Chalice User Guide

for Chalice 2.0

December 1999

© 1996–99 by Silicon Grail Corporation. All rights reserved.

Silicon Grail Chalice Version 2.0 User Guide for Silicon Graphics, Linux–based, and Windows NT–based computers.

Silicon Graphics is a registered trademark and IRIX is a trademark of Silicon Graphics, Inc. UNIX is a registered trademark in the United States and other countries, licensed exclusively through X/Open Company, Ltd. Windows NT is a registered trademark of Microsoft Corporation.

Cineon is a trademark of the Eastman Kodak Company. Photoshop and PostScript are trademarks of Adobe Systems Incorporated. QuickTime is a registered trademark of Apple Computer Inc. Substance is a trademark and Ultimatte is a registered trademark of the Ultimatte Corporation.

Acrobat® Reader © 1987–1996 Adobe Systems Incorporated. All rights reserved. Adobe and Acrobat are trademarks of Adobe Systems Incorporated which may be registered in certain jurisdictions.

All rights to all other trademarks and registered trademarks are the property of their respective owners.

The information in this manual is subject to change without notice and Silicon Grail Corporation assumes no responsibility or liability for any errors that it may contain.

This document was created at Silicon Grail Corporation, 710 Seward Street, Hollywood, CA 90038, USA.

CONTENTS

GETTING STARTED

1 Introduction

What Is Chalice? 1.1
Learning Chalice 1.1
Product Support for Chalice 1.2
How to Contact Silicon Grail 1.3

2 What's New in Chalice?

3 Setup

System Requirements 3.1
Installation 3.2
Optimizing Performance 3.6

4 Key Concepts

Dataflow Mechanics of the Chalice System 4.2
Image Representation in Chalice 4.6
How Chalice Facilitates Teamwork 4.9
Memory Usage in Chalice 4.10

WORKING IN CHALICE

5 Building Node Networks

Launching Chalice 5.2
Overview of the Chalice Screen Layout 5.3
Working with Grail Files 5.6
Creating Node Networks 5.10
Overview of the Node Layout 5.15
Previewing Node Output 5.18

6 Node Panels and Parameters

Overview of the Node Panel 6.2
  Parameter Tabs 6.3
  Info Tab 6.4
Setting Node Parameters 6.7

Using Tuners 6.7
Using Data Entry Fields 6.8
Animating Parameter Data 6.9
Enabling Expressions 6.9
Creating Key Frames 6.10
Using Preset Parameter Values 6.11
Setting Global Variables 6.12

7 Managing Large Networks

Shortcuts 7.2
Finding Nodes Fast 7.5
Grouping Nodes 7.7
Creating Custom Networks 7.11

8 Monitors and Flipbooks

Using Monitors 8.2
Opening a Monitor 8.2
Updating the Monitor Display 8.3
Setting Monitor Display Options 8.3
Using the Display Buffers 8.4
Performing A/B Wipes 8.5
Using ROI to Update the Monitor 8.6
Using the Monitor Toolbar 8.7
Using Monitor Presets and Display LUTs 8.9
Using the Monitor Popup Menu 8.10
Using the Utility Monitors 8.11
Using Flipbooks 8.14
Creating a Flipbook 8.14
Using the Flipbook Controls 8.15
Using the Flipbook Popup Menu 8.16

9 Space-Time Editor (STED)

10 Main Menu Bar

File Menu 10.2
Edit Menu 10.4
Group Menu 10.6
Layout Menu 10.7
Preferences Menu 10.9
Admin Menu 10.10
Help Menu 10.12

11 Setting Preferences

Monitor Preferences 11.2
Flipbook Preferences 11.4
Node Preferences 11.6
Proxy Preferences 11.7
Data Format Preferences 11.8
Autosave Preferences 11.10

Node Reference

12 I/O Nodes

Image File Formats Supported by Chalice 12.2
Using Proxies in Chalice 12.5
Color Node 12.9
Disk Input Node 12.12
Disk Output Node 12.23

13 Tool Nodes

Channel Swap Node 13.2
Comment Node 13.4
Exec Node 13.5
Pixel Meter Node 13.9
Sequence Node 13.12
Switch Node 13.17
Text Node 13.18
Time Stretch Node 13.24
Track Node 13.25
UNIX Node 13.31

14 Conversion Nodes

Bit Expand Node 14.2
Bit Reduce Node 14.7
Cineon Extract Node 14.17
Cineon Insert Node 14.19
Linear Remap Node 14.21
Premultiply and Unpremultiply Nodes 14.23

15 Composite Nodes

Mcomp Node 15.4
Zcomp Node 15.6

Over, Atop, Inside, and Outside Nodes 15.9
Diff, Minus, Multiply, and Plus Nodes 15.11
Dissolve Node 15.13
MinMax Node 15.14
Ultimatte AE (AdvantEdge) Node 15.15

16 Matte Nodes

Chromakey Node 16.2
Rotospline Node 16.5
The Ultimatte Nodes 16.13
  Ultimatte CSC (Classic Screen Correction) Node 16.18
  Ultimatte GK (Grain Killer) Node 16.22
  Ultimatte RSC (Roto Screen Correction) Node 16.26
  Ultimatte PFG (Processed Foreground) Node 16.36
  Ultimatte CC (Color Control) Node 16.44

17 Movement Nodes

Corner Pin Node 17.2
Crop Node 17.6
Rotate Flip Node 17.8
Scale Node 17.9
Transform Node 17.12
Match Move and Stabilize Nodes 17.17

18 Filter Nodes

About the Color Filters 18.3
  Color Correct Node 18.4
  Brightness Node 18.14
  Contrast Node 18.16
  Gamma Node 18.18
  Hue Adjust Node 18.19
  Inverse Node 18.21
  Monochrome Node 18.22
About the Area Filters 18.23
  Blur Node 18.24
  Convolve Node 18.26
  Degrain Node 18.31
  Edge Node 18.34
  Erode/Dilate Node 18.36
  Grain Node 18.37

Noise Node 18.40
Rank Node 18.41
Sharpen Node 18.45
Unsharp Mask Node 18.48

19 Effects Nodes

Bump Node 19.2
Surface Normal Node 19.3
Emboss Node 19.4
Vector Warp Node 19.6
Posterize Node 19.7
Time Convolve Node 19.8
Vector Blur Node 19.11
Xpresso Node 19.15

20 Video Nodes

3:2 Pulldown Node 20.2
3:2 Pushup Node 20.5
Deinterlace Node 20.6
Interlace Node 20.8
Merge Node 20.9
Split Node 20.10
Video Safe Node 20.12

21 Plug-in Nodes

Appendixes

chal A.1

Chalice Expression Language B.1

Softclip Conversion C.1

Index

1 INTRODUCTION

What Is Chalice?

Chalice is a powerful image composition and adjustment software tool designed for the film industry professional to perform multilayer compositing, tracking, color correction, and other image processing tasks required by digital special effects artists.

The Chalice interface is based on the concept of a network of nodes, wherein each node performs a specific function. By creating and connecting these nodes, you can build complex networks to create sophisticated digital effects. Monitors can be opened at any point in a network to allow you to view every change in your imagery at each step in the process.

This "dataflow" paradigm lends itself particularly well to the creation of special effects, for it enables a flexible, extensible architecture that is at once both powerful and yet easy to comprehend. Basic image editing tasks can be mastered within minutes. Yet experienced users can create plug-in nodes, utilize UNIX filters, and craft precise mathematical expressions to control image processing.

Chalice can work in 8 bits per channel, 16 bits per channel, or floating point (32 bits) per channel, even within the same dataflow. And Chalice supports all the basic image formats, including comprehensive conversion options to and from the Kodak Cineon 10-bit log data format. Chalice can accommodate very high resolution images and is able to perform image caching and to process images in multiple slices. You can specify disk and RAM cache values as well as the number of slices to facilitate working within the memory constraints of your system.

