently being processed is the center pixel in a kernel (a matrix, or grid) that you specify using the Edge node parameters.

The Edge node accepts one or two inputs, the primary input and an optional mask input. For more information, see "Using Mask Inputs" in chapter 7 (p. 102).

EDGE PARAMETERS

METHOD

This parameter enables you to choose from the following edge-detection methods:

- 2 x 2 Cross
- Sobel (Two Direction)

The 2 x 2 Cross is the faster method, but the default is Sobel, which provides excellent results without being too computationally intensive. Sobel also enables you to optimize the size of the filter area for the specific image being processed.

FILTER SIZE

The Filter Size parameter becomes active when Sobel is the selected method. It enables you to specify the size of the kernel, or grid, of adjacent pixels that are examined and compared to the pixel currently being filtered. The value is expressed as a percentage of the total width of the input image, with the default being equivalent to 1 percent.

BORDER STYLE

The pixel currently being modified is in the center of a grid of pixels that define the operation area (the kernel). This means that when processing a pixel that borders the image, or when the operation area is sufficiently large, there will not be adjacent pixels on all sides to contribute to the calculation.

The Border Style parameter enables you to specify where the Edge node will obtain the values it will use to fill the empty cells in the kernel as it processes each border pixel. Select one of the following options from the menu:

- Hold – Use edge value: This option (the default) assigns the value of the border pixel to the cells that fall outside the image border.
- Black – Blend with global space: This option assigns a value of 0 to cells that fall outside the image border.
- Mirror – Mirrors edges: This option assigns the values of the inner pixels in the grid to the corresponding cells in the grid that fall outside the image border.
- None – Do not process edges: When this option is selected, border pixels are not processed by the node.

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. To select a channel, depress the button labeled with that channel letter (such as A for Alpha). By default, all channels of the input image are selected.

EMBOSS NODE

The Emboss node embosses an image by using a bump map, in effect raising in relief certain elements of the primary image. You can specify the magnitude of the effect as well as the direction of the simulated light source illuminating the embossed image, and other lighting characteristics.

The Emboss node requires two inputs, the image to be embossed, and the bump map. The bump map input must be a two-channel image such as that created by the "Bump Map Node" (p. 350). The bump map can be generated from the input imagery or from a different image, depending on the effect you want to achieve.

EMBOSS PARAMETERS

The Emboss parameters are extremely interactive, as they simulate dramatic changes in light and shadow across their range. If you need to match the lighting in other imagery, start by adjusting the Light Direction parameters before making any other adjustments. If you are trying to create an effect less directly related to real-world conditions, however, you could start with any parameter.

MAGNITUDE

The Magnitude parameter controls the general level of the embossing. The default value is 1, in a range of 0 to 2. As you increase or decrease this value, the Emboss node lengthens or shortens the vectors of the bump map accordingly.

LIGHT DIRECTION

The Light Direction parameter is used to specify the location in *xyz* coordinate space of the simulated light source.

AMBIENT

The Ambient parameter controls the level of non-directional, or surrounding, illumination in a range of 0 to 1. The default value is 0.3.

DIFFUSE

The Diffuse parameter controls the level of reflected light in a range of 0 to 3. The default value is 1.7.

SPECULAR

The Specular parameter controls the level of specular highlighting, the bright highlights from shiny surfaces. The parameter range is 0 to 1, with a default value of 0.8. The higher the value, the more white will be visible on the raised areas that are reflecting the most light.

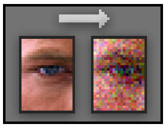
Shiny

The Shiny parameter represents the specular exponent and controls the size of specular highlights. The default value is 15, in a range of 0 to 30.

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, the RGB channels of the input image are selected. To select or deselect a channel, press the button labeled with that channel letter (such as A for Alpha).

GRAIN NODE



The Grain node is used to simulate the look of film grain. Adding grain to CG elements that will be composited over film footage is one of the most effective ways to increase the realism of a composite.

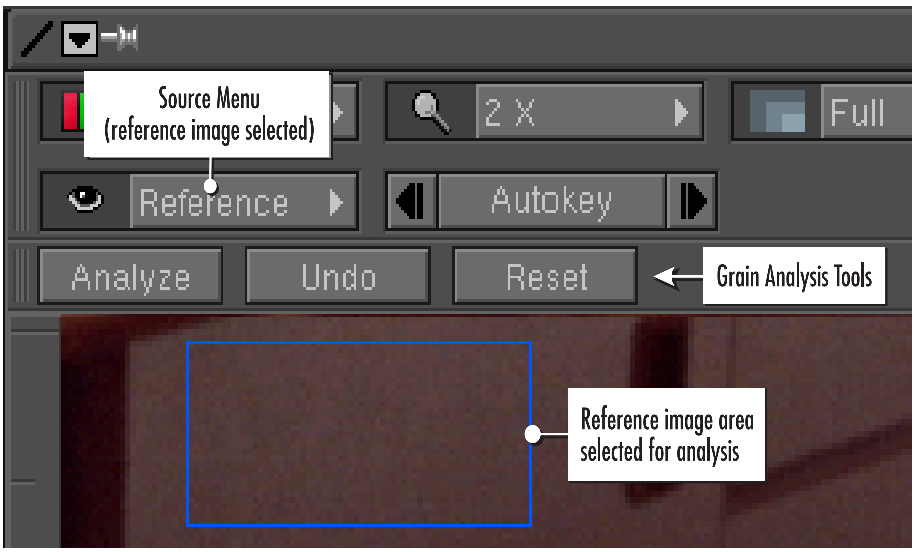
The Grain node accepts three inputs, the primary image to which the grain will be added, an optional reference input (its use is explained next), and an optional mask input, which is explained in "Using Mask Inputs" in chapter 7 (p. 102).

USING THE GRAIN NODE

There are three ways to use the Grain node. The Film Type menu offers presets to match a number of popular film stocks. The selected grain pattern can be modified, if necessary, by adjusting the Weight and Grain Size parameters.

In addition to using the Film Type presets, however, you can select Custom mode, which activates parameters in which you can enter values derived from statistical analysis of a specific film type not included in the menu.

You also have the option to connect a reference image to the Grain node and use the Grain tools in the Image Viewer to select areas for analysis. The node will set the custom parameter values based on the grain pattern in the reference image, as explained in "Analyzing the Grain in a Reference Image" (p. 366).



19.2 Grain analysis can be performed on a reference image.

In addition to the primary input, the Grain node accepts two optional inputs:

- The second input, labeled "R," is used for a reference image that has a grain pattern you want to match. (The next section explains how to analyze the reference image to set the Grain node parameters.)
- The third input, labeled "M," is used for a mask image to control which areas of the image are affected by the grain operation. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

Analyzing the Grain in a Reference Image

To generate a grain pattern that matches the grain in another image, follow these steps:

1. Connect the image with the grain you want to match to the reference input of the Grain node.
2. Display the Grain node in an Image Viewer, and use the Source menu to view the reference image. The hotkey is **s** for "source," which cycles the display through all input images and the output image in turn.
3. Examine the reference image to locate areas of homogenous color or smooth gradations. Be sure to avoid image areas with detail, which may be interpreted as grain.

   The backing area of a bluescreen image is an ideal choice, as is a gray card or similar calibration frame, assuming such footage was shot and digitized.
4. Drag across an appropriate area to draw a bounding box around it. You do not have to select a single contiguous area; you can create as many selection boxes as you need. This is recommended if the image has so much detail throughout that it is hard to select a single large area.

   If necessary, you can use the Undo button in the Grain tool strip in the Image Viewer to delete the last selection box. Use the Reset button to delete all of them.
5. Press the Analyze button in the Grain tool strip. The node will analyze the selected areas in the reference image to set the Custom parameters in the Film Type group.

   As soon as you press the Analyze button, the Film Type menu in the Grain Node Panel switches to Custom, if that option was not already selected.
6. Use the Source menu to switch the Image Viewer display to the output image. You can make further adjustments to the node parameter values manually, if necessary.

NOTE:

If the results seem extreme, the problem may be that the node has interpreted details in the image as grain.

In this case, delete the selection boxes by pressing the Reset button and try selecting a different area of the reference image to analyze.

Grain Parameters

Film Type

The Film Type menu enables you to select preset values optimized to match the grain characteristics of various film stocks:

- Eastman 5245/7245 EXR 50D ac
- Eastman 5247 ac
- Eastman 5248/7248 EXR 100T ac
- Eastman 5248/7248 EXR 100T nc
- Eastman 5287/7287 EXR 200T ac
- Eastman 5293/7293 EXR 200T ac
- Eastman 5293/7293 EXR 200T nc
- Eastman 5296/7296 EXR 500T nc
- Eastman 5298/7298 EXR 500T nc
- Kodak Vision 5246/7246 250D ac
- Kodak Vision 5246/7246 250D nc
- Kodak Vision 5274/7274 200T ac
- Kodak Vision 5274/7274 200T nc
- Kodak Vision 5277/7277 320T nc
- Kodak Vision 5279/7279 500T nc

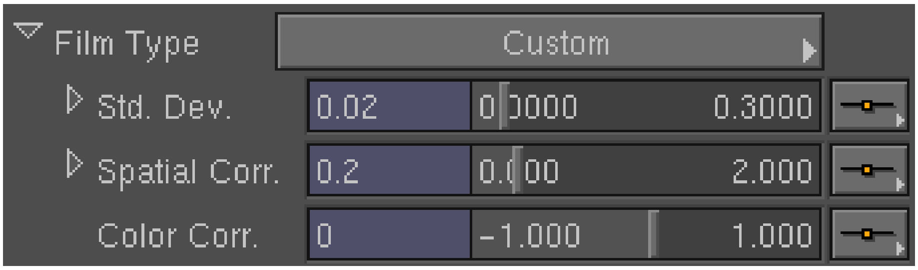
NOTE:

When two versions of a film stock, ac and nc, are available, choose "ac" to reproduce film scanned with aperture correction, and "nc" for film scanned without aperture correction.

The pattern generated by the selected preset can be adjusted using the Weight and Grain Size parameters, if necessary.

Custom Film Type Parameters

The Film Type menu also includes a Custom option that you can use to set the parameters manually by specifying values in the Custom Film Type parameters, which offer a level of precision suited to measurements derived from statistical analysis of a particular film stock. If you select Custom from the Film Type menu, additional parameters are activated, which can be accessed by expanding the Film Type group.



19.3 Expand the Film Type group to access these parameters, which become active when Custom is selected from the Film Type menu.

The Custom parameters—Standard Deviation, Spatial Correlation, and Color Correlation—can be used alone or in conjunction the Weight and Grain Size parameters that are used to adjust preset film stocks.

The Standard Deviation value is multiplied by the Weight value, and the Spatial Correlation value is multiplied by the Grain Size value to generate the custom grain pattern. (The default Weight and Grain Size values are 1, however, which obviously will not affect the output.)

NOTE:

The Custom parameters are set by the Grain node automatically whenever you use the grain analysis tools in the Image Viewer to match the grain pattern in a reference image.

Standard Deviation

This parameter specifies the strength of the grain in units representing the standard deviation value measured by statistical analysis of a particular film sample. Higher standard deviation values reflect more variation in the grain, which makes the grain more apparent.

The Standard Deviation parameter can be expanded to access individual controls for the Red, Green, and Blue channels.

Spatial Correlation

This parameter specifies the size of the grain in units representing the spatial correlation value measured by statistical analysis of a particular film sample. Higher spatial correlation values increase the average size of the grain.

The Spatial Correlation parameter can be expanded to access individual controls for the Red, Green, and Blue channels.

Color Correlation

This parameter specifies the apparent colorfulness of the grain in units representing the color correlation value measured by statistical analysis of a particular film sample.

The value represents how closely the grain in each channel overlaps. This means that negative color correlation values decrease the amount of overlap, which increases the apparent color of the grain, while positive values decrease its colorfulness.

Weight

The Weight parameter value is a scale factor that adjusts the apparent strength of the grain in a range of 0–2. The default value is 1, which has no effect on the output. Larger Weight values make the output look grainier, while smaller values make the grain less apparent.

You can expand the Weight parameter if you need to adjust the red, green, and blue channels individually.

Grain Size

The Grain Size parameter value is a scale factor that adjusts the average size of the grain in a range of 0–2. The default value of 1 has no effect on the output. Higher Grain Size values make the grain flecks appear larger on average, while lower values make the grain appear smaller.

Random Seed

The seed value is used by the node to generate a sequence of random numbers for the grain operation, which means that animating the Seed parameter will vary the grain pattern for each frame.

Since this is almost always preferable, the Random Seed parameter is animated by default using $F. This global variable represents the current frame number, which means that by definition it generates a different seed value for every frame.

Posterize Node

The Posterize node remaps the pixels of an input image to a reduced palette of colors to create large areas of solid color. You can specify the number of colors per channel to use, as well as which channels of the input to modify.

For example, if you reduce an image to 16 colors per channel, for each pixel in the image the Posterize node will examine the value of the red channel and reassign that value to the nearest red value in a predetermined set of 16 available reds. Then it will do the same for the green channel, substituting the nearest green value in a predetermined set of 16 greens, and so on. The resulting output image will have a maximum of 16 different values assigned to each channel.

The Posterize node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

Posterize Parameters

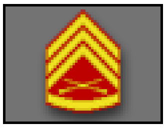
Output Colors

The Output Colors parameter specifies the number of colors per channel to use for the output. The default is 16 colors per channel for 8-bit inputs, 4096 for 16-bit, and 0.0625 for floating point.

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, all channels of the input image are selected. To deselect a channel, press the button labeled with that channel letter (such as A for Alpha).

RANK NODE



The Rank node is a versatile filter that is commonly used to suppress noise. The Rank node is a neighborhood operator; that is, the values of neighboring pixels influence the output value of the pixel being processed. Rank sorts the pixels in the sample area and picks one to use for the value based on its ranking in the sort list. The way the pixels are sorted depends on the type of rank operation selected.

The Rank node enables you to specify parameters that affect how each pixel is processed, including

- the type of operation to perform (Operation parameter),
- the size of the neighborhood, or operation area (Filter Size parameter),
- the shape of operation area (Filter Coverage parameter), and
- how pixels bordering the image will be treated (Border Style parameter).

NOTE:

To remove film grain, you may prefer to use the Degrain Node (p. 357) instead of Rank. Another option for reducing noise in still imagery is the Time Blur Node (p. 382).

The Rank node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

SELECTING A NOISE REDUCTION OPERATION

The choice of filter operation to use for noise reduction depends on the degree and character of the noise. As a general guideline, it is best to use the option that will create the fewest image artifacts and still produce an acceptable level of noise reduction.

To remove spot noise, "Threshold Replace (with avg)" is effective. For more information about this option, see the "Threshold" (p. 373) parameter description.

If the noise problem is more extensive, try using "Remove Bright Noise (maximin)" or "Remove Dark Noise (minimax)":

- The Remove Bright Noise operation starts by calculating the minimum value of a series of subsets of the operation area. Then the maximum value of these mins is used as the output. This is the "maximin" operation.
- The Remove Dark Noise operation does the opposite—it calculates the maximum value of each subset and uses the minimum of these as the output value. This is the "minimax" operation.

For more comprehensive noise reduction, before resorting to the Median operation, try using "Average of Maximin and Minimax." As the name implies, this operation calculates both a maximin and a minimax for each pixel and uses the average of the two as the output value.

NOTE:

For the best result with noise reduction filters, specify Cross for the filter shape. See the "Filter Coverage" (p. 372) parameter description for more information.

RANK EDGE OPERATORS

The Rank Operation parameter includes two options for modifying edges: "Min Pixel Value (Erode)" and "Max Pixel Value (Dilate)." If you need to shrink and grow edges with subpixel accuracy, however, use the "Erode Dilate Node" (ch. 15, p. 207) instead.

RANK PARAMETERS

OPERATION MENU

The Operation menu is used to specify the type of operation to be performed by selecting one of the following options:

- Min Pixel Value (Erode)
- Max Pixel Value (Dilate)
- Average Pixel Value (Blur)
- Median (Rank 0.5)
- Threshold Replace (with avg)
- Remove Bright Noise (maximin)
- Remove Dark Noise (minimax)
- Average of Maximin and Minimax
- Specify Pixel Rank

FILTER COVERAGE

The Filter Coverage parameter provides two choices for defining the shape of the operation area: Box and Cross. (The size of the area is specified in the Filter Size parameter.)

BOX: This option specifies a square grid of pixels. Box is usually the best choice for edge operations such as Min Pixel Value and Max Pixel Value.

CROSS: This option specifies a cross-shaped area consisting of a horizontal and a vertical line of pixels, with the two lines intersecting at the pixel currently being processed. It has a more subtle effect than using Box. Try the Cross option first when using the noise suppression operations as it may reduce unwanted blurring while still proving effective at removing noise.

Filter Size

The Filter Size value determines the size of the operation area—the neighborhood of surrounding pixels that will be used to calculate the output value of the central pixel. The parameter value is expressed as a percentage of total image width.

Threshold

This parameter becomes active when "Threshold Replace (with avg)" is chosen from the Operation menu. Threshold Replace is a good choice for removing spot noise without introducing excessive blurring to the result. If too much detail is lost, try adjusting the Threshold value to bring it back.

The Threshold Replace operation calculates the average of all the pixel values in the operation area and compares this average value to the value of the central pixel (the pixel currently being processed). If the difference between the two is greater than the threshold value specified in the Threshold parameter, the value of the central pixel is replaced by the average value.

Rank

This parameter becomes active when "Specify Pixel Rank" is chosen from the Operation menu. The Pixel Rank operation sorts, or ranks, each pixel in the operation area by value, from lowest to highest, and assigns the value of one of these pixels to the pixel currently being processed.

The pixel to use for the value is determined by its rank, in a range of 0–1, which you specify in the Rank parameter. For example, if you specify 0.5, the median pixel value in the ranking is assigned as the output pixel value.

Border Style

When using a neighborhood operator, the pixel currently being modified is in the center of a grid of pixels that defines the operation area. Therefore, when processing a pixel that borders the image (or when the operation area is sufficiently large), there will not be adjacent pixels on all sides to contribute values to the calculation.

The Border Style parameter enables you to specify where the node will obtain the values it will use to fill the empty cells in the grid as it processes each border pixel. Select one of the following options from the menu:

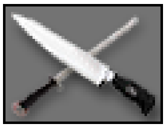
- Hold – Use edge value: This option assigns the value of the border pixel to the cells that fall outside the image border.
- Black – Blend with global space: This option assigns a value of 0 to cells that fall outside the image border.

- Mirror – Mirrors edges: This option assigns the values of the inner pixels in the grid to the corresponding cells in the grid that fall outside the image border.
- None – Do not process edges: When this option is selected, border pixels are not processed by the node.

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, all channels of the input image are selected. To deselect a channel, press the button labeled with that channel letter (such as A for Alpha).

SHARPEN NODE



The Sharpen node enables you to enhance the detail in an image, effectively sharpening it. You can choose from a number of different methods, including several convolution filters and frequency analysis.

The best method to use will depend on the nature of the image being sharpened. The default method, a high pass filter, will be satisfactory in many cases. If it is not, try adjusting the Mix Amount parameter value. If this doesn't give you the result you want, experiment with the other options in the Method menu.

The Sharpen node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see "Using Mask Inputs" in chapter 7, p. 102.

SHARPEN PARAMETERS

All of the methods available in the Sharpen node use the Mix Amount parameter value to control the level of sharpening. The effect of the Statistical Difference and Frequency methods can also be controlled using one or more parameters in the Advanced menu.