In addition to the interactive interface, Chalice features a command line mode called "chal," which enables you to batch process grail scripts automatically. (Chal and its options are described in appendix A of this manual.)

Learning Chalice

Tutorials and example files are provided to help new users get started using Chalice and to help experienced users become more productive.

Chalice Tutorials

Several hands-on tutorials are available to learn the basics of working in Chalice, from an overview of the interface and dataflow paradigm to task-specific procedurals covering such topics as matte adjustment and tracking.

> **Note:**
>
> If you are a new Chalice user, we recommend starting with these tutorials. Even if you are familiar with Chalice, these tutorials still provide useful tips and shortcuts.

The tutorials are provided in html format and as Adobe Acrobat pdf files, which are optimized for printing hardcopy. Included with the tutorials are

- sample images you can import into Chalice as you follow the instructions in the tutorials to create your own networks, and

- example grail files of the completed network for each tutorial, which you can open in Chalice and review.

The Chalice tutorials are available on the Chalice distribution CD (in the Tutorials directory) and in the Support section of the Silicon Grail website (www.sgrail.com).

Annotated Sample Networks

The Custom menu (located at the right end of the Chalice node menu) includes annotated example grail files that illustrate various composite operations and related tasks, including image stabilization and match moves.

To review these files, select one from the Custom menu and place it in the workspace. Click and hold the Info button on each node in the sample network, starting with the "Read First" Comment node, if one is included. This will enable you to read explanatory notes in the Info popup boxes.

How to Use this Manual

This manual is not a compositing textbook, and readers are assumed to be experienced digital compositors. However, the background of compositing professionals working in today's effects facilities can vary greatly. Your previous experience may be in traditional optical composition for film, in video editing, in computer graphics, and/or in 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/linear conversion for the Cineon film format, while a compositor with a C programming background may have no trouble writing complex expressions.

However, we recommend that even if you are expert in a particular area you at least skim the relevant explanations in this manual. For example, even if you are familiar with Cineon conversion issues, it will be profitable to find out how Chalice handles this conversion, what conversion options are available, and exactly what we mean by the various terms we use.

New users of Chalice should peruse the chapters in the first half of the manual, which cover basic concepts relating to building nodes into a network that defines a shot and using the tools common to most operations. This knowledge is assumed in the node reference chapters, which experienced users are most likely to consult for specific information about the use of individual nodes.

Even experienced Chalice users may profit from at least skimming the overview chapters, however, which provide shortcuts and tips to make the job easier by preventing the repetition of mundane tasks.

Product Support for Chalice

We have designed support systems for Chalice with the same philosophy that guides the development of Chalice itself—to be powerful, flexible, easy to use, and responsive. Documentation, problem reporting capabilities and solutions, expert tips, and other support features are all available in the most convenient and up-to-date form possible. Information and support tools are included with Chalice and on the Silicon Grail website.

Chalice Help System

Chalice includes a comprehensive, hyperlinked, context-sensitive online help system, which is available for each node via the Node Help buttons on each node panel as well as by accessing the Online Manual item under the Help Menu.

This manual is available in electronic form (html and pdf) on the Chalice CD-ROM, as well as in printed form. The Support section of the Silicon Grail website also contains the latest version of this manual in html format.

Problem Reports

We not only welcome problem reports from Chalice users, but make reporting easy by incorporating a problem-reporting email feature into Chalice, which is described in the Help Menu section of chapter 10, "Main Menu Bar."

Silicon Grail Website

Keep current with the latest news and information about Chalice by visiting the Silicon Grail website. In addition to publishing the latest documentation, the Support section of our website provides a problem report form which can be used to report both software and documentation problems. For example, if you think a description in this manual is unclear or incomplete, use the form to let us know so that we can fix it.

Chalice Users Email List

If you have questions or comments about Chalice, you can now post an email message to the Chalice Users list and receive feedback from other users as well as from our programming, support, and documentation staff.

To subscribe:
Send email to majordomo@sgrail.com with the words "subscribe chalice-users" in the body.

To receive more detailed instructions:
Send email to chalice-users-request@sgrail.com, which will email instructions back to you.

To post a message:
Send to chalice-users@sgrail.com.

How to Contact Silicon Grail

Silicon Grail Corporation
710 Seward Street
Hollywood, CA 90038, USA
PHONE: 323/871-9100
FAX: 323/871-9199
WEBSITE: http://www.sgrail.com
EMAIL: info@sgrail.com (for general information)
sales@sgrail.com (for sales information)
key@sgrail.com (to request a license key)
support@sgrail.com (for customer support)
problems@sgrail.com (to submit a problem report)
gimp-bugs@sgrail.com (to report a problem with the version of the GIMP paint program distributed with Chalice)

2 WHAT'S NEW IN CHALICE?

NEW IN VERSION 2.0

Version 2.0 of Chalice has substantial performance improvements to computation speed and memory use, which will make it faster for interactive use, especially with large images.

The user interface has been revamped to be clearer and more consistent. For example, the node menus have been reorganized into new categories, and in some cases renamed. Many parameters have been relabeled to make them more obvious.

New features include a file proxy system, integration of motion blur into transformation operations, expanded grain-related tools, and a new time-stretch node. New stand-alone tools included with Chalice are 16-bit paint and a pair of image file utilities.

Note:

Refer also to the release notes section of the chalice readme document for the latest and most detailed changes list.

Performance Enhancements

Improvements in computation time and disk access will both speed up performance. Chalice 2.0 features direct I/O and a new caching system, which reduces memory swapping. In addition, the caching algorithms are smarter about what to save and what to give up. Monitors can now slice images, which also reduces memory consumption. See also "Memory Usage in Chalice" in chapter 4, "Key Concepts."

Platform Support for Linux

Chal2.0 will render on Linux running on both Intel and Alpha platforms.

New Stand-alone Tools

Chalice 2.0 comes with a ready-to-run version of **GIMP, the Open Source paint software**. Silicon Grail did not write GIMP, and does not support it. To answer the need for a 16-bit paint package that can work with Chalice, however, we have packaged it with the Chalice distribution. The program can be found in /usr/grail/gimp.

GIMP is not automatically installed with Chalice; you must tell the installation program to install it. Refer to the installation instructions in chapter 3, "Setup." For more information on the GIMP program itself, see the separate readme document included with GIMP.

Chalice 2.0 also comes with two file utilities: A new tool called **fconv** (**frame convert**) will convert and renumber ranges of frames. For usage details, type `fconv` at the prompt.

And a new tool called **fcp** (**frame copy**) will read in an image of one format and copy it to a different format. It uses the image suffixes to determine format, unless overridden. Type `fcp` to get usage details.

New Features

See also the "List of Changes by Node" at the end of this chapter for other new features specific to a node operation (such as motion blur incorporated into Transform).

Proxies

The Disk Input node has a new tab which allows the user to create and use low resolution proxy files for the full resolution input files. Proxies are helpful in two types of situations. One is when the full resolution files are not available, as with CG elements that are tested at lower resolutions than the final shot. These low-res tests can be used to start setting up the shot. The other situation is when all the elements are available, but the user wants to speed up interactivity by working with smaller versions. See the new manual section "Using Proxies in Chalice" in chapter 12 for more detail.

Monitors

Monitor and flipbook options have been refined and improved. For example, the A/B Wipe function no longer asks for confirmation before cooking the B image, and a new hot key (F9) has been added to snap an image back to the center of the monitor display.

And monitors can now use slicing: large images can be processed in strips, which reduces memory consumption and speeds throughput. See also chapter 8, "Monitors and Flipbooks."

Chal Options

The existing -f option can now use the name of the Exec input instead of its number, if desired. The new chal -t option will display the average time for computing frames, and chal -j will execute the file sequentially (rather than interleaved, which is the chal -i option). For a list of all options, refer to appendix A, "Chal."

STED

In the Chalice Space-Time Editor, channels can now be saved as raw ASCII data, and raw channel values can be imported into existing channels. See also chapter 9, "Space-Time Editor (STED)."