METHOD

Select the method you want to use to sharpen the image from the Method menu:

- High Pass One (3x3 only)
- High Pass Two
- High Pass Three (3x3 only)
- Statistical Difference
- Cine Sharpen
- Frequency Analysis
- Frequency Blue Prefilter
- Frequency Grain Prefilter

The **High Pass** methods are convolution filters that modify each pixel in turn based on the values of surrounding pixels. Each high pass filter uses a different kernel to weight the contribution of these pixels:

HIGH PASS ONE			HIGH PASS TWO			HIGH PASS THREE		
0	-1	0	-1	-1	-1	1	-2	1
-1	5	-1	-1	9	-1	-2	-5	-2
0	-1	0	-1	-1	-1	1	-2	1

High Pass Two tends to enhance edges more than High Pass One or Three. (For a detailed explanation of convolution kernels, refer to "How Convolve Works" on p. 352.)

The **Statistical Difference** method often does a good job of preserving detail. It calculates the standard deviation of a pixel area and examines the difference between the standard deviation and the actual pixel values to determine how to add or subtract values from the pixel being modified.

In addition to the Mix Amount, this method also uses the Gain parameter value to control the output. (Expand the Advanced parameter group to access Gain.)

CineSharpen is a Cineon sharpening filter that performs a specialized type of unsharp mask operation. (For an overview of the steps involved in an unsharp mask operation, see the description of the "Unsharp Mask Node" on p. 385.)

The **Frequency Analysis** methods sharpen the image by analyzing the high-frequency part of the signal, which tends to include important image detail, like edges or small features (as well as noise). The high frequency component is separated out, smoothed if possible to remove extra noise, and then added back into the original image. Because several filtering steps are performed, the Advanced parameters must be set appropriately for the image being sharpened.

Two of the frequency analysis methods also perform a grain-filtering step that is done before the actual sharpening:

- **Frequency Grain Prefilter** will prefilter the RGB channels before sharpening.
- **Frequency Blue Prefilter** will prefilter the Blue channel only. This option may be desirable when sharpening a filmed image, as the blue layer in film is subject to more noise than either red or green.

Using Frequency Analysis

Start by expanding the Advanced parameter group and setting the Noise Threshold parameter, which sets the threshold for the high frequency signal being analyzed—anything above the threshold is considered content (image detail) and anything below is considered noise and removed.

The next step is to specify how much of the "cleaned up" high frequency signal gets added back into the original image. Examine the sharpened image in the Image Viewer and adjust the Mix Amount and Exponent parameters until you see the result you want.

Higher values mean more sharpening for both of these parameters. The difference is that the Exponent is used to concentrate the sharpening effect in the higher frequencies (i.e., edges) or spread it out over the entire image.

NOTE:

For the most detailed explanation of frequency analysis, see the description of the "Degrain Node" on p. 357.

Kernel Size

For the high pass sharpening filters, the Kernel Size menu is used to specify the size of convolution kernel used in the sharpening operation; that is, to specify how many surrounding pixels are examined as each image pixel is calculated. The larger the convolution kernel, the more pronounced the effect.

For other sharpening methods, however, the convolution kernel used actually performs a blur. This is an initial step in the operation that helps identify image detail, which is the difference between the original image and the blurred image.

You can choose from a 3 x 3, 5 x 5, 7 x 7, or 9 x 9 kernel, unless you have selected High Pass One or High Pass Three in the Method menu. For these two sharpening operations, which by definition use a 3 x 3 kernel, the Kernel Size method is disabled.

Mix Amount

This parameter specifies the amount of sharpening. Lower values produce less sharpening and higher values, more (from no sharpening at 0 to a very harsh result at 1).

Advanced Parameters

The Advanced parameters, Gain, Exponent, and Noise Threshold, are activated when a frequency analysis option is selected from the Method menu. In addition, the Gain parameter is activated when the Statistical Difference method is selected.

Gain

Increasing the Gain parameter may enhance detail, up to a point; after that, the image will begin to look harsh, or over-sharpened.

Exponent

The Exponent parameter, in conjunction with the Mix Amount parameter, modifies how much of the high frequency signal falling above the Noise Threshold parameter levels will be added back into the original image.

The Exponent differs from the Mix Amount parameter in that the exponent is a power function that controls the slope of the nonlinear curve that represents the frequency distribution. This means basically that it controls the steepness of the slope:

- Higher exponent values concentrate the sharpening effect (as specified in the Mix Amount parameter) in areas of greatest frequency change in the image, typically edges.
- Lower exponent values will tend to spread the effect throughout the image, making it look soft.

Noise Threshold

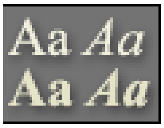
The Noise Threshold parameter sets a threshold for filtering the high frequency signal before it is added back into the original image. Larger values will remove more of the original signal—they will remove more noise, but also more picture detail. Smaller values will preserve detail, but allow more noise through.

The Noise Threshold parameter can be expanded to adjust the value separately for each channel. This is most applicable to filmed imagery, as each film layer responds differently to light. In particular, the blue layer typically has much more noise than the green layer.

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, all channels of the input image are selected. To deselect a channel, press the button labeled with that channel letter (A for Alpha, e.g.).

TEXT NODE



The Text node enables you to add text to an image to create titles and effects for final output or, more commonly, to add slates and frame numbers to temporary renders. You can specify font characteristics, color, and opacity, as well as rotate, scale, and move the text layer.

The Text node accepts one or two inputs. The second, optional input is used for a mask image. For more information, see “Using Mask Inputs” in chapter 7, p. 102.

The output of the Text node will be the same number of channels as the input image, with the text composited into all channels specified in the Channel Select parameters.

TEXT PARAMETERS

The parameters in the Text Node Panel are used to generate the text and specify the font, color, and opacity in which it will be displayed.

You can also adjust the size and position of the text by using the appropriate Node Panel parameters or by using the interactive overlay in the Image Viewer.

TEXT FIELD

Use the Text field to enter and edit the text. As you type, the field will scroll automatically if the text extends past the visible edge. When you press the Enter key, the text will appear over the image in the Viewer, at least wherever it fits within the image frame.

To break the text into multiple lines (to type a line break), press Shift-Enter while the cursor is in the text field.

USING GLOBAL VARIABLES IN THE TEXT FIELD

The default text in the field is “Frame:$F4,” where $F is a global variable that returns the current frame number. The numeral 4 appended to the variable specifies that the frame value be padded with zeros as necessary to generate a 4-digit number. So, assuming that you are at frame 1, the text in the image will read “Frame:0001.”

The text node can evaluate $F and any other global variable you define. No special syntax is necessary—just type in the variable. If you want the variable itself to appear, however, type a backslash (\) to prevent the global from being evaluated.

For example, typing `$F` will display the number “1” in the image (assuming that to be the current frame number); however, typing `\$F` will display “$F” in the image. (To display the backslash character, you would type it twice.)

Font Menu

The Font menu specifies the font in which the text will be displayed. You can choose regular, bold, or italic fonts in Baskerville or Nimbus Roman. You can also choose regular or bold sans serif and monospaced fonts: Nimbus Sans and Nimbus Mono. The menu also includes Brush Script, a calligraphic script font, and Liquid Crystal, which mimics LCD readouts.

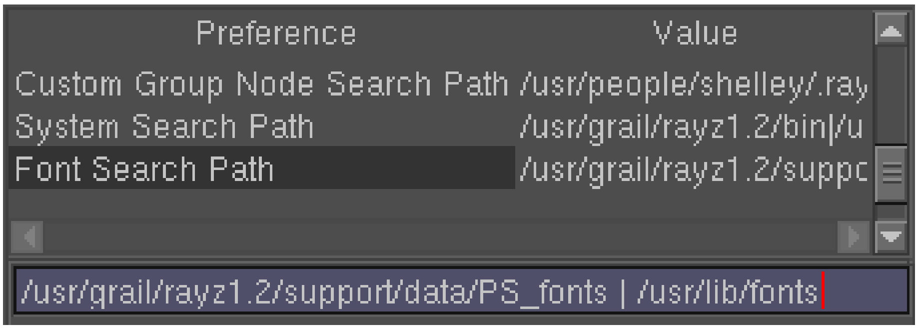
TIP:

Use a monospaced font such as Nimbus Mono to display frame numbers in the Text node. The equal width of all characters in monospaced fonts keeps the changing values from seeming to jump from frame to frame.

Adding Fonts to the Font Menu

You can use any fonts available to your system by adding their location to the Font Search Path in Edit > Preferences > File Paths. The Font menu in the Text node will update to include the additional fonts.

You could add fonts to the default font directory; however, this is not recommended as this directory could be overwritten when RAYZ is updated. Instead, use the pipe character (the vertical bar: |) to add a separate path in which RAYZ should also search for fonts.



19.4 You can use the pipe (|) character to add paths to the Font Search Path parameter. This example specifies that RAYZ search in the default directory and in "/usr/lib/fonts."

Character Size

This parameter specifies the size of the characters. The default value is a small percentage of the full size of the input image, expressed in units approximating points (used in typesetting).

Extra Kerning

Use the Extra Kerning parameter to increase the space between letters.

Extra Leading

Use the Extra Leading parameter to increase the space between lines of text.

Render Color

Render Color specifies the color of the text, which is white by default, using the color parameters that are common to many nodes in RAYZ. If you need more information, these controls are described in detail in the section on "Using the Color Parameters" (ch. 14, p. 168) in the Color node description.

OPACITY

The Opacity parameter specifies the opacity of the text layer over the image in a range of 0–1. By default the opacity value is set to1; that is, the text is totally opaque.

TRANSFORM PARAMETERS

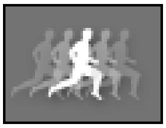
The **Rotate, Scale, Translate,** and **Pivot** parameters control the position and orientation of the text layer only, relative to the input image. They can be adjusted numerically in the Node Panel, or you can use the text transform overlay in the Image Viewer to adjust them interactively.

These parameters are identical to the spatial transformation parameters used in a number of nodes in RAYZ. For detailed information about their use, refer to the description of the "Transform Node" in chapter 17, p. 319.

CHANNEL SELECT

Use this parameter to specify which image channels will be affected by the node; that is, to which channels the text will be applied. By default, all channels of the input image are selected. To deselect a channel, press the button labeled with that channel letter (A for Alpha, e.g.).

TIME BLUR NODE



The Time Blur node enables you to modify each pixel in an image by averaging the values of corresponding pixels across a range of frames. This means that Time Blur only affects those pixels that change from frame to frame.

Time Blur can be used to create various trailing or strobe effects. Depending on the nature of the movement in a shot, this node can also create an effective motion blur because it blurs moving elements without blurring the background. This characteristic means that Time Blur is excellent for noise reduction on background plates without moving elements.

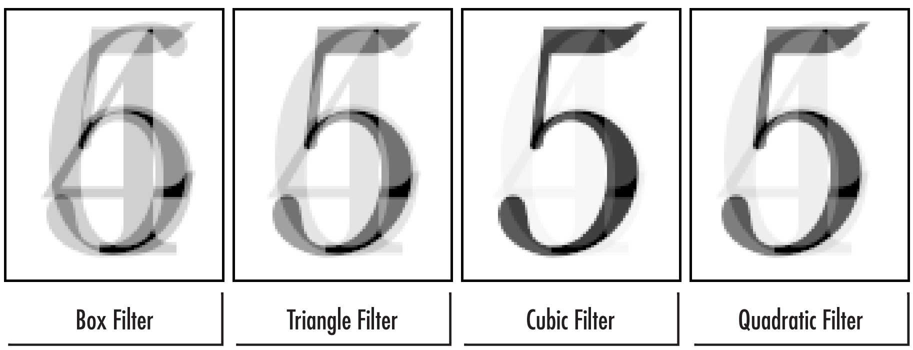
USING TIME BLUR

You can specify the following options for the Time Blur operation:

- the size of the filter, in frames (how many frames will contribute to the operation for each frame processed)
- the type of filter, which specifies how the values will be weighted
- the direction (from current frame) in which the filter will operate
- an offset from the current frame (so that the frame being modified is no longer the central frame of the filter)

You may want to experiment with different settings to get a feel for the effect they will have on your imagery. For example, try using the Triangle filter with the Past Only setting in the Direction parameter to create a diminishing trail behind a moving object.

TIME BLUR PARAMETERS



19.5 Examples of the effect of various filters on a sequence of images, each of which displays the current frame number. The Filter Size is 3 and the current frame is 5.

KERNEL SIZE

The Kernel Size parameter enables you to specify the size of the filter in frames. The default value is 3, which is the minimum size.

You can enter any odd integer value in the range, which is constrained to 3–31. (If you enter an even value, the node will actually use the next odd integer.) The odd value is stipulated because there must be a center frame to reference by other parameters that modify the filter.

NOTE:

The higher the Kernel Size value, the longer the node takes to process, since the data from that number of input frames must be accessed and calculated for each frame of the input as it is modified.

How Start and End Frames Are Processed

When the first frame of the input sequence is being processed, no previous input frame exists to use as a segment of the convolution filter. In such cases, the image data from the first frame of the input sequence is used as the value for all filter segments that would otherwise use the values from previous input frames.

The same holds true when the final frame of the input is being processed: the data from the last input frame is used for all filter segments that would otherwise use the values from subsequent input frames.

Kernel Shape

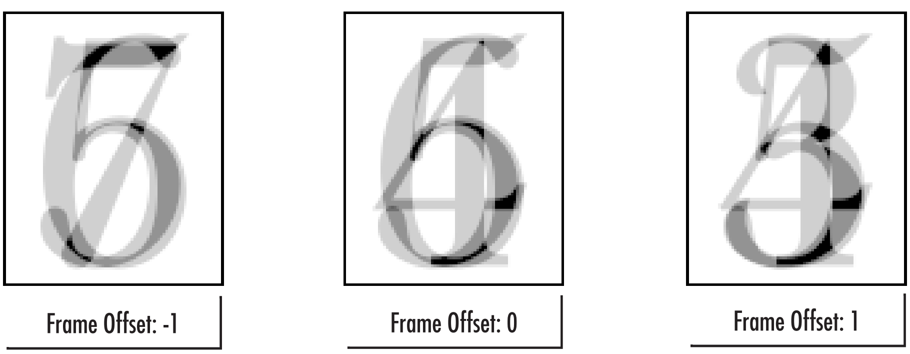
The frame averaging used in the Time Blur operation is weighted in proportion to the shape of the filter specified in the Kernel Shape menu. You can choose Box, Triangle, Cubic, Quadratic, or Gaussian (the default).

For example, the box filter (constant function) averages the values of each frame evenly, so that each contributes in equal measure to the result, while the triangle filter (linear function) performs a weighted average of the values in each frame, with the center frame contributing most.

The default Gaussian filter uses a bell shaped curve to weight the distribution. The Cubic or Quadratic filters often produce the most subtle effect.

Frame Offset

This parameter allows you to specify an offset from the center of the filter. If you set the Frame Offset parameter to a value other than the default value of 0, the current frame (the frame being modified) is no longer the center segment of the filter.



19.6 Examples of the effect of negative and positive Frame Offsets on a sequence of images of frame numbers. The Filter Size is 3 and the current frame is 5.

The range of valid Frame Offset values is -15 to 15, which corresponds to the maximum Kernel Size value of 31. It may be helpful to visualize the offset as sliding the frames of the sequence back and forth under a stationary window (the filter), as dictated by the offset value entered.

DIRECTION

The Direction parameter enables you to specify that only part of the filter be used in the convolve operation. The Direction menu enables you to specify Past and Future, Past Only, or Future Only. The terms *past* and *future* in this case refer to the direction in time from the current frame.

NOTE:

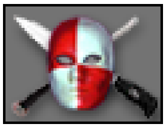
When Past Only or Future Only is selected, which eliminates one or more frames from contributing values to the filter operation, the results are normalized to prevent overall changes in brightness in static areas of the frames. This ensures that only pixels that change from frame to frame are modified.

19.7 Examples of the effect of Direction settings on a sequence of images of frame numbers. The Filter Size is 3 and the current frame is 5.

CHANNEL SELECT

Use this parameter to specify which channels of the input image will be processed by the node. By default, all channels of the input image are selected. To deselect a channel, press the button labeled with that channel letter (such as A for Alpha).

UNSHARP MASK NODE



The Unsharp Mask node simulates a technique used in traditional photography to sharpen an image. Unsharp Mask works by increasing the contrast between adjacent pixels in those areas with the greatest visual detail.

The process involves three steps:

1. A low-pass convolution filter is used to blur the source image.
2. The blurred image is subtracted from the source image. In effect, the blurred image is used as a mask to perform edge detection on the original image (hence the term "unsharp mask").
3. The result of the unsharp mask operation is added to the source image.

UNSHARP MASK PARAMETERS

The Unsharp Mask parameters enable you to control the blur level of the operation performed in the first step and how much of the unsharp mask is added to the source image in the third step:

- The Kernel Size, Shape, and Type parameters, along with the Border Style parameter, control the level of blur. (For general information about convolution kernels, see "How Convolve Works" on p. 352.)
- The Mask Amount parameter controls the contribution of the unsharp mask to the source image.

MASK AMOUNT

The Mask Amount parameter is used to specify the extent to which the source image is modified by the unsharp mask to generate the sharpened output image. The parameter range is 0–1. The default value of 1 adds 100 percent of the value of the mask to the source image, while a value of 0 would result in no change to the output image.

KERNEL SIZE

The Kernel Size parameter controls how strong the unsharp mask will be by specifying the size of the convolution kernel used for the blur. The parameter range represents approximately 5 percent of the total width of the input image, although the upper end of the range is unconstrained. The default Kernel Size value is equivalent to 1 percent of the total width.

KERNEL SHAPE

The Kernel Shape menu specifies the distribution function to use to weight the kernel values; that is, the extent to which surrounding pixels will contribute to the blur performed as part of the unsharp masking (see step 1 above). The default function is Gaussian, the familiar bell-shaped distribution curve. The other choices are Box (constant), Linear (triangle), Quadratic, and Cubic.

Border Style

The pixel currently being modified is in the center of a matrix of pixels that contribute to the effect. This means that when processing a pixel that borders the image, or when the operation area is sufficiently large, there will not be adjacent pixels on all sides to contribute to the calculation.

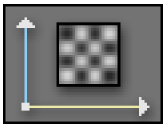
The Border Style menu specifies where the node will get the values it will use to fill the empty cells in the matrix as it processes each border pixel:

- Hold – Use edge value: This option assigns the value of the border pixel to the cells that fall outside the image border.
- Black – Blend with global space: This option assigns a value of 0 to cells that fall outside the image border.
- Mirror – Mirrors edges: This option assigns the values of the inner pixels in the grid to the corresponding cells in the grid that fall outside the image border.

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, all channels of the input image are selected. To deselect a channel, press the button labeled with that channel letter (A for Alpha, e.g.).

VECTOR BLUR NODE



The Vector Blur node is used to create directional blurs; that is, blurring along a specified vector. Vector Blur is most often used to create motion blurs.

The Vector Blur node blurs each pixel in an image based on parameter values that you set to control the number and location of neighboring pixels to be used in the blur computation, as well as the weight of the contribution of each neighboring pixel value to the blur effect.

The Vector Blur node computation is essentially a convolution whose kernel, or matrix of cells, consists of a single row of cells with the pixel to be convolved in the central cell of the row and neighboring pixels in the cells on either side. (For a general explanation of the convolution process, see the description of the "Convolve Node" on p. 352.)

Pixels around the border of the image, which do not have adjacent pixels on all sides, are mirrored; that is, they are treated as if there were adjacent pixels on all sides by mirroring the values of the inner adjacent pixels.

USING VECTOR BLUR

To control the direction of the blur effect, you specify the vector used to define the kernel. The vector can be thought of as a line drawn on the image, with the current pixel to be blurred in the center of the line.

You can control the following, using the indicated Node Panel parameter:

- the length of the vector, which affects the overall magnitude of the blur ("Magnitude" parameter),
- the angle of the vector, ("Angle" parameter),
- the type of filter to use to weight the contribution of each cell in the kernel ("Filter Type" parameter), and
- the portion of the vector used in the computation ("Extents" parameter).

USING A MASK INPUT WITH VECTOR BLUR

The Vector Blur node accepts one or two inputs. The second, optional input is used as a mask image for the blur. The mask input controls for Vector Blur are slightly different than they are for other nodes, however, because an extra option is offered.

In addition to selecting "On/Off" or "Mix," as described in "Using Mask Inputs" in chapter 7, p. 102, you can also select a third type of action: "Direction." The Direction option uses the mask image to determine the magnitude and direction of the effect.

The direction is determined by using two of the mask input channels to represent the *x* and *y* coordinates of a vector, in effect generating a surface normal (a vector perpendicular to the surface) for each pixel. In fact, you can use the output of the Bump Map Node (p. 350) as the mask input to Vector Blur.

When Direction is selected, therefore, the associated Channel menu for the mask input is deactivated. If the mask input contains more than two channels, the Vector Blur node will automatically select the red and green channel data to use.

VECTOR BLUR PARAMETERS

MAGNITUDE

The Magnitude parameter enables you to specify the length of the vector used to define the blur in a range of 1–50, with a default value of 10. You set this parameter to specify the general level of blur, with higher values resulting in a greater blur effect.

ANGLE

The Angle parameter enables you to set the direction of the vector used to define the blur. This parameter, along with the Extents parameter, is used to specify the apparent direction of motion in a motion blur.

You set the Angle parameter value in the range of 0–360 degrees. The default angle is 0, which equates to a horizontal left-to-right direction for the blur effect.

EXTENTS

The Extents parameter enables you to “clip” the function curve used to control the blur. In effect, this eliminates the contribution of any pixels in the kernel that fall outside the boundaries you set in the following range:

- A value of -1 represents one end of the established vector.
- A value of 0 represents the center pixel of the vector.
- A value of 1 represents the other end of the established vector.

The Extents parameter can be used to help create the effect of motion in a motion blur by enabling you to specify that only a portion of the pixels in the kernel be used in the computation.

For example, to create the effect of an object moving left to right you could set the Angle to 0 and the Extents parameters to 0 (left field) and 1 (right field). This would eliminate the contribution of all pixels in the kernel that fall to the left side of the central pixel.

EXTENTS VALUES IN RELATION TO ANGLE OF VECTOR

As the selected vector deviates from 0 degrees (the default), you can think of it as rotating counterclockwise to determine which edge of the vector is represented by the -1 value, and which by the 1 value. For example, for a

90-degree (vertical) vector, a value of 1 represents the top edge of the vector and -1 represents the bottom edge.

Filter Type

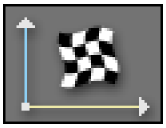
The Filter Type menu specifies the type of filtering to use to weight the distribution of the blur effect. For example, if a Box filter is used, each pixel contributes in equal measure to the blur effect. With the Triangle filter, the central pixels contribute most heavily to the result. The Gaussian filter, on the other hand, is a bell-shaped distribution function.

When some of the other filters, such as Mitchell or Sinc, are selected, additional parameters become active that are used to modify these filters. They are described in more detail in the section on "Filtering Transformations" in chapter 17 (p. 278).

Channel Select

Use this parameter to specify which channels of the input image will be processed by the node. By default, all channels of the input image are selected. To deselect a channel, press the button labeled with that channel letter (such as A for Alpha).

VECTOR WARP NODE



The Vector Warp node will warp an image using data from a second input image. Vector Warp can be used to create animated displacements, such as ripples or wakes, and other distortion effects.

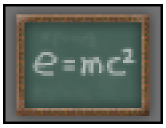
The Vector Warp node requires two inputs. The top input is the image to be vector warped, and the bottom input must be a two-channel floating point image, such as the output produced by the Bump Map Node (p. 350). The vector data in the bottom input is used to control the displacement of corresponding pixels in the top input image.

VECTOR WARP PARAMETERS

MAGNITUDE

The Magnitude parameter is used to adjust the magnitude of the effect by specifying a multiplier value for the vector data in the bottom input, in a range of 0.5 to 5. Larger values create greater distortion.

Xpresso Node



The Xpresso node is used to program custom effects by writing mathematical expressions that define each pixel in each channel of the output image.

All pixels and channels of the input imagery can be referenced in the expressions you devise, and the Xpresso node can have as many inputs as you choose. Each time you connect an input to Xpresso, another input connector is automatically created.

Xpresso must have at least one input, which is used to set the frame size and number of channels of the output, even if the input is not referenced by the expression.

When inputs of different sizes are connected to Xpresso, their active areas are merged to create the active area used by Xpresso to perform the node operations. The size of the output frame, however, is determined by the first (top) input to the node. The number of channels in the output is also determined by the top input.

When inputs are different bit depths, the lower are promoted to the highest bit depth among them. When all inputs consist of integer data (that is, 8-bit or 16-bit imagery), expressions are calculated in integer. Floating point inputs are always calculated in float. If the inputs are mixed, all values are converted to float. Integer inputs are also converted to floating point when a function that requires float values (sin, e.g.) is called.

TIP:

Use the "Save As" command in the Presets menu in the Xpresso Node Panel to save an expression under a unique name, which will then appear in the Presets menu of every Xpresso node. You can create a library of presets that you can select in any Xpresso node, without remembering and retyping the expression each time.

How the Xpresso Node Works

The Xpresso node begins processing at the first pixel of the first channel and continues until all pixels in the channel have been addressed and assigned a value. Then the Xpresso node addresses the subsequent channels in turn until each pixel in each channel has been assigned a value.

You define this process of assigning values to pixels by typing a mathematical expression into the Expression field in the Xpresso Node Panel.

Xpresso expressions use the following variables, functions, and operators. The section on "Guidelines for Using Xpresso" (p. 393) covers syntax and provides some examples.

SYSTEM VARIABLES

The following system variables are available to the Xpresso node:

X	the maximum x value, or the width of the picture
x	the current x value
Y	the maximum y value, or the height of the picture
y	the current y value
Z	maximum value (at 8 bits per pixel, this is 255; at 16 bits per pixel, this is 65,535; if you are using floating point data for pixel representation, this is 1.0).
SX	scale in current X specified by Image Viewer Size menu (scales expression when using integers to work with medium and low proxies; SX is a fractional, or float, value)
SY	scale in current Y specified by Image Viewer Size menu (scales expression when using integers to work with medium and low proxies; SY is a fractional, or float, value)
radius	distance from center of image (X/2, Y/2) to current pixel (x, y), measured in pixels; radius value is fractional (123.4, e.g.)
RADIUS	maximum radius, that is, distance from center of image (X/2, Y/2) to image edges (X, Y)
angle	angle, in degrees (from 0.0 to 360.0, counterclockwise), of vector from center of image to current pixel

You can also use the $F global variable, which represents the current frame number.

FUNCTIONS

The following functions are available in the Xpresso node:

abs	clamp	cos	rand
acos	contains	max	sign
and	cos	min	sin
angle	deg	odd	sqrt
asin	even	pow	tan
atan	hypot	pulse	wrap

atan2	if	rad	rand
between	log	radius	

OPERATORS

The following is a list of operators that can be used in the Xpresso node. The order of precedence (the order in which they are evaluated in the expression) is similar to the C programming language.

CONDITIONAL	condition ? true : false `if`(condition, true, false)
COMPARISON	`== != > < >= <=`
ARITHMETIC	`+ - * / % **` (e.g., base expression**exponent expression)

NOTE:

Due to inherent round-off errors, you should avoid comparing floating point numbers for equality: do not use the "==" operator with floats.

GUIDELINES FOR USING XPRESSO

An Xpresso expression tells RAYZ what values to use for each pixel of each channel in the output image. You can specify any values you choose for the output, and any pixel or channel in any input image can be referenced and modified to generate the output.

The output image is defined as buffer 0, the first (top) input to Xpresso is 1, the second, 2, and so on. To identify the number as referring to one of these image buffers, it is enclosed in brackets: [0] for the output or [1] for the top input.

Assume that there is one input to the Xpresso node. To set the RGB channels so that the output image is equal to the input image, you could enter the expression `[0] = [1]`.

Similarly, to set the output to black, you could type `[0] = 0`. And to set the output to white, you could type `[0] = Z`, since Z is the variable used to represent the maximum value of the colorspace.

Since setting the output buffer is the whole point of using Xpresso, however, it can be treated as implied behavior, which simply means that the expression `[0] = [1]` can be shortened to `[1]` and the expression `[0] = 0` can be shortened to `0`.

You can use more than one expression to define the output buffer. Expressions must be separated by a semicolon (;).

ADDRESSING PIXEL COORDINATES

To address pixel coordinates, use the form [n,x,y]. For example, [1,x,y] represents each pixel in the first input image. To flip the input image, you could enter this expression:

```
[0, x, y] = [1, y, x]
```

This expression could be also shortened to `[0] = [1, y, x]` or even to just `[1, y, x]`.

To shift the output image, say, 100 pixels to the right and 200 pixels down, use this expression:

```
[0, x + 100, y - 200] = [1, x, y]
```

As in previous examples, this could be shortened:

```
[0, x + 100, y - 200] = [1]
```

ADDRESSING SUB-PIXELS

To address fractional pixel values, which will be interpolated bilinearly, use braces {curly brackets} instead of square brackets. For example:

```
{0, x + 1.333, y - 2.333} = [1]
```

Another example might be:

```
[0] = {1, sqrt(x), sqrt(y)}
```

ADDRESSING CHANNELS

To address a particular channel of an input or the output, specify the channel number: [n,x,y,c]. The red channel is 0; green is 1; blue is 2, alpha is 3, and so on. To use the red channel of the first input to fill all channels of the output, for example, you would enter:

```
[1, x, y, 0]
```

To use all channels of the input in the output, but modify only one of the channels, start by specifying that the output should equal the input and then use another expression to modify a particular channel.

For example, the expression `[1] * 0.8` would darken the input image by 20 percent. To darken just the red channel, however, enter this instead:

```
[1]; [0,x,y,0] = [1,x,y,0] * 0.8
```

CONDITIONALS

To specify that only those pixels in a certain value range or location are modified, you can use conditional statements.

To darken the image only in a vertical strip running down the left side of the image, for example, use this expression:

```
[1] = x <= 200 ? [1] * 0.8 : [1]
```

This instructs Xpresso to check the x value of input pixel: if it is less than or equal to 200, multiply the color value of that pixel by 0.8; if it is not, use the color value unchanged.

You could get the same result by using the alternate syntax for conditional statements:

```
[1] = if(x <= 200, [1] * 0.8, [1])
```

To darken a vertical strip down the middle of the image, say from pixels 300 to 400 in X, nest another conditional statement in parentheses:

```
[1] = x >= 300 ? (x <= 400 ? [1] * 0.8 : [1]) : [1]
```

This instructs Xpresso to check the current pixel's x value to see if it is greater than or equal to 300. (If that is true, check to see if it is less than or equal to 400; if so, multiply the input color value by 0.8; if not, use the input color value unchanged.) If the x value of the current pixel is not greater than or equal to 300, use the input color value unchanged.

Here is an example that clamps the color value in the blue channel of an 8-bit input image [1,x,y,2] to a maximum of 128:

```
[1]; [0,x,y,2] = ([1,x,y,2] > 128 ? 128 : [1,x,y,2])
```

As in some of the earlier examples, the first expression `[1];` copies the input image to the output buffer, and it is separated from the next expression by a semicolon.

Note also that the color value (128) is expressed in the colorspace units of the input image rather than in generic floating point notation (0.5).

XPRESSO PARAMETERS

EXPRESSION FIELD

This is the field in which you enter expressions. The default expression is "Z," which generates a white image.

To type a line break, hold down the Shift key as you press Enter. For long, complex expressions, you may prefer to maximize the Node Panel to increase the size of the text box.

chapter 20

CONVERSION NODES

The nodes in the Convert menu are used to perform various conversion operations on the input image.

IN THIS CHAPTER

The **Bit Depth** node is strictly for converting linear data to a larger or smaller linear colorspace, such as 8-bit linear data to 16-bit linear data and vice versa. Bit Depth is easy to use and allows you to change the default remapping values based on the actual high and low values in your imagery.

Lin To Log and **Log To Lin**, on the other hand, are strictly for converting nonlinear (log) data to linear colorspace and vice versa. Automatic conversion settings can be used, or you can customize the conversion for specific imagery.

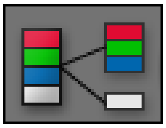
Channel Split is used to split an RGBA image into separate streams of RGB and Alpha channel data.

The **Premultiply** node multiplies the RGB channels of the input by the alpha channel, while **Unpremultiply** restores the original RGB channel values of a premultiplied RGBA image.

NOTE: More video-related nodes are described in *Chapter 21: Timing Nodes* (p. 413).

Interlace and **Deinterlace** are designed to work with video fields. The Deinterlace node takes a single input and outputs frames of either even or odd fields, as you specify. The Interlace node requires even and odd field inputs, which it reintegrates into interlaced frames.

Channel Split



The Channel Split node is used to split an RGBA image into two separate streams of image data: the RGB image and the Alpha channel.

Some users prefer to send the matte channel through the network separately rather than integrating the Alpha into the RGB image. This is not strictly necessary, insofar as you can use the Channel Select parameters available in most nodes to control whether the RGB or the Alpha of an RGBA input is affected by the node operation. But it can make the node network more "legible" in that you can tell at a glance whether the color or matte data is being modified by a node.

20.1 The Channel Split node in the Worksheet, with the RGB output connected to a Color Correct node and the Alpha output connected to Roto.

The Channel Split node accepts one input, which must be a four-channel (RGBA) image. Unlike other nodes, Channel Split has two output connectors:

- the top connector outputs the RGB image and
- the bottom connector outputs the Alpha.

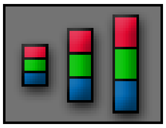
TIP:

To merge separate RGB and Alpha channels back into a single RGBA image, use the "Channel Swap Node" (ch. 16, p. 248).

Channel Split Parameters

There are no node-specific parameters available in the Channel Split Node Panel. The node automatically splits any RGBA image you connect to it.

BIT DEPTH NODE



The Bit Depth node converts images to and from 8-bit, 16-bit, and floating point linear data.

To use the Bit Depth node, select the output format from the Bit Depth menu in the Node Panel and RAYZ will do the rest. However, you also have the option of modifying the white and black mapping values to optimize the conversion for your imagery.

OPTIMIZING LINEAR CONVERSION

By default, the Bit Depth node remaps the maximum values of one bit depth to another. For example, assume that you wish to convert 16-bit data (0 to 65535 bits) to 8-bit data (0 to 255 bits). The 16-bit colorspace is scaled linearly by remapping 65535 to 255.

However, imagine that the data in your 16-bit image actually occupies a range of only 50 to 5050. You can scale the actual range of your input data (50 to 5050) down to the full range of the 8-bit colorspace (0 to 255) by entering a value of 5050 into the White In parameter and a value of 50 into the Black In parameter.

This enables you to spread the actual range of your input data over the maximum possible range of the output you specified.

BIT DEPTH PARAMETERS

BIT DEPTH

Select the bit depth to which the input image should be converted: 8-bit linear, 16-bit linear, or floating point linear.

WHITE IN AND WHITE OUT PARAMETERS

Optionally, you can adjust either the White In or White Out parameter value, or both, to optimize the conversion. These parameters adjust all channels equally; to control individual channel values, expand the parameter group to reveal the channel controls.

The “White” parameters represent the maximum values to be used for the conversion, with White In representing the input data and White Out representing the output data.

USING THE WHITE IN AND OUT EYEDROPPERS

The White In and White Out parameters provide an eyedropper tool you can use to sample an area of the image by clicking on an individual pixel or by scrubbing across an image area. (Remember to set the Image Viewer’s Source menu to Input Image to sample the input image values.)

The *maximum value per channel* of the sampled pixels will be used to set each individual color channel parameter. This means that you have expand the group to see the resulting values in the Red, Green, and Blue parameters. The master parameter value is not modified by the eyedropper selection.

Black In and Black Out Parameters

Optionally, you can adjust either the Black In or Black Out parameter value, or both, to optimize the conversion. These parameters adjust all channels equally; to control individual channel values, expand the parameter group to reveal the channel controls.

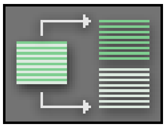
The "Black" parameters represent the minimum values to be used for the conversion, with Black In representing the input data and Black Out representing the output data.

Using the Black In and Out Eyedroppers

The Black In and Black Out parameters provide an eyedropper tool you can use to sample an area of the image by clicking on an individual pixel or by scrubbing across an image area. (Remember to set the Image Viewer's Source menu to Input Image to sample the input image values.)

The *minimum value per channel* of the sampled pixels will be used to set each individual color channel parameter. This means that you have expand the group to see the resulting values in the Red, Green, and Blue parameters. The master parameter value is not modified by the eyedropper selection.

Deinterlace Node



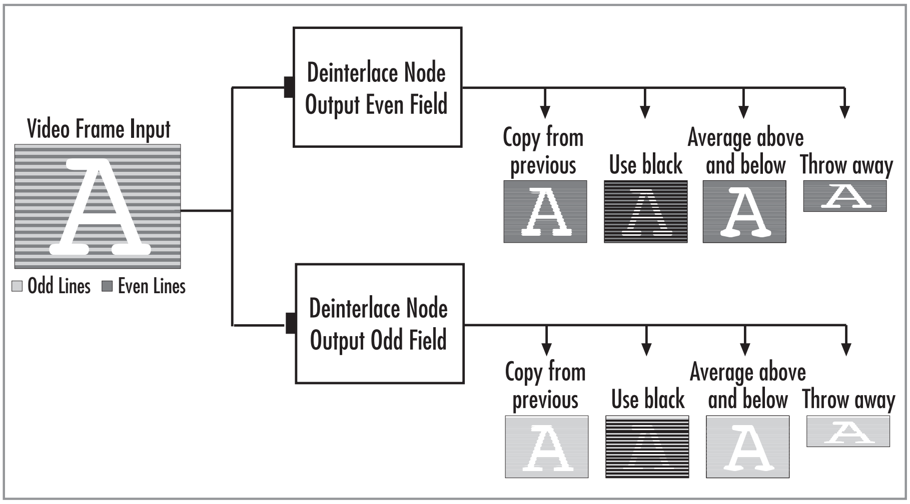
The Deinterlace node enables you to de-interlace video frames into frames of either odd or even fields, as you specify.

You can use two Deinterlace nodes to create separate sequences of the odd and even fields, which can be reintegrated in the complementary “Interlace Node” (p. 404).

NOTE:

To create a sequence of alternating odd and even fields for each frame, use the “Split Node” (ch. 21, p. 421) instead.

20.2 You can specify whether to output the odd or even field, and whether to fill in the missing field lines or to throw them away.



Deinterlace Parameters

Field

Use the Field menu to select the odd fields (lines 1, 3, 5, etc.) or the even fields (lines 2, 4, 6, etc.). If you choose “Odd,” for example, the Deinterlace node will retain the odd field data.

Other Field Fill

This parameter enables you to specify how to treat the vacant spaces that remain once the Deinterlace node removes either the odd or even scan lines. Select one of the following options from the menu:

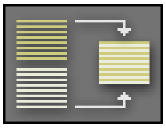
Use Black This option fills the vacant lines with black.

Copy from Previous This option fills the vacant lines with data replicated from each previous line.

Average Above and Below This option fills the vacant lines with data obtained by averaging the preceding and subsequent lines together.