Changes to the User Interface

Both the Chalice main menu bar and node menu bar have been reorganized and menus have been added or renamed. See also chapter 10, "Main Menu Bar."

- Preferences and Layout settings are better organized in the Chalice main menus.
- New categories have been added to the node menu bar for easier access to nodes.
- It is now easy to simultaneously change the way multiple nodes appear in the workspace (size, icon image display) using the Layout Menu.
- License tracking reports can now be generated by picking the appropriate command from the new Admin Menu.

Other changes include improvements to error reporting—now only the source of the error turns red, instead of all the nodes in the error path. Also, the nodes have been redesigned to make the error flag easier to spot in a crowded layout.

A large, consistent resolution menu is now available throughout Chalice. And UI colors can be customized by each user in a resources file.

Name Changes to Nodes

Some nodes have been renamed in 2.0 and in some cases two nodes have been combined into one or a single node has been split into two separate nodes:

Apply Track has been split into **Match Move** and **Stabilize**.
Blur and *Low Pass* have been combined into a single **Blur** node.
Combine is now called **Channel Swap**.

Field is now called **Deinterlace**.
Linear Xlate is now called **Linear Remap**.
Meters is now called **Pixel Meter**.
Xform is now called **Transform**.
Cineon to Linear is now called **Cineon Insert**.
Cineon from Linear is now called **Cineon Extract**.

New Nodes

Two new nodes have been added to Chalice 2.0: the **Degrain Node**, which provides sophisticated grain removal using frequency analysis, and the **Time Stretch Node**, which changes the length of a clip.

List of Changes by Node

Color Node: Ramps now dither by default to avoid banding in 8-bit or 16-bit gradients. A popup menu of possible resolutions has been added as an alternative to keying in XY values.

Color Correct Node: You can now match levels of two separate images in the Contrast Stretch layer automatically by picking representative pixels in the work monitor.

Corner Pin Node: Clearer labels in the node panel.

Crop Node: Changed from specifying two corners to specifying a corner and a size. Added icons for ease of use.

Disk Input Node: New file proxy section allows you to create or point to existing proxy images. Sequencing has been expanded. The default Cineon conversion has been changed from "insert" to "remap." For clarity, the format option tabs are now disabled unless they are relevant to the input imagery. Now supports HD-resolution (1920 x 1080) YUV files.

Disk Output Node: File specification has been split into separate tabs for final output and temp output. For clarity, the format option tabs are now disabled unless they are relevant to the selected output format. Now supports HD-resolution YUV files.

Exec Node: Clearer node panel. Can execute finals, or low resolution temps.

Grain Node: Library of presets for common film stocks added. Node panel updated.

Scale Node: More filtering options. Rewritten to be considerably faster.

Sharpen Node: New filtering options added.

Transform Node (formerly "Xform"): Much faster. New motion blur option. More filtering options. Allows specification of pivot point, which applies to both rotation and scaling operations. Allows specification of transform order.

Xpresso Node: Now works from bottom to top (i.e., 0,0 is in the lower left corner of image), to be consistent with the rest of Chalice. Orientation in Y can be switched, however, for backward compatibility.

Deleted Nodes

Two nodes have been deleted from Chalice 2.0. The Black & Rolloff In and Black & Rolloff Out conversion nodes were removed because they became redundant. (The Bit Reduce and Bit Expand nodes provide the option of using the black-and-rolloff operation.)

3 SETUP

The Setup chapter provides information about installing and optimizing Chalice, including

System Requirements ..... p. 3.1
Installation ..... p. 3.2
Optimizing Performance ..... p. 3.6

Note:

Be sure to check the Chalice readme document for the most up-to-date information on setup. For custom installations, contact Silicon Grail Customer Support.

System Requirements

Chalice and its associated batch render program, chal, will run well on

- **IRIX:** any Silicon Graphics workstation with at least an R4000 CPU running the IRIX 6.2 (or later) operating system. Chalice requires a minimum of 64 MB of RAM, although 128 MB or more is recommended for optimal performance. A 24-bit graphics accelerator and a large, 19- to 21-inch color monitor are highly recommended.

- **NT:** any workstation with Intel PentiumPro/Pentium II/Pentium III or DEC Alpha processors running Windows NT 4.X. A minimum of 64 MB of RAM is required, although 128 MB or more is recommended. Chalice also requires an OpenGL-capable video card with at least 8 MB of video memory, although 16 MB is recommended.

In addition, chal runs under **Linux** on both Intel and Alpha workstations.

The Chalice application files require approximately 45 MB of hard disk space. In addition, Chalice requires a substantial amount of temporary disk space while the application is running. Refer to "Optimizing Performance" at the end of this chapter to learn how to get the most out of Chalice on your workstation.

Note for IRIX Users:

We strongly recommend that you increase the number of shared memory segments from the default as shipped by SGI. This procedure is described in "Enhancing Operation of Chalice on IRIX Systems."

INSTALLATION

This document explains how to install Chalice on an IRIX or Windows NT workstation from the Chalice CD-ROM. Follow the procedure that is appropriate for your system configuration:

IRIX 6.X Installation (Inst) ..... p. 3.2
Windows NT 4.X Installation ..... p. 3.4

Once you have installed the software, you will need to contact Silicon Grail to receive a numerical software key that will enable you to inform Chalice of the number of floating licenses to which you are entitled based on your purchase agreement. (Licensing instructions are included herein for each type of installation.)

IRIX 6.X INSTALLATION (INST)

1. **Log in to your workstation as root to become SuperUser.**
2. **Insert the CD-ROM into your local CD-ROM drive.**
3. **Select Software Manager from the Desk1 (toolbox) System Menu.**
4. **In the Software Manager window, enter the following in the Available Software text field:**

   ```
   /CDROM/dist
   ```

5. **Push the Custom Installation button at the top of the window.**
   The Software Manager will scan the CD for available software to install. “Chalice 2.0” and “Silicon Grail License Manager” should be displayed in the Software Inventory window. In addition, the “GIMP” open source paint program distributed with Chalice should appear here.
6. **Click the gray checkboxes for “Chalice 2.0” and “Silicon Grail License Manager” under the Install column in the Software Inventory window.**
   A red checkmark will appear in the box and the Start button (located below the Software Inventory window) will highlight in blue.

   Note:

   Optionally, you may check the box for “GIMP” installation also.
7. **Press the Start button to begin installation.**
   The Software Manager will install Chalice on your hard disk.
8. **After completion of the installation, select Exit from the File menu of the Software Manager.**

   Note:

   To use the Chalice Online Help System, you will need to set the environment variable CHALICE_HELP_BROWSER to the path of your web browser. Although this step is not strictly necessary to install and run Chalice, you will probably want to enable Online Help soon after you complete the installation.
9. **Follow the instructions below to obtain a license key.**

10. **Follow the instructions for using "chalicetune" in "Enhancing Operation of Chalice on IRIX Systems."**

    This step is optional, but highly recommended.

Keying the Software on IRIX Systems

Follow these instructions to obtain a key to run Chalice on an IRIX workstation:

1. **Log in to your workstation as root to become SuperUser.**
2. **Type the following command to update the file system:**

   ```
   rehash
   ```

3. **Run the Chalice license manager software using the following command:**

   ```
   ./makelic
   ```

   The system will display your server code with a prompt similar to the following:

   ```
   Code for host "[your hostname]" is: 001 9973 8031 9842 0814 8

   Please enter key for server "[your hostname]":
   ```

4. **Record your host code and exit the license manager software by pressing the Control and C keys.**
5. **Contact Silicon Grail Corporation to communicate your server code.**

   Send the host codes for your host to the attention of Technical Support at Silicon Grail by fax at (323) 871-9199, by email at key@sgrail.com, or by phone at (323) 871-9100.

   Silicon Grail will respond by building a key and communicating it to you. The key will be a long sequence of numbers that will inform the Chalice software of the quantity of licenses to which you are entitled.