Throw Away This option discards the data that formerly occupied the now-vacant lines. It has the effect of shrinking your file by 50% in the Y dimension. (In other words, an image that was originally 720 x 486 will become 720 x 243.)

INTERLACE NODE



The Interlace node recombines video imagery that has been split into separate sequences of odd and even field frames, which means that the Interlace node requires two inputs.

You can use the Interlace node to recombine frame fields that you de-interlaced using the "Deinterlace Node" (p. 402).

INTERLACE PARAMETERS

NOTE:

The Field Order and Input Style parameter settings should be based on the method used previously to de-interlace the frames using the Deinterlace node.

FIELD ORDER

The Field Order menu is used to specify which input should be used to create the odd fields (lines 1,3,5, etc.), and which input should be used for the even fields (lines 2, 4, 6 etc.):

- Select **Even, Odd** when the top input image should be used to create the even fields and the bottom input, the odd fields.
- Select **Odd, Even** when the top input image should be used to create the odd fields and the bottom input, the even fields.

INPUT STYLE

The Input Style menu is used to specify how the fields will be interlaced. The options are complementary to those available when using the "Other Field Fill" parameter in the Deinterlace node:

- **Copied** re-interlaces video fields that were de-interlaced using any option other than "Throw Away": "Copy from Previous," "Use Black," or "Average Above and Below."
- **Thrown Away** re-interlaces video fields that were de-interlaced using the "Throw Away" option.

Lin To Log Node

The Lin To Log node enables you to convert linear imagery to log (10-bit Cineon) format. The node will perform the conversion for you automatically; however, you can adjust the individual parameters if necessary to optimize the conversion for your imagery.

The complement to this node is the “Log To Lin Node” (p. 407), which converts Cineon 10-bit log imagery to a linear format. If you need to convert linear imagery to another linear bit depth, use the “Bit Depth Node” (p. 400).

NOTE:

The log image output by the node is displayed in the Image Viewer using the Raw Log display conversion method. You can choose Cineview emulation instead

Lin To Log Parameters

See also the description of the “Log To Lin Node” on p. 407 for information about the Kodak specifications on which default Cineon conversion values are based.

Conversion Menu

By default this menu is set to “Automatic,” which converts the imagery to log using default parameter values. Select “Manual” to activate the other conversion parameters and set the values yourself.

Conversion Parameters

These parameters become active when you choose a Manual from the Conversion menu. Each is a master control that affects all channels equally. To adjust any channel individually, expand the parameter group to access the RGB controls.

Reference Black

The Reference Black parameter enables you to modify the default log value (95).

Log 90% White

The Log 90% White parameter enables you to modify the default log value of 685 for reference white.

Lin 90% White

The Linear 90% White parameter enables you to modify the default linear value (65535 in 16-bit) for peak white.

Lin 90% White Eyedropper You can use the eyedropper tool associated with this parameter to sample an appropriate area of the image. Set the Image Viewer Source menu to Input Image and click on a pixel or scrub across an area. The maximum value sampled per channel will be used in the Red, Green, and Blue channel parameters (expand the group to access). The master Linear White value is not affected by the eyedropper selection.

Display Gamma

The Display Gamma parameter enables you to modify the default conversion value (1.7) for display gamma.

Film Gamma

The Film Gamma parameter enables you to modify the default conversion value (0.6) for film gamma.

LOG TO LIN NODE

The Log To Lin node converts Cineon 10-bit log data to linear format at the bit depth per channel you specify.

To convert linear image data into 10-bit log, use the "Lin To Log Node" (p. 405). To change one linear bit depth to another, use the "Bit Depth Node" (p. 400).

NOTE:

Cineon log files are usually converted to a linear format when they are loaded into an Image In node. However, this conversion can be disabled when you wish to work in log space. If at some point later in the network you want or need to work in linear space, you can use the Log To Lin node to convert the data.

LOG TO LIN PARAMETERS

The default values used in the Log To Lin Node Panel parameters are based on specifications published by Kodak Motion Picture & Television Imaging and Cinesite Digital Film Center in "Grayscale Transformations" (1993) and "Conversion of 10-bit Log Film Data to 8-bit Linear or Video Data" (1995).

TIP:

These documents can be downloaded in Acrobat PDF format, suitable for printing, from Silicon Grail at ftp://ftp.sgrail.com/pub/reference/cineon. They are also posted on the Technical Documents page of the Cinesite Hollywood website at http://www.cinesite.com/la/scanrec/techdocs.html.

CONVERSION MENU

Select the linear bit depth to which the log input image should be converted: 8-bit linear, 16-bit linear, or floating point linear. The node will then perform the conversion for you automatically; however, you can adjust the individual parameters if necessary to optimize the conversion for your imagery.

CONVERSION PARAMETERS

You can use the default values for these parameters, or you can modify them to suit your needs and imagery. Each is a master control that affects all channels equally. To adjust any channel individually, expand the parameter group to access the channel-level controls.

Reference Black

This parameter specifies the value used for reference black in the conversion operation. The default value is 95, which represents Dmin (the minimum printing density, or blackest black that can be recorded, about equivalent to the 1% black card).

Log 90% White

This parameter specifies the value used for reference white in the conversion operation. The default is 685, which represents the code value of the 90% white card for a normally exposed film negative.

Log 90% White Eyedropper You can use the eyedropper tool associated with this parameter to sample an appropriate area of the image. Set the Image Viewer Source menu to Input Image and click on a pixel or scrub across an area. The maximum value sampled per channel will be used in the Red, Green, and Blue channel parameters (expand the group to access). The master Log White value is not affected by the eyedropper selection.

Lin 90% White

This parameter specifies the value in linear space to which the log 90% white value will be mapped in the conversion operation. The default is 65535 (in 16-bit), which will clip any values above 685 in the original log file. When converting to linear float, however, values are not clipped at 1.

Display Gamma

The Display Gamma parameter enables you to modify the default conversion value (1.7) for display gamma.

Film Gamma

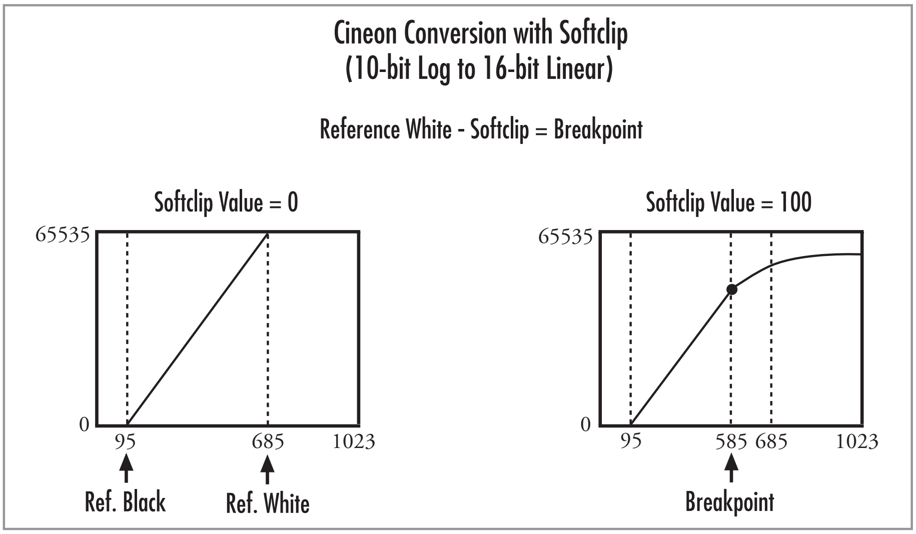
The Film Gamma parameter enables you to modify the default conversion value (0.6) for film gamma.

Softclip

Softclip can be used to reduce the effects of harsh clamping at the high end when remapping log data to 8- or 16-bit linear (highs are not clamped when converting to linear float). Check the Softclip box to enable the Softclip parameter and set a positive value in the range of 0–100.

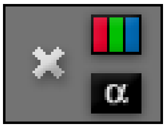
If the Softclip parameter is left at its default value of 0, the distribution curve used to remap the image data remains linear along its entire length from low to high. In other words, this is a “hard” clip of input values above reference white.

When the Softclip is a positive value, the softclip value is subtracted from the Log 90% White value to create a breakpoint below peak white. Above this breakpoint, the slope of the distribution curve becomes nonlinear, gradually leveling off the highs.



20.3 The graph on the right shows how the maximum softclip value affects the redistribution curve, as compared to the default conversion (left).

PREMULTIPLY NODE



The Premultiply node takes a single RGBA input. For every pixel in the image, the Premultiply node multiplies the value of each color component (the red, green, and blue channels) by the corresponding alpha channel value.

Premultiplication is an integral step in the digital compositing process, as it transfers the transparency level represented by the alpha channel to the RGB components themselves before the RGB channels of two separate images are blended in a compositing operation.

In general, CG (computer-generated) imagery is premultiplied, while digitized film footage is not.

WHEN TO USE THE PREMULTIPLY NODE

It is not necessary to use the Premultiply node to composite imagery in RAYZ. Composite nodes will premultiply the input images for you, unless the image has already been premultiplied.

However, the Premultiply node can be used, if desired, to premultiply imagery that will not be subject to a composite operation, as when RAYZ is used strictly for color correction and you want the output rendered as premultiplied imagery for use in another application.

HOW PREMULTIPLY WORKS

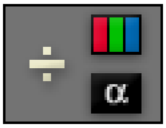
The Premultiply operation is automatic and does not require user modification; therefore, there are no parameters to set in the Premultiply Node Panel.

The Premultiply node performs the following computation, where "A" represents the input image; and *r, g, b,* and *a* represent the red, green, blue, and alpha channels:

Argb * Aa

The multiplication in this formula is done at floating point precision, and for floating point images the alpha channel value is assumed to be in the range of 0–1. For 8-bit and 16-bit images, however, the alpha channel value (Aa) is converted into a decimal fraction first by dividing it by an appropriate value: for 8-bit, the alpha is divided by 255; for 16-bit, it is divided by 65535.

Unpremultiply Node



The Unpremultiply node attempts to restore the RGB channels of a premultiplied RGBA image to their original, non-premultiplied values.

It is important to note that the accuracy of this operation depends on the alpha channel values in the image. If the alpha channel value of a pixel is zero, you lose all information that would allow you to accurately reconstruct the original value of each color component.

When to Use the Unpremultiply Node

Premultiplication is an essential first step in most composite operations (see also “About Premultiplication” in chapter 18, p. 328 for more information), and computer generated imagery, such as the output from a 3D rendering program, is almost invariably premultiplied.

An image is premultiplied with the assumption that it is ready to be composited and that the individual image channel values will not be further adjusted independently.

However, you may very well need to adjust such an image before compositing it over a background—you may need to color correct the RGB channels without affecting the opacity or to adjust the matte edges without affecting the color characteristics. In these cases, you need to work with non-premultiplied image channels to get the results you want.

For example, in a non-premultiplied image, a full red pixel with half alpha coverage would be represented as [1, 0, 0, 0.5]. After premultiplication, the same pixel would be represented as [0.5, 0, 0, 0.5].

Now assume that you adjust the matte such that the alpha channel value for this pixel changes from 0.5 to 0.75.

If this matte adjustment occurs after the pixel has been premultiplied, the value becomes [0.5, 0, 0, 0.75], whereas if the adjustment occurs before premultiplying, the pixel value becomes [0.75, 0, 0, 0.75]. Obviously, these two values would generate a different result for the same pixel in a composite.

Therefore, you may choose to run an image through the Unpremultiply node before adjusting it, and then have the composite node premultiply the adjusted image.

How Unpremultiply Works

The Unpremultiply operation is automatic and does not require user modification; therefore, there are no parameters to set in the Unpremultiply Node Panel.

The Unpremultiply node performs the following computation, where "A" represents the input image; and *r, g, b,* and *a* represent the red, green, blue, and alpha channels:

Argb * Q

"Q" represents the alpha channel value, modified as follows:

Q = Aa == 0 then 1.0 else 1.0/Aa

For 8-bit and 16-bit images, the alpha channel value (Aa) is converted into a decimal fraction (floating point value) first by dividing it by an appropriate value: for 8-bit, the alpha is divided by 255; for 16-bit, by 65535.

chapter 21

TIMING NODES

The Timing node menu provides nodes that change the timing of input sequences, either as their main purpose (the Sequence and 3:2 nodes) or as a secondary effect (Merge and Split).

IN THIS CHAPTER

The **Sequence** node is a comprehensive tool for adjusting image sequences. It duplicates much of the functionality of the Clip Editor to enable you to re-sequence imagery at any point in a network. In addition, the Sequence node enables you to splice frames from different input sequences together. The **Switch** node is used to switch among frames from multiple input images.

The 3:2 nodes, on the other hand, are designed specifically to perform telecine (film transfer) type operations, converting image sequences to and from film and video frame rates. Use **3:2 Pulldown** to transfer film to video rates. Use **3:2 Pushup** to transfer video to film rates.

The **Split** node separates each frame of the input into a pair of "field frames," one odd, one even, such that the output of the Split node is twice the length of the input sequence. The **Merge** node takes an input of these

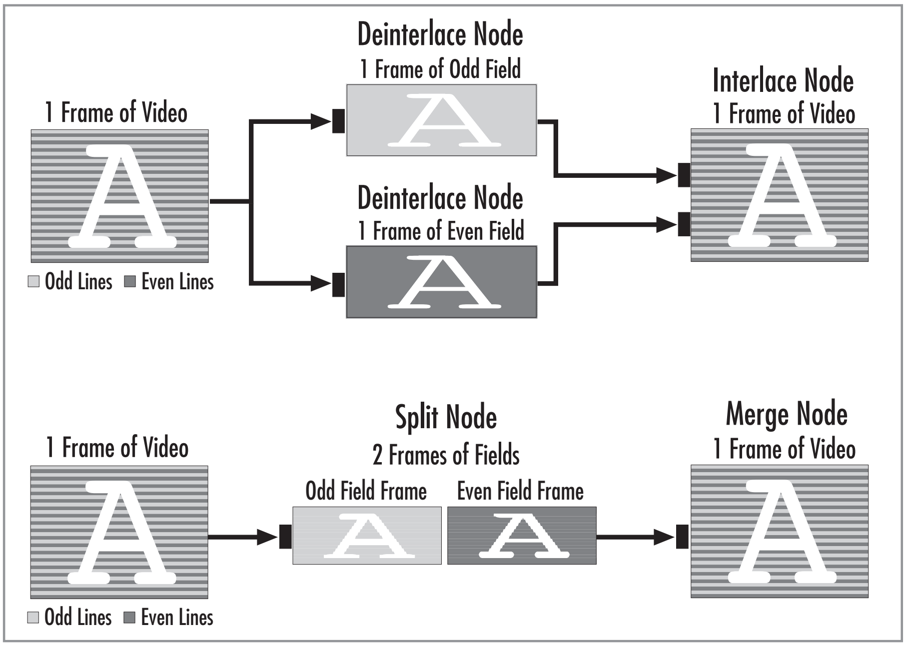
Note: Two more video-related nodes, Interlace and Deinterlace, are described in *Chapter 20: Conversion Nodes* (p. 397).

field frames and merges the fields back together. This means that the output of the Merge node is half the length of the input.

GUIDELINES FOR CHOOSING THE RIGHT VIDEO FIELD NODE

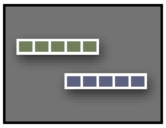
To extract either the odd or even field of each frame, use the "Deinterlace Node" (ch. 20, p. 402). To re-interlace an input consisting of an odd field sequence with a corresponding input consisting of the even field sequence, use the "Interlace Node" (ch. 20, p. 404). (For both of these nodes, the output sequence is the same length as the input.) These nodes are located in the Convert node menu.

To create a single sequence of alternating odd and even fields in separate frames, use the Split node. The Merge node is used to reintegrate the output of the Split node.



21.1 Note the difference between the output of the Deinterlace node and the Split node. The Interlace node is used to reintegrate the output of two Deinterlace nodes, while the Merge node reintegrates the output of a Split node.

SEQUENCE NODE



The Sequence node is a multi-input node that is used to re-sequence image frames. Unlike the Clip Editor, which is used to re-sequence source imagery when it is created or imported, the Sequence node can be used at any point in a network. It also enables you to combine frames from more than one node into a new sequence.

The Sequence node can be used to

- specify which input frames to use
- hold (repeat) frames
- skip frames
- reverse frame order
- loop sequences of frames
- combine sequences from multiple inputs

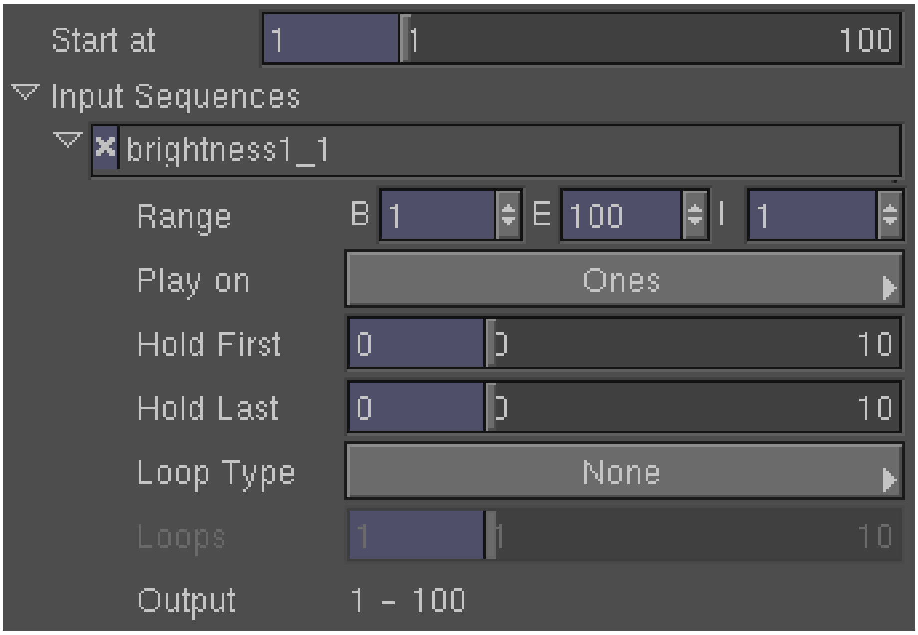
You can connect as many inputs to Sequence as you need. Each time you connect an input, a new connector is created on the Sequence node and a corresponding entry is created in the Sequence Node Panel.

NOTE:

You can connect the same node to Sequence multiple times and a separate entry will be created in the Sequence Node Panel for each input connection. This gives you the option to specify a different range of frames in each entry, and apply different holds and loops to each.

Sequence Parameters

21.2 Sequence node parameters.



At the top of the Sequence Node Panel is an overall parameter used to specify the starting frame of the output sequence. The other parameters are dynamic; that is, a parameter group is created for each input to the node.

Start At

This parameter specifies the frame number in global time (that is, in the Time Scooter) at which the output sequence will begin.

Input Entries

Each input entry includes the same set of parameters to specify range, increment, frame repeats, and loops. At the bottom of each entry is a readout of the output range, which updates as the parameters are modified.

The input entry can be dragged up or down to reorder the entry in the list, as the output sequence is ordered from the top entry to the bottom, as explained in “Working with Multiple Inputs” on p. 418.

Range

The Range parameter provides fields in which to enter the first and last frame of the input to use, as well as the increment. The fields are labeled “B” for Begin, “E” for End, and “I” for Increment.

The default range is the range of the input sequence:

- To use a single frame of the input, enter the same frame number into the Begin and End fields.
- To reverse the frame order—to run the sequence backward—enter the last frame number in the Begin field and the first frame number in the End field.

The default increment is 1, which specifies that every frame be used. An increment value of 2 would specify every other frame, and so on.