6. **Once you have received your key, be sure you are still logged in as root and restart the Chalice license manager with the following command:**

   ```
   ./makelic
   ```

7. **Enter the software key numbers at the prompt:**

   ```
   Please enter key for server "[your hostname]":
   ```

   The software will respond with the status of the key installation. If the installation was successful, proceed to step 8. If you entered the key incorrectly, the system will notify you and ask you to enter the key again.

8. **Log off the system.**
9. **Log in using your usual login and password.**
10. **To verify that Chalice has been installed correctly, launch the program by keying in the following command:**

    ```
    chalice
    ```

Windows NT 4.x Installation

Follow the procedure below to install Chalice on your Intel or DEC Alpha Windows NT workstation:

1. **Log in as Administrator or as a user with administrator privileges.**
2. **Insert the CD-ROM into your local CD-ROM drive.**
3. **Use desktop icons to open the CD-ROM drive content window.**
4. **Navigate to (assuming CD-ROM drive letter D:)**
   D:\WindowsNT
5. **Double-click on the Setup icon ("setup.exe") to launch the Chalice installation and License Manager programs.**

   Note:

   There is no option to install the "GIMP" open source paint program that is distributed with Chalice, as it is not yet available on the NT platform.
6. **Follow the instructions given by the installation program dialogs.**
7. **Restart the computer as instructed by the installation program.**

   Note:

   In order to allow other users to run Chalice, they must have the environment variable HOME set to some existing directory path. This can be done in the Windows NT System Control Panel.
8. **Follow the instructions below to obtain a license key.**

Using the NT License Manager

1. **Navigate to the Control Panel (Start > Settings > Control Panel) and double-click the SG_LM icon in the Control Panel window.**
   This will open the panel in which you launch the Silicon Grail License Manager.
2. **In the SG_LM panel, press the Launch button, and then press OK to close the panel.**
3. **Navigate to the Chalice licensing directory on your hard drive:**
   C:\Program Files\Silicon Grail\SG_LM
4. **Execute "makelic.exe" by double-clicking on the file.**
   This will bring up the Chalice License Manager user interface to generate a host code for your machine.
5. **Record your host code and quit the License Manager.**
6. **Contact Silicon Grail Corporation to communicate your server code.**
   Send the host codes for your host to the attention of Technical Support at Silicon Grail by fax at (323) 871-9199, by email at key@sgrail.com, or by phone at (323) 871-9100.

   Silicon Grail will respond by building a key and communicating it to you. The key will be a long sequence of numbers that will inform the Chalice software of the quantity of licenses to which you are entitled.

7. **Once you have received your key, restart the Chalice License Manager.**
   Be sure that you are still logged in with administrator privileges.
8. **Follow the instructions given by the software to enter the key, and then quit the License Manager.**
9. **Log off the system and log in again using your usual login and password.**
10. **To verify that Chalice has been installed correctly, launch the program by double-clicking on the Chalice icon.**
    You can also launch the program by typing the word "chalice" in the Command Prompt window.

Note:

To use the Chalice Online Help System, you will need to set the environment variable CHALICE_HELP_BROWSER to the path of your web browser. Although this step is not strictly necessary to install and run Chalice, you will probably want to enable Online Help soon after you complete the installation.

OPTIMIZING PERFORMANCE

Chalice may require a substantial amount of temporary disk space while in operation. Each frame of film at 2K resolution requires 2 to 8 MB, depending on the file format. Cineon 10-bit 4K files require in excess of 75 MB per frame. In addition, performing multiple-layer composites can take several times these amounts. You will also require substantial hard disk space for your source imagery; this amount will be doubled by the imagery Chalice will generate during processing.

Therefore, we recommend that you set the environment variable "CHALICE_TMPDIR" to a file system path that has a substantial amount of disk space available, preferably *not* on the root partition. We suggest that you allocate as much hard disk space as you can afford. For more information, refer to the readme document on the Chalice CD-ROM.

In addition, we recommend that you review the following information, located elsewhere in this manual, to find out more about

- **memory usage** in Chalice, to enable you to optimize your cache settings: chapter 4, "Key Concepts."
- **preference settings**, to enable you to set the defaults for basic Chalice interface elements to fit your work habits and projects: chapter 11, "Setting Preferences."
- **file proxies**, which improve performance most when used knowledgeably: "Using Proxies in Chalice" in chapter 12, "I/O Nodes."

Chapter 7, "**Managing Large Networks**," may also be useful to compositors working on complex shots.

ENHANCING OPERATION OF CHALICE ON IRIX SYSTEMS

The default number of shared memory segments available to each user of IRIX workstations, as shipped from SGI, is 100. For optimal use of Chalice, this setting must be increased. The instructions below tell you how to run the "chalicetune" program to increase this setting, based on the number of processors in your system.

Warning:

This script modifies your kernel, and you will have to reboot for its changes to take effect. If you have a system administrator, you may want to have that person look at this script before you run the command. This script will set shmmni and sshmseg to $size.

1. **To run chalicetune, log in to your workstation as root to become SuperUser.**
2. **Type the following command to start the program:**

   ```
   chalicetune
   ```

   Chalicetune will warn you that it is about to change the value, and how much, and ask you to confirm that you want to continue (yes/no).
3. **Answer "yes" to have the chalicetune program modify your kernel as specified above.** If you answer "no," chalicetune will abort.
4. **Reboot your system for the change to take effect.**

4 KEY CONCEPTS

Virtually every aspect of Chalice has been designed to help film compositors work faster, more efficiently, with less duplication of effort, and produce the highest quality imagery. To take advantage of these features, however, requires some knowledge of how Chalice works.

The purpose of this chapter is to help users get the most out of Chalice by reviewing both the concepts behind the Chalice system and the way these concepts are implemented.

The Dataflow Paradigm

Chalice employs a procedural dataflow paradigm for effects creation. The procedural part means that you build a description of a shot, with feedback from monitor previews throughout, and only render out actual image frames when you are satisfied with the result. This is ideal for a process that almost invariably requires numerous changes—you change only the description, and only where the changes are needed.

The dataflow aspect refers to the way you build a shot. Each operation is represented as a node in a flowchart-like network of nodes. The connections among the nodes represent the flow of data through this network. The dataflow model offers the most flexibility because it is "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 tweaked—by the shot's creator or by another artist.

In this Chapter

The section on dataflow mechanics is a nuts-and-bolts description of how data flows through the system. This background can help you make sense of the system and use it more effectively. It covers the basics of processing, sequencing, frame navigation, and previewing effects in monitors, with examples of how to connect nodes in the most efficient order for faster processing.

Of course, no other advantages will matter in the long run unless the image quality is there. The section on Image Representation in Chalice discusses how image formats vary, and how Chalice imports and represents image data.

As important as working effectively within an individual grail file is taking advantage of the system's ability to manage data throughout a facility. The section on teamwork discusses the sharing concept, which helps eliminate duplication of effort wherever possible and ensure consistency of elements from shot to shot.

The final section of this chapter covers memory usage, and especially image caching, to help you get the best performance from Chalice.

Dataflow Mechanics of the Chalice System ..... p. 4.2
Image Representation in Chalice ..... p. 4.6
How Chalice Facilitates Teamwork ..... p. 4.9
Memory Usage in Chalice ..... p. 4.10

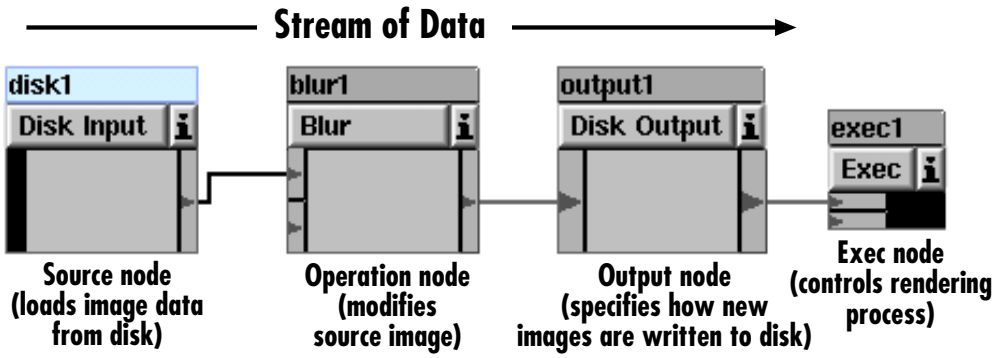
DATAFLOW MECHANICS OF THE CHALICE SYSTEM

As discussed elsewhere, Chalice employs a dataflow paradigm in which you create a description of a shot by building a flowchart consisting of nodes, in which each node represents a different operation. A dataflow network starts with one or more Disk Input nodes, which point to the source image files you will be modifying.

Whatever operations you perform on this imagery, finally you will want to render new image files of the completed shot to disk. In Chalice, you use a Disk Output node to specify how and where to write the image files, and connect it to an Exec node. A Chalice dataflow always terminates at an Exec node, the master control you use to instruct Chalice to begin processing.

Image Dataflow

You can think of the data as a stream, which flows left to right across the interface, from the source (Disk Input) nodes to other nodes downstream, being modified by the nodes you connect, in the order you connect them, until it reaches the end of the network. This left-to-right flow is considered to be the most natural for English speakers, who read and write using the same directional flow.



Sequence (Frame Number) Dataflow

Just as image data is passed through the network, the command to process each frame is also passed through the network, in the opposite direction. The function of the Exec node is to send a sequence of frame numbers, one at a time, upstream in the dataflow to which it is connected.

For instance, to process a sequence of frames 1 to 15, the general operation would be to first send the number "1" to the dataflow node that is directly connected to the Exec node (to the left). Each node passes the frame number upstream until it reaches the source, where the image data for that frame is read and delivered back downstream, being modified by each connected node until it reaches the Disk Output node and can be written to disk. The Exec node waits for the nodes upstream to finish computing before delivering the number "2," and so on.

This is important to know because the purpose of some nodes is not to modify the image but to modify the sequence—to remap the frame numbers that are passed through the network:

- Most nodes in Chalice will pass the frame number unchanged to the node connected on its left. Filters and composite nodes, for example, fall into this category.

- Another class of node, however, modifies the frame number received on the right and delivers a new number upstream. These include some of the video-type nodes. For example, the 3:2 Pulldown node creates five video frames from every four film frames and therefore changes the length of a sequence. This category also includes the node we will discuss next, the Sequence node.

Resequencing Frames

Imagine that the Exec node passes the number "5" to the dataflow upstream. The actual processing of data will not occur until a source node is reached. Once that is found, the node will start data processing and pass the data downstream as it is calculated. As each node completes its own processing, it notifies the node or nodes connected to it downstream of the data that it is presenting, until the last node in the chain connected to the Exec node prompts the Exec node to hand out the next frame number (assuming there are more numbers to process). When the Exec node has no more numbers to hand out, the processing is complete.

In the simple example given above, in which the Exec node sends the numbers 1-15, it could be the case that the source imagery really starts on frame 101 and goes to 115. Or it could be that only frame 101 is used repeatedly for 15 frames. Or the 15 frames could consist of 6 frames from one source node and 9 frames from another.

The Sequence node—and the Sequence tab in the Disk Input node—allow us to "shift" time so that when we are trying to process output frame 1, we can actually use input frame 101 if desired. In fact, the Sequence node enables non-matched frame numbers to be combined in any fashion.

Using the example of source frames 101 to 115, the simplest way to remap them is to use the Sequence tab directly in the Disk Input node. The parameter controls on this tab will also enable you to add holds, loops, and so forth.

For the example of frames 1–15 originating from two separate sources, both Disk Input nodes would be connected to a Sequence node. If the first Disk Input node contained frames 101 to 115, you could remap frames 101 to 106 to numbers 1 to 6. Likewise, whatever the frame numbers of the imagery in the second Disk Input node connected to the Sequence node, any specific nine frames in the input could be remapped to frames 7–15.

The Sequence node can be inserted anywhere in a network to resequence frames, and it accepts as many different inputs as necessary. This is why the frame number being passed upstream is relayed through each node. The Exec node does not deliver the number "1" directly to all the nodes; it just passes that number upstream, where it may be altered by a Sequence node or any other node of the same class.

Developing a Grail Script

A grail script refers to the network of nodes you build in a grail file. Let's assume that we wish to blur some frames and have set up the Exec node to process numbers 1 through 10 with an increment of 1. A Disk Input node points to 10 files, which are labeled "sharp.0001.rgb" through "sharp.0010.rgb." The Disk Output node points to a path and filename of "blurred." The Blur node is connected between the Disk Input and Disk Output. If we were to run this grail script by specifying the Execute command in the Exec node, we would create 10 files named "blurred.0001.rgb" through "blurred.0010.rgb."

Debugging a Grail Script

Now assume that we have hooked this up for the first time and would like to "debug the script"—that is, we would like to examine the result and make any necessary changes to the dataflow. For example, we could decide to make the image a little more or less blurry.

Say we want to see what frame 5 looks like. We could conceivably use the Exec node to write frame 5 to disk and then look at the frame to make sure that it met our requirements. However, another technique is actually used that enables us to debug an image without requiring that we write the image to disk to review it.

Node Monitor Previews

We can use the Local Node Menu on the Blur node to open a node monitor. This has an effect on the system that is very similar to when we executed the network in the Exec node. The node monitor acts like an Exec node in this case, instructing upstream nodes to send it data, except that it gets the frame number from the Frame Controller.

Once the source image data is accessed and passed downstream, perhaps being modified by other nodes along the way, it reaches the Blur node, which processes it (in this case, blurs it) and presents it at the resolution specified when the monitor was opened. To view the effect of the blur operation on any specific frame, the Frame Controller would be changed to that number.

The Frame Controller is used to specify a frame number for whichever node is currently selected. (This is important both for viewing imagery at a node and for setting and animating node parameter values, as discussed in chapter 6, "Node Panels and Parameters.")

Both a Monitor actuation and the "execute" command in the Exec node cause the system to calculate, or "cook," the image data through all nodes connected to it upstream in the network—the difference is how they get the number they will subsequently deliver to the upstream node(s) and, of course, whether the processed image data is merely displayed in a monitor or written to disk.

By merely looking at the Monitor of a node we have not output any data to disk. Image data is only written to disk by issuing the Execute command in the Exec node. Then it is written in the format and to the location specified in the Disk Output node, which is connected directly to the left of the Exec node.

Comparing Node Monitor Views

You can view an image at any stage of modification in the network by opening a monitor at a specific node. In fact, you can open multiple monitors to view imagery side-by-side for comparison. For example, you can examine the unmodified source image by opening a Disk Input monitor. To examine the same image cropped, assuming this was one of the operations in the network, you would open the Crop node monitor. To view the cropped, blurred image, you would open the Blur node monitor, and so on. To view another frame of the sequence at any stage, you would specify that frame number in the Frame Controller.

This principle applies no matter how large the node network becomes. You can always open a monitor at any node in the network to display any frame of the imagery at that stage of processing.

Nodes as Building Blocks

The flexibility of the system whereby each node represents a separate image processing operation means that you could just as easily crop an image after blurring it by connecting the

Crop node downstream of the Blur node. As it happens, for these two particular operations, the Crop-to-Blur order is preferable because it takes less processing time—only the pixels in the cropped image are blurred. If you discover this fact after creating a Blur-to-Crop connection, you can easily rearrange the nodes at any time, not matter how many more nodes are connected upstream or downstream in the flow.

For More Information

We recommend that you read chapter 5, "Building Chalice Networks," for an overview of the Chalice interface and the steps involved in creating grail files; chapter 6, "Node Panels and Parameters," for an explanation of how to set and change node parameters; and chapter 8, "Monitors and Flipbooks," to find out more about the features of Chalice monitors and how to use them.