RANGE SETTINGS	RESULTING SEQUENCE
Begin: 1; End: 10; Inc: 1	1 2 3 4 5 6 7 8 9 10
Begin: 10; End: 1; Inc: 2	10 8 6 4 2

Play On

The Play On menu is used to repeat, or hold, every frame specified in the Range parameters. The default is to play on Ones, which uses each frame once. To double frames, select play on Twos; to triple frames, play on Threes; and to quadruple frames, play on Fours.

PLAY ON SETTINGS	RESULTING SEQUENCE
Begin: 1; End: 10; Inc: 1 Play on Twos	1 1 2 2 3 3 4 4 5 5 6 6 7 7 8 8 9 9 10 10
Begin: 10; End: 1; Inc: 2 Play on Threes	10 10 10 8 8 8 6 6 6 4 4 4 2 2 2

Hold First and Hold Last

Use the Hold First and Hold Last parameters when you want to hold (repeat) just the first or last frame of a sequence.

The default value in each parameter field is 0, which indicates that the corresponding frame will not be repeated. To hold, say, the last frame for 1 second at film speed, you would enter 24 as the value in the Hold Last field.

HOLD SETTINGS	RESULTING SEQUENCE
Begin: 1; End: 10; Inc: 1 Play on Twos Hold First: 1	1 1 1 2 2 3 3 4 4 5 5 6 6 7 7 8 8 9 9 10 10
Begin: 10; End: 1; Inc: 2 Play on Threes Hold First: 2; Hold Last: 1	10 10 10 10 10 8 8 8 6 6 6 4 4 4 2 2 2 2

Loop Type

The Loop Type menu is used to add one or more loops; that is, to repeat the entire sequence of frames specified in the input entry. The default is

None (no looping); to loop the sequence, select one of the following types of loop from the menu:

- A **Cycle loop** is a simple repetition from first frame to last and then starting over again at the first frame: 1 2 3 4 5 1 2 3 4 5 1 2 3 4 5.
- A **Bounce loop** zig-zags forward and backward, from first frame forward to last and then backward to first: 1 2 3 4 5 4 3 2 1 2 3 4 5.

Loops

The Loops parameter becomes active when a Cycle or Bounce loop is selected in the Loop Type menu. Enter the number of iterations into the Loops field. The default number of loops is 1 (the minimum).

LOOP SETTINGS	RESULTING SEQUENCE
Begin: 1; End: 10; Inc: 1 Play on Twos Hold First: 1 Loops: 1; Cycle Loop	1 1 1 2 2 3 3 4 4 5 5 6 6 7 7 8 8 9 9 10 10 1 1 1 2 2 3 3 4 4 5 5 6 6 7 7 8 8 9 9 10 10
Begin: 10; End: 1; Inc: 2 Play on Threes Hold First: 2; Hold Last: 1 Loops: 1; Bounce Loop	10 10 10 10 10 8 8 8 6 6 6 4 4 4 2 2 2 2 4 4 4 6 6 6 8 8 8 10 10 10 10 10

Working with Multiple Inputs

The output of the sequence node will be the combined sequence created by all of the input entries, in the order in which they appear in the Node Panel list.

Say that you have input the same node into a Sequence node three times, with the parameters for each entry set as follows (any parameter value not explicitly stated is assumed to be at its default value):

First (Top) Input Entry	Begin: 1; End: 10; Inc: 1
Second Input Entry	Begin: 10; End: 1; Inc: 2
Third Input Entry	Begin: 1; End: 5; Inc: 1; Hold First: 4

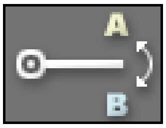
With a Start At parameter value of 1, the output to such a sequence node would be frames 1 to 24; with the image data used for each frame consisting of the following input frames in this order:

1 2 3 4 5 6 7 8 9 10, 10 8 6 4 2, 1 1 1 1 1 2 3 4 5.

TIP:

Use the "Fit Button" (ch. 6, p. 83) in the Flipbook controls to update the frame range to reflect the changes made in the Sequence parameters.

SWITCH NODE



The Switch node enables you to switch among multiple inputs in a specified order.

You can use Switch to control, from one render pass to another, which image data flows into downstream nodes. You can also generate a new sequence made up of frames from multiple input sequences. One common use of Switch is to change the layer order of inputs to the Multi-comp node, so that an object can start out in front of another object in the composite and then move behind it as the sequence progresses.

The Switch node accepts an unlimited number of inputs. (Each time you connect a node to Switch, a new input connector is created.) All inputs must contain the same number of channels and be of the same bit depth.

SWITCH PARAMETERS

The Switch Node Panel provides a single parameter, Use Input, in which you can enter a numerical value or, most likely, a global variable or other expression that specifies the parameter value.

USE INPUT

This parameter specifies which input to use at the current frame: the top input is 1 and subsequent inputs are numbered sequentially in ascending order.

You can create a global variable in the Project Settings panel to control the parameter value, as described in "Creating a New Global" in chapter 13, p. 157.

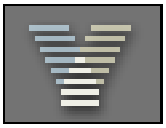
To change the input value over time, animate the parameter, either in the Curve Editor or in the Node Panel. If you need more information about how to animate parameters, see "Using Autokey Mode" (ch. 7, p. 99) in chapter 7 or *Chapter 8: Using the Curve Editor* (p. 105).

You can also use an expression to control the value. For example, to output the imagery in input 1 for the first 24 frames, and then output the imagery in input 2 for the remainder of the sequence:

```
if($F <= 24, 1, 2)
```

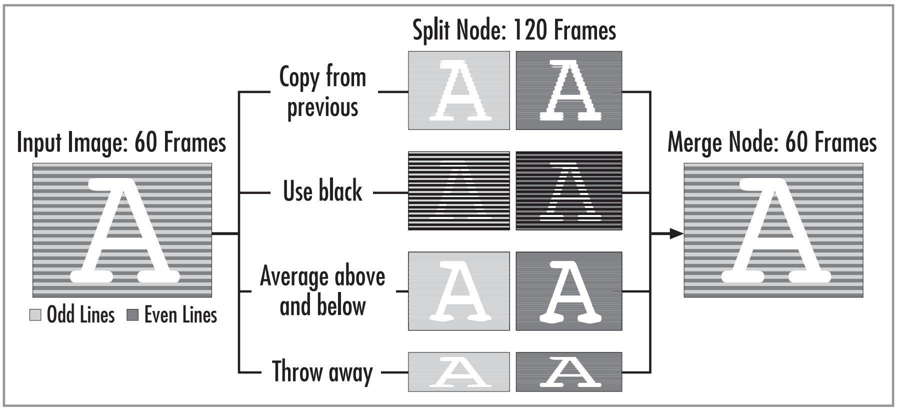
For more information about how to write expressions, see also "Appendix C: Using Expressions in RAYZ" (p. 445).

MERGE NODE



The Merge node enables you to recombine video imagery that has been split into a single sequence of alternating odd and even field frames.

The Merge node accepts one input and can be used to recombine frame fields that you split apart using the "Split Node" (p. 421).



21.3 The Merge node recombines the alternating odd and even field frames output by the Split node.

MERGE PARAMETERS

The Merge Node Panel offers menus for selecting field order and input style. Your choices should be based on the method used previously to split the frames in the Split node.

FIELD ORDER

This parameter allows you to specify the order in which the field frames were generated in the Split node: "Field Two, Field One" (field two first), or "Field One, Field Two" (field one first).

INPUT STYLE

This parameter specifies the input style used when the fields were split:

THROWN AWAY This input style is appropriate for fields that were previously split using the "Throw Away" option in the Split node, which has the effect of shrinking the input image by 50 percent in height.

COPIED This input style is appropriate for all fields that were previously split using an option other than "Throw Away." Select this option to merge fields that were previously split using the "Use Black," "Copy from Previous," or "Average Above and Below" options in the Split node.

SPLIT NODE

The Split node enables you to separate a series of frames into alternating "field frames." Take, for example, a video clip of one second duration (30 frames). The Split node will split such a video clip into a 60-frame sequence that alternates even and odd fields as separate frames. Therefore, the Split node accepts one input and outputs twice as many frames as in the input sequence. See also *Fig. 21.3* (p. 420).

NOTE:

Use the complementary "Merge Node" (p. 420) to recombine the field frames. To de-interlace frames and keep only the odd or even field information in each frame, without creating additional frames, use the "Deinterlace Node" (ch. 20, p. 402) instead of Split.

SPLIT PARAMETERS

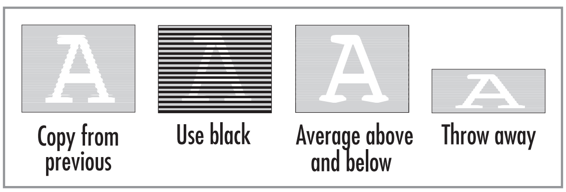
The Split Node Panel provides a pair of menus for specifying the field order and how to treat the other field.

FIELD ORDER

Use the Field Order menu to specify whether the alternating field frames generated by the Split node should begin with field 1 (the odd field lines: 1, 3, 5, etc.) or field 2 (the even field lines: 2, 4, 6, etc.).

OTHER FIELD FILL

Use this menu to specify how to treat the vacant spaces that remain once the odd and even scan lines have been split into separate field frames.



21.4 Options available for filling the discarded field.

Use the Other Field Fill menu to select one of the four available options:

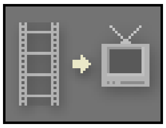
USE BLACK This option fills the vacant lines with black.

COPY FROM PREVIOUS This option fills the vacant lines with data replicated from each previous line.

AVERAGE ABOVE AND BELOW This option fills the vacant lines with data obtained by averaging the preceding and subsequent lines together.

THROW AWAY This option discards the data that formerly occupied the now-vacant lines. The Throw Away option has the effect of shrinking your file by 50% in the Y dimension. (In other words, a file that was originally 720 x 486 will become 720 x 243.)

3:2 PULLDOWN NODE



The 3:2 Pulldown node enables you to convert imagery from the frame rate of film (24fps) to the frame rate of video (30fps).

By video, we are actually referring to the 525/60 scan rate associated with NTSC video. The PAL video standard is associated with the 625/50 scan rate, and the transfer from 24fps to 50fps is typically accomplished by a simple doubling. The "Sequence Node" (p. 415) can be used to double frames by selecting Play On Twos.

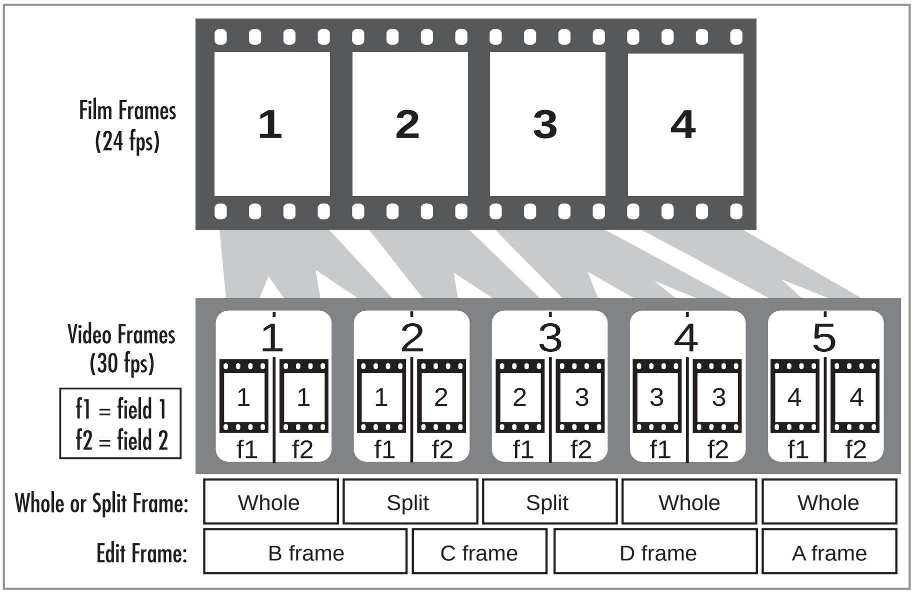
NOTE:

To convert video to the frame rate of film, use the complementary "3:2 Pushup Node" (p. 425).

HOW 3:2 PULLDOWN WORKS

A 3:2 pulldown creates five video fields from every two input frames by replicating one field from every other frame. There are two basic pulldown patterns used to pad the frames to 60 fields per second: 3:2 and 2:3.

21.5 Typical 3:2 pattern: the first film frame is repeated for three video fields and the next film frame for two fields, and so on until the end of the sequence.



SELECTING A PHASE SHIFT

There are actually a total of five different permutations, or phase shifts, of the pulldown pattern to choose from when you need to match other existing video footage that has been edited on fields rather than frames.

The choice of phase shift only becomes important if the film sequence to be transferred will be spliced into other video sequences or if the footage will eventually be transferred back to film speed:

- For footage that will be combined with an existing video sequence, the important thing is to sync the phase shift pattern based on how the other video clip has been edited.
- For footage that will eventually be transferred back to 24fps, the key is to use a matching phase shift pattern for both transfers.

PHASES A – E

In RAYZ, the phase shifts are labeled A, B, C, D, and E. The letters refer to the type of edit frame on which the 3:2 or 2:3 pattern starts. There are four types of edit frame (which are illustrated in *Fig. 21.5*):

- Edit frames A and C each represent a film frame that is used in two video fields. In the case of the A frame, both fields are in the same video frame; while for the C frame, each field is used in a different video frame.
- Edit frames B and D, on the other hand, are each used in three fields. For B, the first two fields are in the same frame, with the third field in a split frame. For D, the first field is in a split frame and the last two fields are in the same frame.

PHASE A starts on the A frame (ABCD), making it a 2:3 pattern.

PHASE B starts on the B frame (BCDA), making it a 3:2 pattern.

PHASE C, which starts on the C frame (CDAB), is a 2:3 pattern where the first edit frame will used in two separate video frames. This may be necessary to match the sequence to the way another one has been edited, given that video is edited field by field, rather than frame by frame.

PHASE D (DABC) is a 3:2 pattern where the first edit frame will used in two separate video frames, for the same reason as Phase C.

PHASE E is the same as A, except that it repeats the initial A frame (A+ABCD).

Depending on the system used for a pulldown, the five permutations may be referred to using different terminology, such as by phase numbers (Accom, e.g.) instead of letters, or by the pattern of whole and split video frames used.

WHOLE AND SPLIT FRAMES

A whole frame uses the same film frame image for both fields, while a split frame uses a different film frame image in each field (see *Fig. 21.5*). For example, WWSSW indicates a repeating pattern of two whole frames followed by two split frames followed by a whole frame.

The following chart can be used as a guide if you, or those you need to communicate with about the process, are familiar with different terms:

Phase 1	Phase 2	Phase 3	Phase 4	Phase 5
Phase A	Phase B	Phase C	Phase D	Phase E
WWSSW	WSSWW	SSWWW	SWWWS	WWWSS
ABCD	BCDA	CDAB	DABC	A + ABCD
2:3	3:2	2:3 (split first)	3:2 (split first)	2 + 2:3

Pulldown Parameters

The 3:2 Pulldown Node Panel provides parameters for selecting the phase shift and field order to use.

Phase Shift

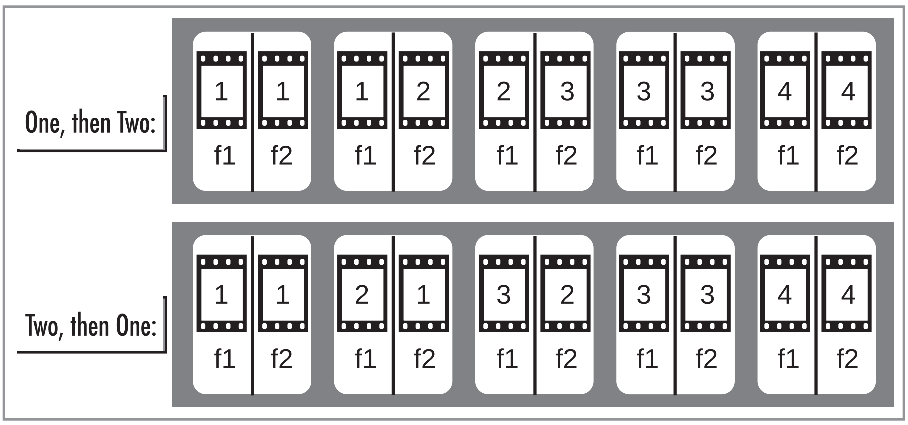
The Phase Shift menu is used to select the type of conversion the 3:2 Pulldown node should use. Select A, B, C, D, or E to specify a repeating pattern. The most commonly used phase shifts are A (a 2:3 pattern) and B (a 3:2 pattern). See also “Selecting a Phase Shift” (p. 422).

Field Order

This menu is used to specify the order in which frames of the input are assigned to the two interlaced fields that make up each video frame of the output. Field 1 has the odd field lines (1, 3, 5, etc.) and field 2 the even field lines (2, 4, 6, etc.).

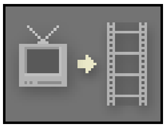
Field Two, Field One This is the default menu choice. Field 2 of the video frame is filled before field 1. In the case of a split frame (where two separate film frames are assigned to the same output video frame), the first film frame image is assigned to field 2, and the subsequent film frame is assigned to field 1.

Field One, Field Two If you select this option, field 1 of the video frame is filled before field 2.



21.6 Field order affects which video field is filled with which frame of film data. Phase Shift B was selected in this example.

3:2 Pushup Node



The 3:2 Pushup node enables you to convert imagery from the frame rate of NTSC video (30fps) to the frame rate of film (24fps). It performs an inverse of a 3:2 pull-down by creating two film frames from five video fields by disposing of one of the fields.

NOTE:

Note: For more information about the film transfer process, refer to the description of the “3:2 Pulldown Node” on p. 422.

Pushup Parameters

The 3:2 Pushup Node Panel provides parameters for selecting the phase shift and field order to use.

NOTE:

When using the 3:2 Pushup node to convert video frames that were originally created from film frames using the 3:2 Pulldown node, use the same Phase Shift and Field Order parameter settings that were used in the 3:2 Pulldown node.

Phase Shift

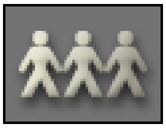
The Phase Shift menu enables you to choose a pattern by which the 3:2 Pushup node will perform the conversion. There are five different phase shifts to choose from, each of which corresponds to a pulldown phase shift as described in the section on “Selecting a Phase Shift” (p. 422) in the 3:2 Pulldown node description.

Field Order

This menu specifies the order in which the two fields that make up each frame of video are processed: “Field Two, Field One” (field 2 first) or “Field One, Field Two” (field 1 first). Be sure to match the field order that was used in the previous pulldown operation, if any, or that of other video footage with which it may be combined.

chapter 22

CREATING GROUP NODES



A Group node is a container that holds other nodes. Nodes can be grouped to help organize large, complex networks into more manageable units or to create customized node operations.

You can group any number of nodes (which can be "ungrouped" later if you wish) and modify individual node parameters and connections within the group node.

You can also customize the Group node itself by specifying which input and output connections from the nodes within the group to use on the Group node, as well as which node parameters to include in the Group Node Panel. This is described later in "Customizing Group Nodes" on p. 430.

Creating and Editing Groups

Here are some key points to remember when working with Group nodes:

- New nodes can be created within a Group node, and nodes can be deleted from a Group node.
- Nodes within a Group node can be connected to each other as well as to external nodes.
- Any parameter in a node within a Group can be added to the Group Node Panel.
- Group nodes can be copied, pasted, cloned, deleted, added to the Custom menu, and otherwise treated like other nodes.

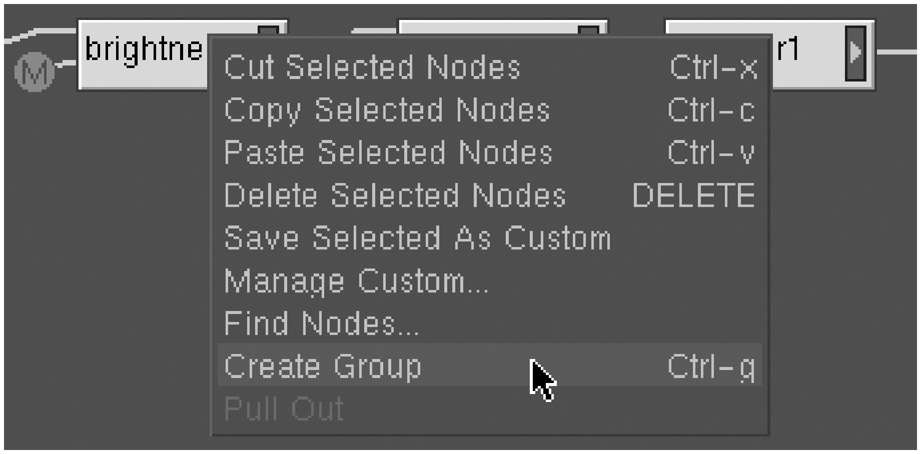
Creating a Group

You create a Group node by selecting one or more nodes in the Worksheet and choosing the "Create Group" command from the Worksheet Actions menu (right-hold in the Worksheet to access this popup menu.) The hot-key for the Group command is **Ctrl-g**.

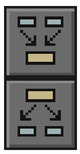
The selected nodes are put in a Group node container, which appears in the Worksheet in place of the selected nodes.

All existing input and output connections to and from individual nodes are retained when the nodes are grouped, including any connections to external nodes (that is, to nodes outside the group).

22.1 The Create Group command is selected from the Worksheet Actions menu to group the selected nodes.



You can also create an empty Group node by selecting Group from the alphabetical list of nodes in the popup Node menu.



22.2 One more way to group or ungroup selected nodes is to click the corresponding button on the Worksheet Toolbar: the top button groups selected nodes; the bottom button ungroups selected Group nodes.

Ungrouping

To "un-group" nodes, that is, to delete the Group container and return the nodes within it to the main level of the Worksheet, select the "Ungroup" command from the Node Actions menu (right-click and hold on the Group node to access). The hotkey is **Shift-Ctrl-g**.

NOTE:

Ungrouping a Group node does not delete its contents. To delete a Group node, along with all the nodes it contains, select the Group node and press the Delete key or use the Delete node command in Node Actions menu of the Group node.

Getting In and Out of a Group

When you create a Group node, you are effectively creating a multi-level Worksheet, in which the nodes within the group are one level down, or in, from the main Worksheet level. The node network in a RAYZ file can be thought of as a directory of nodes, with the Group node being a subdirectory containing the nodes within the group.

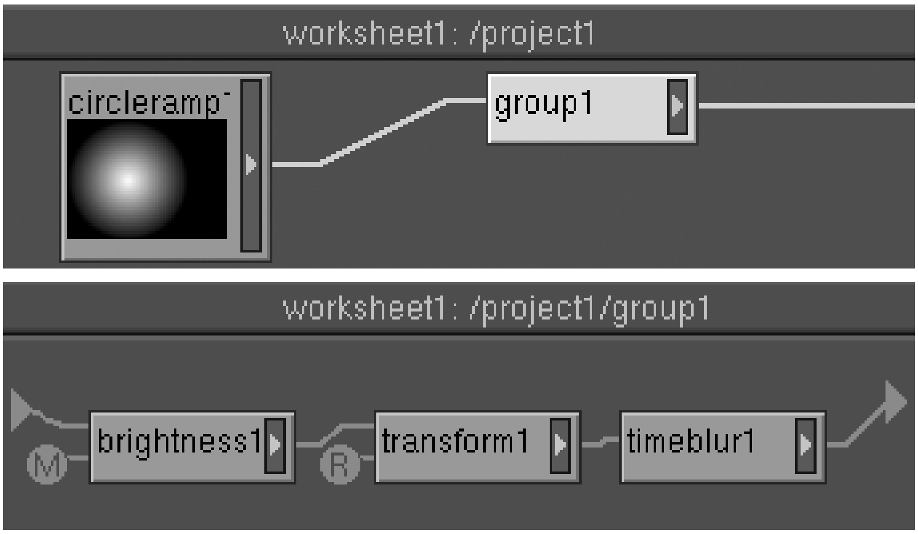
Push In

To access the contents of a Group node for editing, you need to display the Group node level (or "subdirectory") in the Worksheet. This is known as pushing into the Group node.

You push into a Group node by selecting the Push In command from the Node Actions menu (right-click and hold on the Group node to access). When you push into a group, the Worksheet view changes to display the nodes within the Group node.

Pull Out

When you are working within a Group node, select the Pull Out command from the Worksheet Actions menu to pull out of the group; that is, to go back up to the Worksheet level that contains the Group you were in.



22.3 Main Worksheet level with Group node (top), and Worksheet at Group level (bottom).

The title bar of the Worksheet view always displays the current level. At the top level, for example, the title bar would read "/project1" but when pushed into a Group node, the title bar would update to read "/project1/group1" (assuming that the Group node was named "group1").

NOTE:

Group nodes can be nested; that is, new Group nodes can be created within existing Group nodes. However, this is not usually recommended as it is rarely necessary and will probably make the network harder to understand and navigate.

Customizing Group Nodes

Nodes can be grouped to create a customized node, with a unique set of parameters available in its Node Panel, which can then be used like any other single node. This is especially helpful when you have used a combination of several nodes to create a particular effect that may need fine-tuning or is likely to be reused with other imagery.

If you group the nodes, rather than switching from node to node to adjust different parameters, you can add individual parameters from each node to the Group Node Panel where they can all be viewed and adjusted together.

Exporting Parameters

When you create a Group node, the Group Node Panel is blank except for the Edit Group Parameters and Edit Group Inputs/Outputs buttons. But you can assign any node parameter within the group to the Group Node Panel, which is referred to as “exporting” the parameter. Two methods are available for adding/exporting parameters to the Group Node Panel:

- You can use the Edit Group Parameters panel, which lists all node parameters in the group.
- You can push into the group and use the export hotkey.

The panel is easier to use for comprehensive customization, however, if you are already pushed into the Group and know that there are only one or two parameters you need to add to the Group Node Panel, the export hotkey might be the fastest, easiest way.

Using the Edit Group Parameters Panel

Press the Edit Group Parameters button in the Group Node Panel to bring up the parameter editor. By default this panel lists every node parameter in the group, organized by node, in the left pane. Check any parameter to add it and a corresponding entry will be created in the right pane.

The entries in the right pane enable you to organize how the parameters will appear in the Node Panel. You can reorder them by dragging a parameter entry up or down in the list. And you can rename any parameter by double-clicking on the default name and typing new text into the field.

22.4 All nodes in Group are listed in the left pane. The Brightness (in Brightness node) and Translate (in Transform node) parameters have been added to the Group parameter list in the right pane.

SPARE PARAMETERS

To add an extra parameter to the Group Node Panel, press the Add New Parameter button in the lower left corner of the panel. This works just like the extra parameter editor available to all the other nodes. If you need more information about it, refer to the description of "X-Parm" in chapter 7, p. 90.

TIP:

You can add an empty Group node to the Worksheet strictly to create a extra parameter to control multiple parameters in other nodes, even when the nodes are not located within the Group. Giving the extra parameter its own node may make the control more obvious and easy to use in the network.

USING THE EXPORT HOTKEY

Follow these steps:

1. Push into the Group node.
2. Select a node in the group to access its parameters in a Node Panel.
3. Position the cursor over the parameter you want to add to the Group Node Panel and press the **e** key on the keyboard.

To see the result, pull out of the group and select the Group node to display its Node Panel. You still have the option of opening the parameter editor if you want to rename the parameter or make other adjustments.

TIP:

You can create a layout with two Node Panels and pin the Group node to one of them. Then when you push into the Group node and use the other Node Panel to export parameters, you can see the Group Node Panel update at the same time.

Exporting Connections

Like other types of node, a Group node must be connected to other nodes in a network in order to modify those nodes or be modified by them. When the Group node is created, it is given only as many input and output connectors as necessary to maintain any existing connections between the nodes within the group and other nodes in the network.

If you need to make additional connections, however, you can export the input and output connectors of nodes within a group so that they too appear on the Group node. In this way you can make new connections to and from the Group node after it is created.

Two methods are available for exporting inputs and outputs:

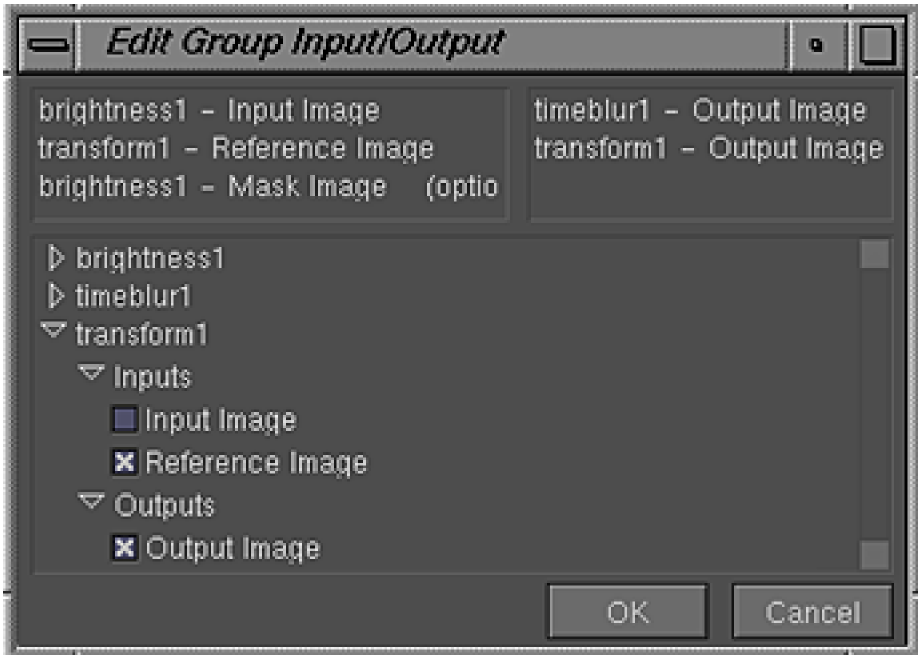
- You can use the Edit Group Inputs/Outputs panel, which lists all connectors available to each node in the group.
- You can push into the group and use the export hotkey.

The panel is easier to use for comprehensive customization, however, if you are already pushed into the Group and know that there are only one or two connections you need to add to the Group node, the export hotkey might be the fastest, easiest way.

Using the Connection Editor

To use the panel:

1. Click the Edit Group Inputs/Outputs button in the Group Node Panel to bring up the connection editor panel.
2. Currently exported connections are listed at the top: inputs on the left; outputs on the right.
3. To add or remove a connection, check or uncheck the box in the list of connections, organized by node, in the bottom of the panel.
4. Click the OK button when you are done to actuate the changes and dismiss the panel.



22.5 All connections are listed at top, inputs on the left and outputs on the right. All possible node connections are listed at bottom, with current exports checked.

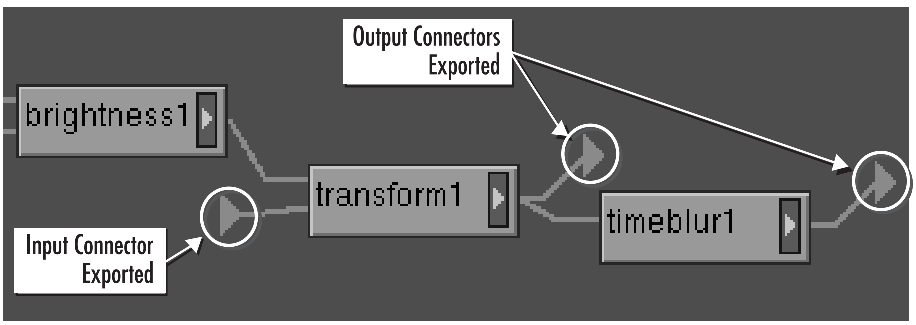
Using the Hotkey

To export a connection using the hotkey:

1. Push into the Group node.
2. Position the cursor over the node connector (input or output) you want to export.
3. Press the **e** key.

How Exported Connections Are Displayed

Whichever method you use, when you are pushed into the group, a truncated connection line will appear to indicate that the connection has been exported, and when you pull out of the group, the additional input or output connectors will appear on the Group node. For this reason, Group nodes can have multiple output connectors.



22.6 This view of the Worksheet (pushed into a Group node) shows input and output connectors that have been exported from the Group.

Input connectors are labeled with the initial of the node to which they connect. For more detailed information about any connector on a Group node, mouse over the connector. The name of the internal node and the type of connector (reference input, garbage matte, etc.) will be displayed in the Status bar.

22.7 Position the cursor over any input or output connector to get information about the connection in the Status bar.

Creating File Groups

The Sources node menu in the Worksheet contains a node called a "File Group Node" (ch. 14, p. 173), which is used to specify an external RAYZ file that has been configured as a group node, with customized Node Panel parameters and node inputs and outputs.

Unlike a Group node you create within a RAYZ file, the contents of a File Group node cannot be edited. File Groups are used to distribute pre-configured effects to compositors throughout a facility to produce consistent results.

The big advantage of File Groups is that the original RAYZ file can be edited, and any File Group node in any other file that points to it can be updated.

To create a File Group, you treat a top-level network of nodes as a group, exporting connections and parameters using the panels accessed from the bottom menu button on the Worksheet Toolbar, which contains the following selections:

- Edit Group Inputs/Outputs
- Edit Group Parameters
- Launch Group Node Panel

The Edit panels work exactly like the ones accessed from the Node Panel of a Group node, which were described in previous sections of this chapter. The only difference is that they affect the top level of the Worksheet. For more information about Worksheet levels, see "Getting In and Out of a Group" (p. 428).

Editing File Group Inputs and Outputs

The Edit Group Inputs/Outputs panel exports connections in the Worksheet, creating the same export icons you would see when pushed into a group: see *Fig. 22.6* (p. 433). This panel works the same way as the one described in "Exporting Connections" (p. 432).

EDITING FILE GROUP PARAMETERS

The Edit Group Parameters panel exports parameters to the File Group Node Panel, and it works the same way as the one described in "Exporting Parameters" (p. 430). To see the contents of this custom Node Panel, however, you must select Launch Group Node Panel from the File Group menu in the Worksheet Toolbar.

SAVING A RAYZ FILE FOR USE AS A FILE GROUP

When a RAYZ project file is imported into the File Group node, the entire project file becomes a group node. This means that the file you create to be imported by other compositors should not have any extraneous nodes in it, and all necessary input and output connectors should be exported. Also, the end user will not be able to push into the File Group, so all necessary parameters must be exported to the Node Panel.

PART

Four

APPENDIXES

Part IV contains the appendixes to the RAYZ manual, which provide technical details about specific topics covered in the main chapters, or auxiliary topics such as scripting and writing plugins.

IN PART IV: APPENDIXES

HOW RAYZ COMPUTES LUMINANCE VALUES

Many RAYZ nodes can create luminance channels for various operations, with the option of selecting Film, NTSC or PAL luminance. Here is how each type of luminance is computed in RAYZ:

Film

RAYZ computes the film luminance value as follows:

red channel * 0.333 +

green channel * 0.333 +

blue channel * 0.333

NTSC Video

RAYZ computes the NTSC luminance value as follows:

red channel * 0.299 +

green channel * 0.587 +

blue channel * 0.114

PAL Video

RAYZ computes the PAL luminance value as follows:

red channel * 0.2125 +

green channel * 0.7154 +

blue channel * 0.0721

IMAGE FILE FORMATS SUPPORTED BY RAYZ

The current version of RAYZ imports and exports the file formats in the list below. However, you can add support for other image formats by writing a RAYZ plug-in.

FILE FORMAT	DEFAULT EXTENSION
Adobe Photoshop	psd
Alias	als
Alias Camera Depth	depth
Chalice Direct Image	cdi
Cineon DPX	dpx
Cineon Fido	cin
FRP (Fast real-time Playback)	frp
JPEG	jpg
Maya	iff
Microsoft Windows Bitmap	bmp
PNG (Portable Network Graphics)	png
QuickTime	mov
Raw	raw
SGI Image	rgb
Softimage	pic
Targa	tga
Tiff	tiff
Wavefront RLA/RLB	rla
YUV	yuv

Image Format Notes

This section provides important information which should be reviewed before importing and exporting the following image file formats.

Refer to the sections of the Image In node description on "Conversion Parameters" (ch. 14, p. 181) and "Format-Specific Import Options" (ch. 14, p. 187) for specific information about using the parameters relevant to importing and converting image file formats.

For more information about specifying format options when writing image files, refer to the "Format" (ch. 14, p. 191), "Conversion" (ch. 14, p. 192), and "Format Specific" (ch. 14, p. 193) parameter descriptions in the Image Out node.

Cineon Fido

RAYZ can read and write Kodak Cineon 10-bit log imagery in Fido format. This 3-channel format is by far the most common type of Cineon file.

By default, the Image In node automatically remaps Cineon 10-bit log files to 16-bit linear colorspace. However, you can change the conversion options or use the unconverted log data by setting the appropriate parameters in the Image In Node Panel.

Displaying Log Images in the Viewer

By default, unconverted log images are displayed in the Image Viewer using Raw Log display conversion, which shifts the values up in the colorspace to make them visible without attempting to remap the nonlinear distribution to linear.

To view log data remapped to linear, choose Cineview emulation instead, or add a custom display LUT. Viewer display options are described in the section on "Display Conversion" in chapter 6, p. 72.

Cineon DPX

RAYZ can also read Kodak Cineon imagery in DPX format, which may be linear or log (or have some other type of non-linear gamma), may contain 1, 3, or 4 channels, and may be encoded as 8-bit, 10-bit, 12-bit, or 16-bit data. RAYZ also recognizes the orientation of the image.

RAYZ can write DPX files of 1, 3, or 4 channels, encoded as 10-bit log or as 8- or 16-bit linear.

FRP

FRP (pronounced "ferp") is a Silicon Grail file format optimized for Fast real-time Playback. It is used to create a compact flipbook file for in-house playback and review of a shot or shot element.

When you select FRP from the Format menu in the Image Out node or the corresponding entry in the Render Control view, RAYZ writes the specified image sequence to disk at its native spatial resolution as a single .frp file of 8-bit data. You can then load this file into any RAYZ project file and play it in a Viewer.

Photoshop

An Adobe Photoshop format file may contain multiple image layers. There are two ways to import such a file into RAYZ, depending on whether you need to maintain the separate layers.

When imported into the Image In node, a multi-layer Photoshop file is flattened by default, although you can override this option in the Image In Node Panel if you want to import any single layer of the image. See also "Format-Specific Import Options" in chapter 14, p. 187.

Another method is available that imports the Photoshop file into a new RAYZ project file and retains each layer as a separate image. The "Import File…" (ch. 9, p. 117) option in the RAYZ File menu automatically imports each layer of the Photoshop file into a separate Image In node, and connects the Image In nodes to a Multi-comp node to recreate the multi-layer Photoshop image.

Maya

Either the RGBA channels or the Z channel of a 5-channel Maya file can be imported into a single Image In node, but not all five channels, because the depth data is encoded as float while the color and opacity channels are encoded as 8- or 16-bit.