IMAGE REPRESENTATION IN CHALICE

The sophistication of today's image processing tools have made it possible for digital artists to create effects that were unimaginable even a few years ago. But these tools generate numerous image types and file formats that are often not inherently compatible. However, all of these disparate file types must be successfully merged during the compositing process.

The advantage of the Chalice system is that it is able to recognize these differences in image formats and types, and it gives the user full control of the technical and creative processing of the imagery.

The first goal of the Chalice system is to easily import any image, in any format. Then the challenge becomes interpreting the image data and, if necessary, converting it to be compatible with other imagery in the composite without sacrificing image quality.

How Image Formats Vary

Digital image file formats can vary in a number ways. These differences can include how much space is allotted to storing information about each pixel, and how that information is stored—as linear or log data, compressed or uncompressed, and so on.

Each file type is tied to specific rules; for example, one format may allot only 8 bits per pixel, while another format will require floating depth (32 bits per channel) precision. Or the data stored by the file may represent different colorspaces, such as RGB or YUV.

Furthermore, a file type may represent a single channel of image data, three channels, four channels, and so on. And a channel of data may represent color data, matte data, or z-depth data. Almost all these formats also support images of varying dimensions (height and width).

Most of these file types also support some kind of compression that reduces the file size of the image using an encoding process that requires an opposite decoding process before the user can display the image. Some compression techniques are "lossless" and some are "lossy." In the case of the lossless techniques, all data in the original image is retained. In the lossy techniques, some data—perhaps not enough to be discernible to the human eye—is lost in the compression stage.

Note:

A complete list of file formats is available in the section on "Image File Formats Supported by Chalice" in chapter 12, "I/O Nodes."

Data Representation in Chalice

Once an image file has been imported into Chalice, the issue becomes how the data is represented. Chalice must concern itself with the number of bits that were used to represent the data, as well as what kind of data those bits represent.

The Chalice system provides maximum flexibility and control by separating the notion of file I/O from color representation. This enables you to control how image data gets converted within Chalice, based on the nature of the specific imagery and the project, so that you can preserve the most image information to get the highest quality output.

Bits per Channel

The Chalice system performs all its math internally as 8- or 16-bit per channel integer, or as 32-bit floating point operations, depending on the image data. You can work with image data in all of these bit depths in Chalice, even within the same file. At some point, however, this disparate data has to be combined in a composite operation.

Chalice enables you not only to control whether data is promoted to a higher bit depth or demoted, but also to customize the conversion, if desired, based on where the most important information lies in any particular image.

Nonlinear Data

There are two major categories of image representation: linear and nonlinear. Many commercial graphics software packages use linear data for their internal calculations, as does Chalice.

Linear data can be described as information that changes evenly no matter where you are in the metric space. The difference between 2 and 3 in the linear colorspace is 1. The difference between 105 and 106 is also 1 in linear colorspace.

Nonlinear systems, on the other hand, represent the data logarithmically. More of the metric space, the precious counts of the numbers you are using, is used where you need more color data and where you do not, it uses fewer counts. This data format is especially appropriate for representing visual imagery.

One of the most common nonlinear representations is that created by Cineon, a Kodak company. They chose to represent digitally scanned film data by taking the log base 10 of the linear data. In order to fully represent the negative film density range, Cineon determined that they required only 10 bits of log data. (To retain all the data found in this format and convert it to a linear form would require about 19 bits of linear data.)

In this nonlinear colorspace, the digital counts of the low numbers have more resolution—smaller increments of film density for each count—giving the blacks adequate information for proper representation.

On the other hand, the whites, the high numbers, do not need the same sensitivity and a count in the high ranges represents a larger increment of film density. This method applies the resolution where it is needed instead of applying the highest resolution to the whole colorspace, as with the linear method.

Cineon Conversion Options in Chalice

For Cineon log file conversion, Chalice offers a combination of flexibility and ease of use. By default, Cineon files are converted automatically when they are imported using a straight linear remapping to the full 16-bit linear space so that you can work with the imagery just as you would any other 16-bit linear format.

Unlike most other compositing programs, however, you can modify the default conversion, choose a different conversion method, or no conversion at all. You can convert Cineon log to floating point linear. (As mentioned above, it actually takes more than 16-bits per channel in linear space to represent all the log data.)

Or Cineon can be converted to 16-bit or floating point linear using the option we call "Cineon Linear," which takes film density factors into account when remapping the values. The Log 90% White value is mapped to Linear 90% White, instead of to the maximum value of the colorspace, and values above Log White are not clipped. This enables you to retain image data at the high end and to accommodate colorspace shifts later in the network. And you can change the default conversion values if appropriate for your imagery.

You can even convert Cineon files to 12-, 10-, or 8-bit linear, if necessary, using a conversion node that will allow you to specify the most important part of the image data to keep for any particular image.

This flexibility is also provided when converting imagery back to Cineon log for output.

You can also choose not to convert the log data at all. All Chalice operations expect to work on linear data, but this does not mean you cannot choose to perform Chalice operations on unconverted Cineon log. For certain operations the results could be unpredictable (although quite interesting), while for others, a crop, for example, you can expect reliable results operating on log data.

Whether converting log or linear data, Chalice always provides optional control over which segments of the input image colorspace are allotted the most room in the converted image.

For More Information

Refer to chapter 12, "I/O Nodes," for information about importing, converting, and rendering image formats. For information specific to Cineon conversion, refer to the DPX and Fido Parameter Tabs section of the Disk Input Node description.

Chapter 14, "Conversion Nodes," describes the individual nodes you can use to convert imagery at any point in a network.

Two documents from Kodak and Cinesite specify Cineon conversion, "Grayscale Transformations of Cineon Digital Film Data for Display, Conversion, and Film Recording," from the Cinesite Digital Film Center (1993), and "Conversion of 10-bit Log Film Data to 8-bit Linear or Video Data," from Kodak Motion Picture and Television Imaging (1995). These documents, in pdf format, are available for downloading from the Silicon Grail website.

HOW CHALICE FACILITATES TEAMWORK

Other sections of this chapter focus on issues that relate to the productivity of individual users working within Chalice. This section introduces another key way in which Chalice increases productivity: it encourages sharing work throughout a facility, not only among composite artists but also between the 2D and 3D departments.

For example, in a Chalice shop the 3D artists can prepare their temp comps with Chalice. Then when they hand their work off to the 2D people, some of the work has already been done. Not only that, but by definition, the artist will have tweaked the comp to make the 3D work better—these tweaks are now available to the composite artist, not as notes or as a discussion, but as an actual composite network. This also makes it possible to get feedback from the director or effects supervisor earlier.

This teamwork approach eliminates a lot of duplication of effort. The ability to easily share data among compositors not only saves time and effort, but it can help ensure consistency in the look of a finished series of shots. This concept is extended as far as possible by Chalice:

- The Chalice architecture allows more than one artist to work on the same shot at the same time.
- A 2D supervisor can stipulate a specific set of preferences for all compositors working on a project.
- Using presets, several artists can share a look or a technique, without bothering to learn how it works.
- Groups of nodes which have been customized for a particular task can be added to the node menus of all Chalice users, who can then add them to their networks as easily as they can any other node. For example, a supervisor can pass out the same color corrections to an entire staff.
- Successful techniques and ideas can be saved for use on future productions—thus Chalice users begin to build an inventory of expertise that they can apply to future work, to either speed up that work or do a better job in the same time, or both.

For More Information

Refer to chapter 7, “Managing Large Networks,” which covers the creation of custom networks, to chapter 6, “Node Panels and Parameters,” which explains how to create and use various presets, and to chapter 11, “Setting Preferences.”

MEMORY USAGE IN CHALICE

Image processing is, of course, memory-intensive. The following information about how Chalice 2.0 uses available RAM and disk space to cache data is provided to help you understand how to use Chalice caching options to your best advantage.