The Image In node provides a parameter you can use to select which Maya channels to import, so that you can import the RGBA data into one Image In node and the corresponding z-buffer into another. See also "Format-Specific Import Options" in chapter 14, p. 187.

QuickTime

RAYZ 2.2 can read QuickTime 3 compliant files that use the Animation, JPEG, Motion-JPEG-A, YUV2, and uncompressed codecs. The file may include audio (although RAYZ 2.2 cannot play the audio track).

YUV

Unless you change the default, YUV files will be imported as floating point to ensure that all the image data is preserved. In addition, RAYZ assumes that imported YUV files have a gamma of 2.2. You can change the default bit depth and gamma specification for YUV files in the Image In node. See also "Format-Specific Import Options" in chapter 14, p. 187.

Wavefront

Like some Maya image files, Alias/Wavefront RLA and RLB file formats may contain channels of data other than the RGBA channels. These channels can be imported into a separate node from the RGBA data by using the appropriate parameter in each Image In node to select which channels to import. See also "Format-Specific Import Options" in chapter 14, p. 187.

appendix C

USING EXPRESSIONS IN RAYZ

An expression is a combination of numbers, symbols, and/or words that represent a numerical value. An expression can be used in a parameter, in place of a fixed numerical value, to animate values across time or to reference values in another parameter.

IN THIS CHAPTER

The Benefits of Using Expressions

Expressions are not just for performing esoteric or complex modifications, although they can certainly be used for that purpose. Rather, expressions are also an easy way to get common tasks done faster and more consistently. This is because an expression can access the values in any other parameter, updating automatically to reflect any change in the referenced value. The value referenced can be used as is, or it can be further modified.

Say, for example, that you've animated a change in brightness in a foreground element to match the fluctuating light levels in a background plate. You can use an expression to access that same brightness curve to adjust other foreground layers. And you can modify the expression to raise or lower the overall brightness level of any image layer without affecting the relative change in brightness from frame to frame specified in the referenced animation curve.

Expressions Are Easy to Use

Expressions are a powerful way to control node operations, but even the most math-phobic user can learn to write basic expressions by reviewing the guidelines in the section on "Writing Valid Expressions" (p. 448). All you need to know are some of the symbols used (the vocabulary) and the order in which they are strung together to create meaningful expressions (the syntax). Then you can use the procedure described in "How to Enter an Expression" to modify a parameter.

In fact, you don't have to memorize a lot of math functions and parameter syntax. The Expression Editor automates part of the process as described next.

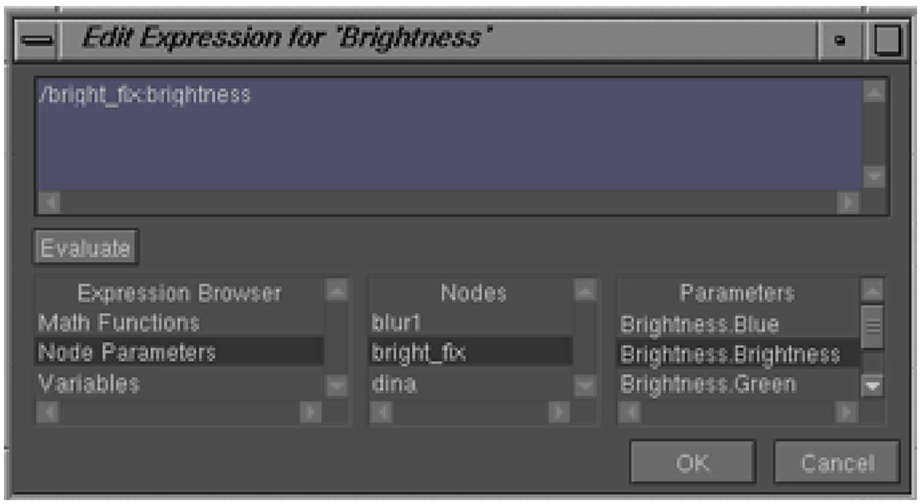
How to Enter an Expression

You can use expressions in any node parameter in RAYZ that features an Animation menu. Expressions can be specified in the Expression Editor, a floating panel that you can access from the Node Panel or from the Curve Editor:

- The Expression Editor can be accessed directly from the Node Panel for all parameters that control a single value, such as blur or brightness.
- For a parameter that controls two or more values—which in most cases means a parameter that specifies both *x* and *y* coordinate values, such as scale or translate—you can access the Expression Editor for each value separately by selecting its curve in the Curve Editor.

Using the Expression Editor

To access the Expression Editor in the Node Panel, select Edit Expression from the parameter Animation menu or use the hotkey equivalent, Shift-e, while the cursor is over the parameter you want to modify. (See also "Entering Expressions in the Curve Editor" on p. 447.)



C.1 Expression Editor.

The Expression Editor consists of a text entry field at the top, in which you can type an expression, and the Expression Browser at the bottom.

You can use the rules outlined in "Writing Valid Expressions" (p. 448) to type an expression into the field. However, you can also use the Expression Browser to help build the expression for you.

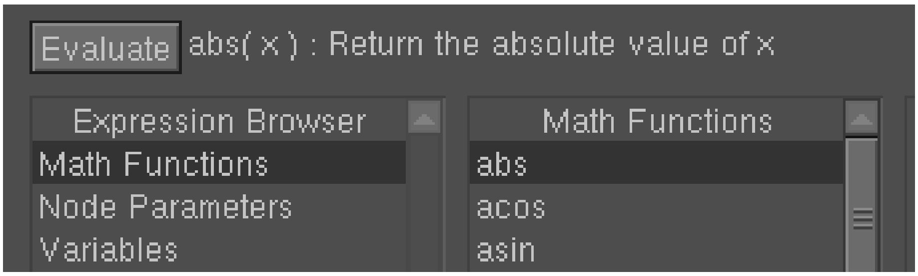
EXPRESSION BROWSER

The Expression Browser lets you choose valid math functions, variables, and node parameters to use in an expression. When you select a category in the browser, a corresponding list appears in the middle pane.

When you select Math Functions, a list of math functions appears; when you select Variables, a list of variables appears, and when you select Node Parameters (as illustrated in *Fig. C.1*), a list of all nodes in the file appears.

When you select a node from the list in the middle pane, a list of all parameters in that node appears in the pane on the right.

If you double-click on a specific math function or variable listed in the middle pane, or a specific parameter listed in the right pane, it will appear in the expression field in the proper syntax. You can then modify it as needed.



C.2 Select any math function in the list to get a description.

EVALUATE BUTTON

Press the Evaluate button to find out if an expression is valid before closing the editor. If there is an error, you can modify the expression and press Evaluate again to check your changes.

ENTERING EXPRESSIONS IN THE CURVE EDITOR

There are two methods you can use to enter expressions in the Curve Editor. You can type expressions directly into fields in the Keypoint Viewer, which shows all animated curves in the graph, or you can use the same Expression Editor that is accessed from the Node Panel.

Follow these steps to use the Keypoint Viewer to enter expressions:

1. Display the parameter you want to modify in the Curve Editor graph and open the Keypoint Viewer panel, which is accessed from the Tools menu in the title bar of the Curve Editor.
2. Ctrl-click on the curve to add a keypoint, which should be positioned one frame past the end of the frame range you want to affect. This defines the curve segment between the keypoints to which the expression will be applied.

3. Right-hold on the curve and select “Expression” from the popup menu An Expression field will appear for the curve in the Keypoint Viewer.
4. Type the expression into the field. When entering similar expressions in multiple fields, you can copy and paste (Ctrl-c, Ctrl-v) them, editing the copies as necessary.

Follow these steps to take advantage of the Expression Editor when working in the Curve Editor view:

1. Display the parameter you want to modify in the Curve Editor graph.
2. Ctrl-click on the curve to add a keypoint, which should be positioned one frame past the end of the frame range you want to affect. This defines the curve segment between the keypoints to which the expression will be applied.
3. Right-hold on the curve and select “Expression” from the popup menu.
4. Right-hold on the curve again, and the popup menu will now offer the “Edit Expression” option. Select it from the menu to open the Expression Editor.
5. Type the expression in the field or use the “Expression Browser” (p. 447) to access valid math functions and node parameter syntax.

TIP:

You can still open the Keypoint Viewer to view and edit expressions created in the Expression Editor, which is helpful when using expressions to control multiple parameter curves.

WRITING VALID EXPRESSIONS

You can use numbers, arithmetic operators, functions, variables, and parameter names to write expressions. Numbers and variables can be thought of as the “nouns” of the expression, while operators and functions act as the “verbs” that express the action to be performed. The numbers can be positive or negative, integers or decimal fractions (i.e., floating point notation).

NOTE:

Floating point numbers can also be expressed in exponential notation, a shortcut you can use to express large numbers. For example, `3.2e-5` is equivalent to 0.000032. (To actually compute a number as an exponent, however, you would use the exponent operator or the pow function.)

OPERATORS

Operations are evaluated in the following order:

( )	operations in parentheses are evaluated first
^	then exponents
* / %	* multiplication, / division, and % modulus next
+ -	+ addition, - subtraction last

Operators that have the same priority, such as multiplication and division, are evaluated in order, reading from left to right, as illustrated in the following examples:

SAMPLE EXPRESSION	RESULT	DESCRIPTION
`3 * 4 / 5`	2.4	Multiplication and division have the same priority, but the multiply operator occurs first in the expression, so 3 * 4 = 12, and 12 / 5 = 2.4.
`3 + 4 * 5`	23	The multiply operation, 4 * 5, is evaluated first and the result, 20, is added to 3.
`(3 + 4) * 5`	35	The add operation is evaluated first because it is in parentheses: 3 + 4 = 7, and 7 * 5 = 35.
`2 * 3 ^ 2 + 4 * 6 / 2`	30	Since the add operation is evaluated last, this expression is equivalent to 18 + 12. That is, 3 ^ 2 = 9, and 9 * 2 = 18. And on the other side of the plus sign, 4 * 6 = 24, and 24 / 2 = 12.
`( (3 + 4) * 5 - 5) / 6`	5	Innermost parentheses are evaluated first: 3 + 4 = 7. Then the operations in the outer parens: 7 * 5 = 35, and 35 - 5 = 30. Finally, 30 is divided by 6.
`13 % 5 + 13 / 5`	5.6	The result of 13 % 5, which is 3, is added to the result of 13 / 5, which is 2.6, to get 5.6. *Note:* "13 % 5" is read as "13 modulo 5." This operator divides the first number by the second and returns the remainder: 3 is the remainder of dividing 13 by 5.

VARIABLES

A global variable is a name consisting of one or more uppercase letters, and each name starts with a dollar sign ($) to identify it as a variable.

Variables often represent a value that changes. The $F variable, for example, represents the current frame number. Many variables, like $F, are defined by the RAYZ system and cannot be changed.

The $JOB variable, on the other hand, which represents a directory path, can be redefined in Edit > Project Settings > Globals. You can also create your own global variables in the Project Settings panel, which will subsequently appear in the Expression Browser Variables list.

Global variables available in expressions include:

$F	current frame number
$W	current image width
$H	current image height
$JOB	current job directory (user definable)

A number of other variables, including $PATH, $USER, $SHELL, $HOME, and $PWD, are also available. Refer to the list in the Expression Editor (click Variables in the Expression Browser) for the most up-to-date listing.

FUNCTIONS

You can use any of the following functions in an expression by typing the function name followed by parentheses. Within the parentheses, you can put any numerical value, or any expression or variable that evaluates to a numerical value.

You can also use a parameter address in a function when you want to reference the values in another parameter (see "Using Expressions to Reference Other Parameters" on p. 452). Some functions require multiple values, which are separated by commas.

FUNCTION	DESCRIPTION	EXAMPLE
abs(val)	Returns the absolute value of val.	abs(-2.6) = 2.6
clamp(val, min, max)	Clamps the range to min through max. Used to prevent val from going outside the specified range. Val is usually a variable or a pointer to another parameter.	clamp(1.2, 0, 1) = 1
floor(val)	Returns the largest integer smaller than val.	floor(2.78136) = 2
min(val1, val2, ...)	Returns the smallest val in the list. Can be used to get min of two referenced parameters, or of a referenced parameter and a specific value.	min(2, 7, 1.6) = 1.6
max(val1, val2, ...)	Like min() function listed above, but returns the largest val in the list.	max(-1, 0.1) = 0.1
pulse(val, min, max)	Creates an on/off pulse. If val is less than min or greater than max, pulse returns 0, otherwise it returns 1.	pulse(1, 0, 0.5) = 0
trunc(val)	Truncates all digits to the right of the decimal in val.	trunc(4.567) = 4.0

LOGARITHMIC/EXPONENTIAL FUNCTIONS

FUNCTION	DESCRIPTION	EXAMPLE
exp(val)	Logarithmic exponentiation function.	exp(2) = 7.38906
log(val)	Natural logarithm of val.	log(2.718281828) = 1
log10(val)	Logarithm base 10 of val.	log10(10)= 1
pow(base, exponent)	Computes base to power given.	pow(2, 3) = 8

TRIGONOMETRIC FUNCTIONS

All angles are in degrees unless otherwise noted.

FUNCTION	DESCRIPTION	EXAMPLE
acos(val)	Trigonometric arc cosine of val.	acos(0) = 90
asin(val)	Trigonometric arc sine of val.	asin(0.866025) = 60
atan(val)	Trigonometric arc tangent of val.	atan(1.73205) = 60
atan2(float y, float x)	Compute the arc tangent of y over x. This is more stable than atan() since it can use the signs of y and x to determine the quadrant the angle is in. It also handles correctly the case where x is zero, returning 90 or -90.	atan2(1, 0) = 90 atan2(0, 1) = 0 atan2(0, -1) = 180
cos(val)	Trigonometric cosine of val.	cos(60) = 0.5
cosh(val)	Hyperbolic cosine of val.	
rad(val)	Convert val to radians (val is in degrees).	rad(180) = 3.1415926
sin(val)	Trigonometric sine of val.	sin(60) = 0.866025
sinh(val)	Hyperbolic sine of val.	
tan(val)	Trigonometric tangent of val.	tan(60) = 1.73205
tanh(val)	Hyperbolic tangent of val.	

USING EXPRESSIONS TO REFERENCE OTHER PARAMETERS

You can use expressions to address other parameters, that is, to access the current value in another parameter. The referenced parameter can be in the same node or a different node, and the other node can be of the same type or a different type. You can enter the parameter name alone to use the value as is, or you can modify it using the operators and functions described previously.

PARAMETER NAMES USED IN EXPRESSIONS

The syntax used to reference parameters, at it simplest, is

/node:parameter

That is, the unique name of the node, as specified in the Name field of the Node Panel, followed by the parameter name. Note the slash preceding the node name and the colon that separates the node name from the parameter name.

Assume, for example, that you have animated the brightness value of an image in a node named “bright_fix1” and you want to apply the same correction to another node image. You have connected the second image to a new Brightness node that you have named “bright_fix2.” You would enter the following expression in the Brightness parameter of “bright_fix2” (see “How to Enter an Expression” on p. 446 if you need instructions):

```
/bright_fix1:brightness
```

TIP:

The parameter name RAYZ recognizes is the same name listed for the parameter in the Keypoint Viewer of the Curve Editor. When in doubt, you can find out the correct name to use by viewing the node in a Curve Editor.

C.3 Expression field for a keypoint in the Keypoint Viewer panel of the Curve Editor.

Sub-Parameters

Now let’s assume that you want to reference the brightness value of the Blue channel only. To reference an individual channel parameter within a parameter group, you would type the name of the master parameter followed by the sub-parameter name:

```
/bright_fix1:brightness.blue
```

This holds true for any type of parameter group, or any parameter with multiple value fields. To address a sub-parameter, type the master parameter name followed by the sub-parameter name, separated by a period.

Parameters in the Same Node

There is a shortcut you can use when addressing another parameter in the same node. Leave out the node name and type the parameter name alone. For example, if you want to use the Blue channel brightness value in the brightness parameter for the Red channel:

```
brightness.blue
```

Dynamic Parameters

Dynamic parameter entries in Node Panel lists, such as track point entries in the Track node and input entries in Multi-comp are named according to the type of list, with each entry numbered in order of creation, starting from 0:

/node:listentry[n].parameter

NOTE:

The name that you give the entry in the Node Panel, as when you give each track point a distinctive name such as “bottom_left_corner,” does appear in the Curve Browser to help you identify the parameters. However, the name used to address the parameter in an expression is determined by RAYZ. This is the name used to label the corresponding parameter in the Keypoint Viewer.

To reference the *x* coordinate data in the first track point in a node named “track1”:

```
/track1:points[0].position.x
```

The following expression would reference the opacity parameter of the second input to a Multi-comp node named “multicomp1”:

```
/multicomp1:inputlist[1].opacity
```

NOTE:

RAYZ uses the order of connection to number the input layers to Multi-comp. This means that the parameter name referenced will not change if you change the order of the layers in the composite.

Parameters in Group Nodes

If you wanted to reference a node that is inside a Group node, add the name of the Group node to the front of the parameter address:

/group_node/node:parameter

The following would reference the brightness parameter in a Brightness node named “brightness1” inside a Group node named “group1”:

```
/group1/brightness1:brightness
```

Modifying Referenced parameter Values

As previously mentioned, you can modify the parameter value being referenced; that is, you can treat the parameter address like any other value in an expression that uses the operators and functions described above.

To reduce the overall brightness specified in the “bright_fix1” node example cited earlier, while still retaining the shape of the animation curve, you could modify the expression as follows (this would reduce the amount of change by 10 percent):

```
/bright_fix1:brightness * 0.9
```

To get the inverse of the brightness curve:

```
1.0 - /bright_fix1:brightness
```

When referencing tracking or other position data, you can offset the position by adding or subtracting the distance of the offset, expressed as a percentage of total width or height:

```
/track1:points[0].position.x - 0.263
```

```
/track1:points[0].position.y + 0.387
```

Or you could use the min() function to get the smaller of two referenced parameter values:

```
min(/track1:points[0].position.x,
/track1:points[1].position.x)
```

Specifying Values by Frame

The $F variable, which returns the current frame number, can be used in a parameter address to specify that the values from specific frames be referenced. The variable should be in braces (curly brackets) at the end of the parameter name:

```
/bright_fix1:brightness{$F}
```

```
/bright_fix1:brightness.blue{$F}
```

```
/track1:points[0].position.x{$F}
```

The expressions above illustrate the syntax to use; in these cases, the result is the same as if the “{$F}” had not been appended. It becomes useful, however, when you want to specify a frame offset.

This expression, for example, would use the referenced parameter value from the previous frame as the current frame value in the parameter being modified:

```
/bright_fix1:brightness{$F-1}
```

This next example shows how you could use frame offsets to create a triangle filter to smooth the values of a track point:

```
(/track1:points[0].position.x{$F-1} * 0.25 +
/track1:points[0].position.x{$F} * 0.5 +
/track1:points[0].position.x{$F+1} * 0.25)
```

It adds a percentage (25%) of the values of the previous and next frame to (50 percent of) the current frame value to generate the smoothed value.

To access the parameter value from a specific frame, replace the $F variable with a specific frame number. This would return the brightness value at frame 6:

```
/bright_fix1:brightness{6}
```

appendix D

SCRIPTING

You can use scripting commands to execute common operations from the command line without opening the RAYZ interface. In addition, you can write scripts to create RAYZ project files that will be opened and further modified in the interface.

IN THIS CHAPTER

COMMAND-LINE EXECUTION OF SCRIPTING COMMANDS

RAYZ 2.2 adds a wealth of new command-line capabilities, allowing Technical Directors and others to execute common activities without opening up the RAYZ interface. Apart from being simpler and quicker to do some things, this also has the advantage of only using a render license, instead of the full interface license.