What Is Caching?

As each Chalice node completes its task, it saves, or caches, a copy of what it has just computed. This is so that if another request comes for the same image data, rather than go through all the computations again, the node can just hand off the cached result. This saves a lot of time, but uses a lot of memory space, as each node in a large network can potentially save one or more images.

Data is cached in two ways: in RAM and on disk. The **RAM cache** is kept in main memory. RAM provides much faster access, but there is less of it, and some RAM must be kept available for the operating system and for Chalice itself.

Disk cache refers to a section of the hard drive that has been allocated to Chalice. This memory space is potentially much larger, but it is also much slower to access, which is why disk caching is used only after RAM cache has filled up.

Also, Chalice 2.0 will only cache to disk if the time to recompute the data is greater than the time it would take to write it to disk. In other words, Chalice will not bother to cache data to disk if it would be faster just to recompute it.

Direct I/O

Chalice 2.0 uses direct I/O to write data to disk. That is, Chalice handles the write directly, bypassing the operating system. This makes all I/O, including caching, both faster and more memory efficient.

This is true for NT, Linux, and IRIX, but it is especially noticeable on IRIX systems. The problem with handing data off to the operating system is that the OS itself buffers data, so that handing 10MB of data to the OS to write to disk when RAM is tight could actually mean that another 10MB of RAM would be allocated by the OS to buffer it. This would cause a vicious circle of less RAM causing more disk writes, which would further reduce RAM, and so on, until performance ground to a halt.

How Caching Works

A simple two-node network in Chalice can illustrate how caching works. Assume that a Disk Input node is connected to a Gamma node. What happens when we ask Chalice to bring up a monitor on the Gamma node?

The Gamma node requests an image on which to do the gamma correction from the Disk Input node. The Disk Input node accesses the disk and reads in the image that is relevant at the current frame. To do so, it allocates a block of image memory, and then reads the image into it.

At the end of this read operation, the Disk Input node does not throw away the image; instead, it hangs on to it, assuming that it may be asked for that same image again in the future. This becomes the first cached image.

When the Gamma node receives the image from the Disk Input, it applies the gamma correction to it. As well displaying the result in a monitor, it also saves the calculations for any future request. This becomes the next cached image.

Controlling Individual Node Caching

Most nodes store a single image, or "cook." Using the Gamma node example, if the gamma value changes and is recomputed, the Gamma node throws away the earlier result and caches the latest one instead.

But the Info tab of every node provides parameters you can use to change this behavior. It is possible to tell a node to store up several images at once by increasing the value of the Cache Size parameter from the default value of 1.

> **Note:**
>
> Certain nodes, such as Time Convolve, will do additional caching anyway unless you turn off the "Auto Cache" feature, because by their nature they will be using data from more than one frame to modify the image.

How Chalice Prioritizes Node Caching

Chalice always tries to ensure that any image data that might be needed again is available without recomputing, and it always tries to avoid using the disk cache if possible.

But Chalice also tries to conserve memory, so when the cache begins to get full, it uses certain guidelines to decide which data is most likely to be reused. For example, a node whose output branches to several other nodes has a high priority, as its data is likely to be requested by more than one node branch.

Chalice never caches if it is not useful. Chal renders are not cached since there is no frame to frame coherence, and the Exec node doesn't save image data (since it has no output).

Image Slicing

Slicing is an option available when writing imagery to disk and when cooking image data in a monitor. Slicing is intended to dramatically reduce the memory requirements of Chalice by slicing an image into segments for processing.

Ideally, slicing an image into eight parts would use only one-eighth the memory, as each slice would be treated as a separate image. This would mean that instead of a node caching an entire image, it could cache a slice at a time for reuse.

But any node that is feeding a node which is being monitored or modified has to store not just one slice, but all of them, so that the complete image is available to the requesting node.

Shared Memory Segments

Each time a node allocates memory for an image, it allocates shared memory. This allows images to be shared between the work process (chal) and the UI process. It is possible for every node in a large network to contain an image, and thus use a shared memory segment. Because *all* allocated (image) memory is set up as shared memory, chal requires a lot of shared memory segments too.

If slicing is being used, the number of segments can go up still further, even though the total amount of memory used is smaller. This is because some nodes end up requesting a lot of small pieces of slices. The Blur node, for example, needs some pixels from slices above and below the current one to calculate the pixels at the top and bottom of the slice. That requires more memory allocation and thus more shared memory segments.

And in the Transform node, rotating can result in many little pieces of slices as images get rotated out of frame or out of the current slice. This means that grail files with numerous Transform nodes can result in high shared memory segment counts.

Fortunately, allocating large numbers (over 2,000, e.g.) of shared memory segments has no real effect on the operating system, except to increase the size of the kernal by a few thousand bytes. For most users of most machines, this will make no noticeable difference to performance.

For More Information

Use of the node caching parameters on the Info tab is covered in chapter 6, "Node Panels and Parameters." The Cache Settings panel is accessed from the Chalice Admin Menu, which is covered in chapter 10, "Main Menu Bar."

5 BUILDING CHALICE NETWORKS

You compose cinematic special effects in Chalice by building a graphical network of nodes, wherein each node represents a step or process involved in the composition of the effect.

Creating a Network

You create a Chalice file (also called a grail file), select a node from the Node menu, and place the node in the large on-screen workspace. Then you assign parameter values to the node to control its functions. Next, you create a subsequent node, assign parameters, and link the previous node to it.

You repeat this basic operation over and over, building a network that represents a logical schematic of all the processes required to create the scene you desire. Monitors and other interactive tools are available at each node to examine and adjust the imagery at any point in the network.

Because each node represents a separate step in the image processing dataflow, you can easily pinpoint the precise places at which you may need to make adjustments to the node parameters or modifications to the dataflow.

The nodes you select, the parameter values you assign to them, and the order in which you connect them all influence the resulting scene.

Note:

For a detailed discussion of the dataflow process, refer to "Dataflow Mechanics of the Chalice System" in chapter 4, "Key Concepts."

The following sections introduce the Chalice interface and explain in detail how to work with grail files, build networks of nodes, and use the features common to most nodes:

Launching Chalice ..... p. 5.2
Overview of the Chalice Screen Layout ..... p. 5.3
Working with Grail Files ..... p. 5.6
Creating Node Networks ..... p. 5.10
Overview of the Node Layout ..... p. 5.15
Setting Node Display Options ..... p. 5.17
Previewing Node Output ..... p. 5.18

LAUNCHING CHALICE

This page explains how to launch the Chalice interactive graphical user interface. In most cases, you will use this interactive mode to build your networks.

However, you can also run "chal," the command line version of Chalice, which enables you to batch process Chalice grail scripts that were previously created in interactive mode. Refer to appendix A, "Chal," for more information about using chal.

To start Chalice, follow the steps below for your operating system:

IRIX Users

1. **Log in to your system and open a UNIX shell.**
2. **Use the IRIX change directory command** "cd" **to change to the directory that contains the files with which you want to work.**
3. **Type** "chalice."
   The system will launch Chalice and open a new file called "untitled.grail." Files created by Chalice are referred to as "grail files" and end with the .grail extension by default.
4. **To open an existing grail file, use the Open command under the File menu.**
   For more information about creating, opening, and saving files, refer to the section on "Working with Grail Files" later in this chapter.

Alternatively, you can invoke Chalice from any directory by typing "chalice" followed by the complete pathname of the grail file in which you plan to work. This will launch Chalice with that specific file open. For example:

```
chalice /usr/people/richard/myfile.grail
```

NT Users

Double-click the Chalice application icon to launch Chalice.