The basic form of command-line execution of RAYZ is as follows:

rayz <input_images> [operation(s)]

where the <input_images> argument is required, and all the [operations] are optional. In general, <> angle brackets indicate required parameters, and [] square brackets indicate optional values.

Usually, one of the operations will be to write out the processed images, though that is also optional. The minimal command of

```
% rayz image.tiff
```

will launch the RAYZ interface with the single "image.tiff" image as input. In general, without the -write command, the command-line version will launch the GUI with the listed operations as nodes. This can be useful for debugging or just getting started on a RAYZ session without having to use the file browser or node menus.

SPECIFYING INPUT IMAGE SEQUENCES

The input image(s) must be specified in RAYZ format, in which $F indicates a frame number, such as 34, 6, and 195. The format $F4 indicates a padded frame number, such as 0034, 0006, and 0195. The '4' indicates how many digits are in the frame number, so that $F1 is equivalent to $F; $F2 will return 34, 06, and 195; $F3 will give you 034, 006, and195; and $F4, 0034, 0006, and 0195.

Generally, the $ symbol must be 'escaped' from the IRIX or Linux shell that you are using, with the backslash character, as in

```
% rayz images.\$F4.tiff
```

If you are constructing a shell script which will issue the commands, you will probably need multiple escape characters, because the shell script is itself interpreted before being passed to the shell. So inside a shell script, you will end up with something like this:

```
#!/bin/csh -f

rayz images.\\\\$F4.tiff
```

When this line is passed to the shell, it is turned into this

```
rayz images.\$F4.tiff
```

which is then received by RAYZ as

```
images.$F4.tiff
```

FRAME RANGES

RAYZ will scoop up all the images that match the input pattern, regardless of their numbering scheme. So in a directory of images which start at frame 120 and go to 125, that is

image.0120.tiff
image.0121.tiff
image.0122.tiff
image.0123.tiff
image.0124.tiff
image.0125.tiff

the command

```
% rayz image.\$F4.tiff
```

would read in the 6 frames, one at a time, from 120 to 125. It is not currently possible to specify a range of frames on the command line. For example, it would not be possible to only process frames 121-124 in the above range; it is all or nothing. This will be addressed in a future release of RAYZ.

Saving the Results

The command -write is used to copy the results of the processing to some new disk files. The syntax of this command is

-write <format> <namestring>

where <format> is one of jpeg, sgi, tiff, or any of the other formats RAYZ supports. You can discover all the currently supported formats by typing

```
% rayz -help-file-types
```

As of this writing, the list is:

adobe
alias
aliasdepth
cdi
cinedepth
com
dpx
fido
frp
icon
jpeg
maya
msbmp
null
pgm
png
quicktime
raw
sgi
softimage
targa
tiff
wavefront
yuv

The <namestring> is a formatted name string just the same as the input file string. So an example of the -write command would be

```
-write yuv video\$F4.yuv
```

Some Simple Examples

You can use the simple tools we have so far to do file format conversion. For example, to convert from .tiff to .jpeg, the command would be:

```
% rayz inputs.\$F.tiff -write jpeg outputs.\$F4.jpeg
```

To blur the contents of a directory, the command would be:

```
% rayz dino.\$F.sgi -blur 5p -write sgi
dino_blurred.\$F.sgi
```

This uses the -blur command, which takes a single parameter, the blur width.

SPECIFYING PIXEL OR FLOAT VALUES

The blur width in the example above is given as "5p," which means 5 pixels.The alternative would be a value of 0.01f, which means a blur size of (0.01 * image_size). The 'f' stands for 'float,' which means that the value is relative to the size of the incoming image.

The 'f' or 'p' character is optional; but if not specified, then 'pixels' is assumed. So the above example could also have been written as

```
% rayz dino.\$F.sgi -blur 5 -write sgi
dino_blurred.\$F.sgi
```

Similarly, doing -resize to change the size of an image can be done to an absolute pixel size, like this:

```
-resize 320 243
```

Or it can do a relative resizing, for example, making the image 50% of its original size, like this:

```
-resize 0.5f 0.5f
```

You can also mix these within the same command, so that the following is completely valid:

```
-resize 320 0.25f
```

Note that 0.25 might be an obvious fraction to you, but without the 'f' character at the end, RAYZ will assume that you want to size to 1/4 of a pixel! To be safe, always specify 'p' or 'f.'

ADDING COMPLEXITY

Commands can be chained together to form an assembly line of processes. This is especially useful when constructing a shell script which does several things at once to a series of images. For example, say you want to take a series of images, flip them horizontally, scale them down, and convert them to jpeg. You can do this all in a single command, which would look like

```
% rayz images.\$F.sgi -flipflop horizontal -resize
320 243 -write jpeg stamps.\$F.jpeg
```

The operations are applied to the images in left-to-right order.

You can also do limited compositing operations on the command line. To do A over B, for example, looks like this:

```
% rayz imagesA.\$F.sgi -over imagesB.\$F.sgi -write
sgi comp.\$F.sgi
```

The syntax for the -over (and -add, -atop, -difference, -inside, -multiply, -outside, -subtract and -under) operation is

-over <b_input_files>

You can also do a dissolve, with the added parameter of the dissolve amount:

```
% rayz imageA.sgi -dissolve 0.5 imageB.sgi -write sgi
out.sgi
```

When using commands with long names, you can use the shortest string which uniquely identifies that command. So instead of -dissolve, you could write -dis, and instead of -difference you could write -dif.

FULL COMMAND LIST

The full list of possible commands can be obtained by typing

```
% rayz -file -help
```

When you type the help command, the options will be listed with the syntax "--name"; but in this document, as you may have noticed, the single dash is used, as in "-name." Either will work.

Here is a list of current commands, with examples where useful. Again, note that the syntax is <required_parameter> and [optional_parameter]:

SOURCES

COLOR

-color <width> <height> <8|16|32> <a[lpha]|rgb|rgba>
<color_values...>

Color values are given in floating point, in the range 0 to 1.0

```
-color 720 486 8 rgb 0.5 0.6 0.25
```

COLOR BARS

-colorbars <ntsc|pal> <75|100> <8|16|32> <rgb|rgba>

```
-colorbars pal 75 8 rgb
```

IMAGE IN

-read <image_file> image_file

is the same syntax as for other input, and for -write, namely

```
images.\$F4.sgi
```

IMAGE OUT

-write <file_type> <output_path>

Color Ops

Brightness

-brightness <brightness> [<offset>]

This only allows a single overall brightness (and offset) value, which is given in the range from 0.0 to 1.0

Channel Swap

-channelswap <channels> [<film|ntsc|pal>]

The channels are listed as input channels (r, g, b, etc.) and placed in the position where they are to be swapped to. For example:

```
-channelswap bgr
```

would put the incoming blue channel into the outgoing red channel, the incoming green into the outgoing green, and the incoming red into the outgoing blue.

Channels can be reused, so that this is allowed

```
-channelswap rrr
```

which would set all channels to the incoming red values.

The full list of valid channels is

`r`	red
`g`	green
`b`	blue
`a`	alpha
`l`	luminance
`0 (zero)`	black
`z`	white

Channels can also be listed by number, so 1 == red, 2 == blue, etc., which allows channels 5 through 9 (assuming they exist) to be specified.

```
-channelswap 50z
```

sets the red channel to the fifth incoming channel, sets the green channel to black, and the blue channel to white.

Contrast

-contrast <contrast> [<pivot>]

Gamma

-gamma <gamma>

INVERT

-invert

MONOCHROME

-monochrome [<amount> [<film|ntsc|pal>]]

The simplest use of this would be

```
-mono
```

If the type of monochrome calculation is not given, the default is "film," which is just an averaging of the 3 color channels. As in RAYZ, an "amount" of 0 means that the output is equal to the input. The default "amount" is 1.0.

VIDEO SAFE

-videosafe <film|ntsc|pal> <intensity|saturation>

XFORMS

FLIP FLOP

-flipflop [<rotate>] [<horizontal>] [<vertical>]

With no parameters set, this operation does nothing to the input image.

RESIZE

-resize [fit] <width>[f|p] <height>[f|p]

The optional "fit" argument will constrain the amount of scaling to be the same in both directions, using whichever is most. This is useful for doing letterboxing, for example.

To letterbox input into a PAL image size, the command would be

```
-resize fit 720p 576p
```

TRANSFORM

-rotate <angle> [<pivot_x>[f|p] <pivot_y>[f|p]]

Angles are given in degrees.

-scale <width>[f|p] <height>[f|p] [<pivot_x>[f|p] <pivot_y>[f|p]]

Scale is different from Resize in that Resize changes the actual size of the resulting image. Scale will position a smaller version of the input within a black image of the original size. If you scale up, the output will be the same size, and the parts which extend outside the incoming borders will be clipped out.

-translate <offset_x>[f|p] <offset_y>[f|p]

COMPS

For all these, the b_input_image_file must be a valid file or files named on the spot, that is, something like

```
-add imageB.sgi
```

You cannot nest operations to form a virtual b image; the following will generate an error:

```
-add -colorbars ntsc 75 8 rgb
```

The image generated just before the comp operation is considered to be the a_input_image_file. So A over B is done as

```
% rayz imageA.sgi -over imageB.sgi -write sgi out.sgi
```

-add <b_input_image_file>

-atop <b_input_image_file>

-difference <b_input_image_file>

-dissolve <amount> <b_input_image_file>

-inside <b_input_image_file>

-multiply <b_input_image_file>

-outside <b_input_image_file>

-over <b_input_image_file>

-subtract <b_input_image_file>

-under <b_input_image_file>

FILTERS

-blur <size>[f|p]

-blurxy <xsize>[f|p] <ysize>[f|p]

-unsharpmask <mask_amount> <kernel_size>[f|p]

-xpresso <"expression">

Note that the quotes are required in order to get the expression to RAYZ intact. So for example, to construct a vector map to use for doing radial blur, you would do this

```
-xpresso "[0,x,y,0] = sin(angle); [0,x,y,1] =
cos(angle);"
```

Note also that Xpresso almost always wants to have float input to work properly, so you probably want to precede it with

```
-bitdepth 32
```

CONVERSION

If you are reading in Cineon images, there is currently no way to avoid converting them to 16 bit linear using the default parameters. This may not be what you want. The way to work around this is to immediately convert back to log, and then reconvert using the parameters you want.

Since these operations concatenate, there will be no loss of precision in working this way.

As an example, say you want to specify a different log white point, say 983, for converting a group of Cineon images. The command for this would be

```
% rayz in.\$F4.cin -lintolog -logtolin 16 95 983
65535 0.6 1.7
```

-bitdepth <8|16|32> [<black_in>[f|p] <black_out>[f|p] <white_in>[f|p] <white_out>[f|p]]

-lintolog [<ref_black> <ref_white> <lin_white>[f|p] <film_gamma> <display_gamma>]

-logtolin <8|16|32> [<ref_black> <ref_white> <lin_white>[f|p] <film_gamma> <display_gamma> [<soft_clip>]]

Generating RAYZ Files

Using the RAYZ interface is not the only way to create and edit a node network. You also have the option of writing a script that describes it. A RAYZ script can be loaded into the interface and edited like any other file, and any file created in the RAYZ interface can be edited as a script.

The ability to modify imagery interactively while viewing the result is, of course, indispensable, and knowledge of scripting is unnecessary to build a shot in the RAYZ interface. But scripting is ideal for automating certain types of tasks, such as generating pre-comp files programmatically which can then be opened in the interface for further editing and refinement.

For example, you could write a script that imports all the image elements available for a shot and generates any other source nodes that might be needed. Then the scripted file could be opened in the RAYZ interface with all of the imagery pre-loaded into Image In nodes and all of the import options set appropriately. The script could even include a basic network of nodes to which the imported imagery would be connected: the script could specify that bluescreen elements be connected to Ultimatte nodes before sending the image data downstream to a composite node, and so on.

Scripting can save a lot of time that would be spent on repetitive setup tasks as well as ensuring that RAYZ files point to the correct image directories or use predetermined settings for certain operations.

Scripting Languages

A RAYZ script can be written using any scripting language you prefer: perl, tcl, python, or csh, for example. This means that you do not have to learn a whole new scripting language with its own set of conditionals, looping, and so forth. You only have to become familiar with the basic structure used in a RAYZ file to describe node operations and connections. Another advantage of this system is that interactive and script-generated RAYZ files always remain compatible, which means that a script can always be opened and edited in the interface, and vice versa.

THE STRUCTURE OF A RAYZ FILE

A RAYZ file contains the following categories of information:

- **version** (of RAYZ) used to generate the file
- **nodes:** all nodes in the network, and their parameter settings and connections
- **node GUI:** how the nodes are laid out in the worksheet (*x,y* positions), current selection, current time, etc.
- **preferences:** all of the file-specific project settings
- **windows:** number, size, and layout of all windows

All of the information listed above is optional except for the node information. If the layout and project preferences are not specified, the RAYZ defaults are used. (Strictly speaking, even the node information is optional, in that you can load an empty file into RAYZ.)

The best way to find out how RAYZ files are structured is to open one in a text editor and examine it, however, an overview of basic node syntax is provided below.

NODE SYNTAX

This is the basic syntax used to describe the node information in a RAYZ file:

```
nodename = nodetype(input_node1, input_node2, …)
{
    parametername1: value;
    parametername2: value;
    …;
}
```

The parameter specifications are optional; at minimum, you could use the following syntax to create a node with no input connections and all parameters set to default values:

```
nodename = nodetype ( );
```

All statements end with a semicolon, and groups of statements are contained in braces (curly brackets).

Take the simple example of a brightness operation on color bars. The following would create a Color Bars node named “cbars” with its output connected to the primary input of a Brightness node named “bright1”:

```
cbars = colorbars( );
```

```
bright1 = brightness(cbars);
```

You do not have to list parameters unless you want to change their default values. To specify a change to the brightness (the default values for the Brightness node do not change the output), specify the parameter and the value:

```
bright1 = brightness(cbars)
 {
 brightness: 0.5;
 }
```

The Brightness parameter is actually the master value for a channel group. To change the brightness of all channels except the Red channel, you could specify a change to the master brightness value and then offset it in the Red channel as follows:

```
bright1 = brightness(cbars)
 {
 brightness: 0.5;
 brightness.red: 0.5;
 }
```

The above example would bring the brightness of the Red channel back up to 1.0, because individual channel parameter values are always stored as deltas of the master value, as explained in "Channel Groups" in chapter 7, p. 93.

The same effect could also be achieved by specifying a decrease in the Green and Blue channel parameter values, without changing the default master value:

```
bright1 = brightness(cbars)
 {
 brightness.green: -0.5;
 brightness.blue: -0.5;
 }
```

For nodes that accept multiple inputs, connections are made in the order listed. In the following example, the output of the Color Bars node would be connected to the primary input of the Brightness node, and the output of a Gradient node named "grad1" would be connected to the optional mask input:

```
bright1 = brightness(cbars, grad1);
```

NODE NAMES

The names you should use to identify node types in RAYZ 2.2 script files are the same as the default name given to a new node of that type, minus the incremental number at the end.

For example, when you create a new Multi-comp node in the RAYZ GUI, it is given the name "multicomp1" by default. To designate a Multi-comp node in a script, therefore, you would use "multicomp" (all lowercase, no spaces, no numeral at the end).

Parameter Names

For node parameters, use the same name that is used to identify them in the Keypoint Viewer of the Curve Editor (see also "Editing Keypoints Numerically" in chapter 8, p. 111). If the parameter cannot be animated and is therefore not displayed in the Keypoint Viewer, such as the frame range parameters in the source nodes, save a RAYZ file with the same type of node and open it in a text editor.

The parameter most commonly used in writing RAYZ scripts is probably the file path in Image In:

```
nodename = imagein()
   {
   full.file_info.file_id: "/directory_path/filename.$Fn.ext";
   }
```

To specify proxies:

```
nodename = imagein()
   {
   full.file_info.file_id: "/directory_path/filename.$Fn.ext";
   medium.file_info.file_id: "/directory_path/filename.$Fn.ext";
   low.file_info.file_id: "/directory_path/filename.$Fn.ext";
   }
```

Script Errors

When RAYZ finds an error in a script, it will open the file if possible and generate an error message. If you misspell the name of a node type, for example, the error dialog will tell you that RAYZ was unable to create the node, but the file will still be opened with the other nodes specified in the script.

If you specify an input to a node that does not accept an input, such as Color Bars, RAYZ will give you an error message about the connection but still create the Color Bars node.

If you give multiple nodes the same name, RAYZ will append a number to them, incrementing it as necessary (cbars, cbars1, cbars2, etc.), without generating an error.

appendix E

PLUGINS

The RAYZ API is as open as we could make it, so that RAYZ can be used by plugin developers as a platform to implement all kinds of image processing tasks, not just for compositing. Plugins can be written to create

- nodes,
- overlays for the Image Viewer,
- LUTs and other image display transformations for the Image Viewer,
- file formats (to support in-house or other proprietary image file formats),
- file managers (to support in-house asset management systems), and
- preferences (read and write values of existing preferences, or create new prefs).

Plugin Nodes

Plugin nodes can be filters or source nodes (nodes that create images, such as the Gradient node). All of the RAYZ internal image operations are available to a plugin, and multiple operations can be chained together. For example, a plugin that will scale and rotate images can use the internal routines for these operations. This frees the plugin developer to concentrate on the unique aspects of the node design.

In addition to the RAYZ internal image operations, which do the actual work that modifies the image, all of the standard node interface elements, such as menus, buttons, and fields, are also available to use in plugin nodes.

Overlays

Overlays are optional graphics displays that overlay the node image in the Viewer. An overlay can be an interactive widget used to control node parameters. You can create your own overlays or assign existing RAYZ overlay widgets to plugin nodes.

In addition, custom overlay plugins can be assigned to existing RAYZ nodes to add more interactive functionality or graphical feedback to them.

Documentation for Plugin Developers

Extensive documentation of the RAYZ API is available in HTML format for plugin developers, along with examples. You can download the tar file from the Downloads section of the Silicon Grail website. Select the version that matches your platform, such as "sdk.linux.tar.gz."

LUTS

RAYZ converts image data as necessary for display in the Image Viewer. A display conversion method is selected automatically, based on the type of image being displayed, but you can override the default method and use your own LUT file, as described in “Display Conversion” (ch. 6, p. 72). This appendix explains how to create a LUT file that can be used in RAYZ.

Creating a Valid LUT File

A LUT file is a plain text document that lists pairs of values for each image channel in a simple table, with the input values in the first column and the output values in the second column.

The LUT file can include lookup tables for 1, 4, or 5 channels. If you use a 1-channel LUT file to display a multi-channel image, the same LUT file data is applied to all channels.

Each list of channel data in the LUT file must be separated by a line with two equals signs (==) on it, and nothing else.

A line can also be a comment, which begins with the // string (double divide char).

Within the section for each channel's data, the lines consist of two values:

<float input value> <8 bit output value>

This allows a completely general lookup table (that is, the input can be 8-bit, 16-bit, or float). The input value is converted to float, and then looked up. If there is no value specified, it uses the next lowest one, rather than interpolate a new one. This way you only need to specify 256 possible outputs, so you just specify a float value for each possible 8 bit value, and you are done.

Notice that you can have any number of values for a given channel (although having a single value is pretty useless, as it sets every input value to that single value).

Example LUT File

```
// This is a test LUT
0.0     0
0.1     25
0.2     50
0.3     75
0.4     100
0.5     125
0.6     150
0.7     175
0.8     200
0.9     225
1.0     250
==
0.0     0
0.5     100
1.0     200
==
0.0     100
0.25    200
0.5     225
1.0     250
==
0.0     0
0.25    25
0.50    50
0.75    75
1.0     100
==
```

INDEX

C

D

E

F

N

O

Q

R

S

X

Y

Z