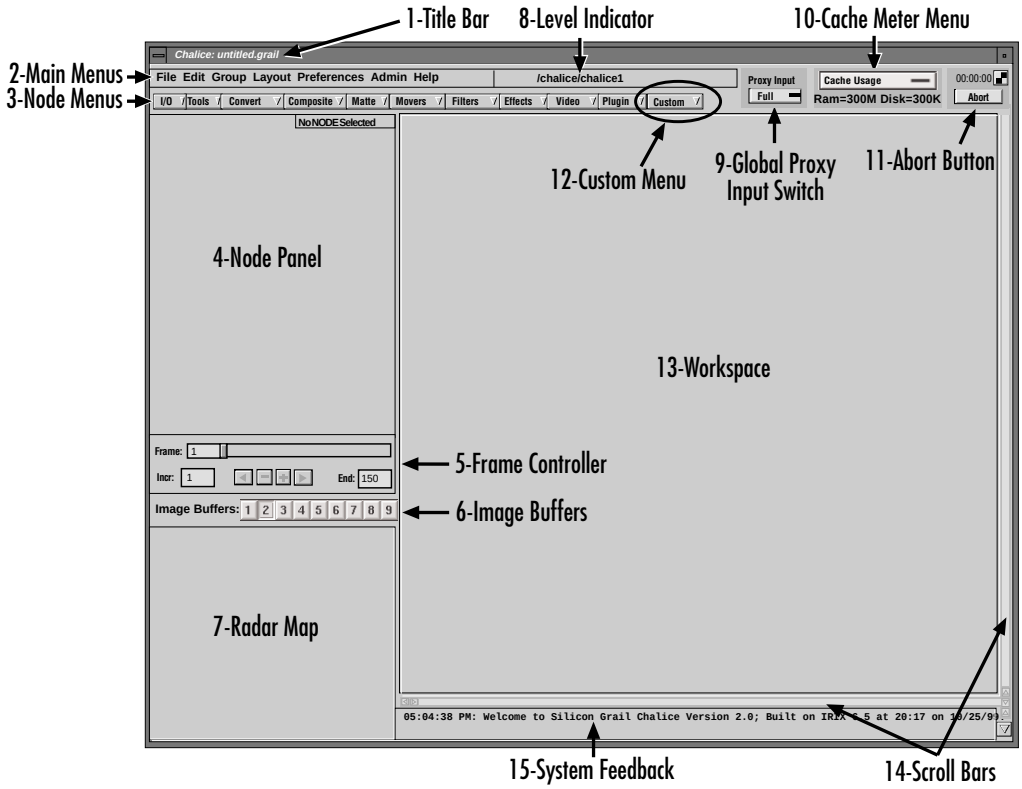
Alternatively, you can select Programs > Command Prompt from the Start menu and type "chalice" in the Command Prompt window.

By default, a new file called "untitled.grail" will be created. Files created by Chalice are referred to as "grail files" and end with the .grail extension by default.

For more information about creating, opening, and saving files, refer to the section on "Working with Grail Files" later in this chapter.

OVERVIEW OF THE CHALICE SCREEN LAYOUT

This section provides you with an overview of the Chalice interface.



1 - Title Bar

The Title bar displays the name of the current grail file.

2 - Main Menu Bar

The Main Menu bar enables you to perform file management functions, such as opening and saving grail files and grouping and deleting nodes. It also enables you to set a variety of options and preferences to control the Chalice layout. The Help menu gives you access to Chalice online help features.

For more detailed information about the function of each menu item, see chapter 10, “Main Menu Bar,” and chapter 11, “Setting Preferences.”

3 - Node Menu Bar

The Node Menu bar enables you to select the type of node you wish to create. You then place the node in the workspace and assign parameters to control its function.

For more information, see “Creating Node Networks” or refer to the individual node descriptions, which are grouped by category in the Node Reference chapters of this manual.

4 - Node Panel

When you create a node, or select a node in a network (by clicking the node title bar), the Node Panel displays information related to the selected node. This information includes the node type, name, and a set of parameter tabs for the node. You set the parameters on the Node Panel to control the function of the node. Refer to chapter 6, “Node Panels and Parameters,” for details.

To learn about the function and parameters specific to an individual node, refer to the individual node descriptions in the Node Reference section of this manual.

5 - Frame Controller

The Frame Controller is a frame navigator for the tools that display or manipulate data in the currently active node, including the Node Panel, Frame Icon, Monitor, and STED. For basic information about operation of the Frame Controller, see "Previewing Node Output."

To control playback of animation, use the Monitor Flipbook controls instead. The Flipbook is described in chapter 8, "Monitors and Flipbooks."

6 - Image Buffers

Chalice image buffers store image data for display in node monitors. By default, each time you open a node monitor the image is stored in the next available image buffer. When one of the image buffer buttons is depressed, it indicates that the corresponding buffer will be used to store the next image that is processed, or "cooked." You can click any button to specify that buffer as the one to use for the next cook. Chapter 8, "Monitors and Flipbooks," explains how to use the image buffers.

7 - Radar Map

The Radar Map provides a macroscopic view of the entire grail network. When your network is larger than the visible workspace, a bounding rectangle appears in the Radar Map to designate the portion currently visible in the workspace. Click and drag this rectangle to center any part of the network in the workspace.

The Radar Map also displays processing status: the node icons in the map flash yellow and green to indicate that data is being sent, received, or calculated. A red node icon indicates an error (click the info button on the actual node in the workspace to read the error message).

8 - Level Indicator

The Level Indicator identifies the currently active level displayed in the workspace. For any new grail file you create, the Level Indicator will always read "/chalice/chalice1." The level indicator only becomes relevant if you group nodes.

You have the option to "push into" the group, which means that Chalice will display the contents of that Group node in the workspace in place of the main network. The Level Indicator will change to display the complete pathname of the Group node, for example, "/chalice/chalice1/group1" (if the group is named "group1").

The process of Grouping Nodes is described in chapter 7, "Managing Large Networks."

9 - Proxy Input

The Proxy Input menu is a global switch that enables you to set the proxy level to work in for the grail file as a whole. The Proxy Input level—Full, Medium, or Low—determines which resolution of an image input file is accessed by the network, assuming that lower resolution proxy files have been specified in one or more Disk Input nodes. For a complete description, refer to "Using Proxies in Chalice" in chapter 12, "I/O Nodes."

10 - Cache Meter

By default, the cache meter displays the current RAM and Disk Cache Usage. Click on the meter menu to choose another meter display: RAM Usage, UI Heap, or Work Heap. (UI Heap and Work Heap refer to the fact that Chalice is a multiprocessed program and the two main processes are the UI process and the work process.) The menu also provides a command to "Flush All Caches."

11 - Abort Button

Chalice does not force you to wait until the current process is done if you need to stop it for any reason. Click the Abort button to stop the current process. The Abort button is associated with a process timer that displays elapsed time for the current processing operation.

12 - Custom Menu

Located at the right end of the Node Menu bar, the Custom menu provides you with convenient access to any node or group of nodes you designate. The custom nodes you create appear in a pulldown menu when you click and hold the Custom menu button.

For more information, see "Creating Custom Networks" in chapter 7, "Managing Large Networks."

13 - Workspace

The workspace is the large area on the right side of the screen in which you build your Chalice network. Once you select the type of node you desire from the Node Menu, you place it in the workspace and connect it to other nodes to form a network.

The workspace contains all nodes in the current level (see the "Level Indicator" description for more information); however, node networks may be too large to view in their entirety without scrolling. Click and drag the workspace, or use the scroll bars, to navigate through large node networks.

You can also use Zoom mode (Alt z) to view and navigate through complex grail files. For more information on using Zoom and other related features, refer to chapter 7, "Managing Large Networks."

14 - Scroll Bars

The Scroll Bars enable you to scroll through the workspace of a large network. Alternatively, you can click and drag the workspace area in any direction.

15 - System Feedback Area

The System Feedback area provides you with information related to file activities in a scrollable area at the bottom of the screen.

WORKING WITH GRAIL FILES

This section explains how to open, name, save, and quit Chalice grail files.

Creating a New File

When you launch Chalice, it creates a new file called "untitled.grail" by default. The name appears in the Chalice title bar in the upper-left corner of the screen. You can also select "New" from the File menu.

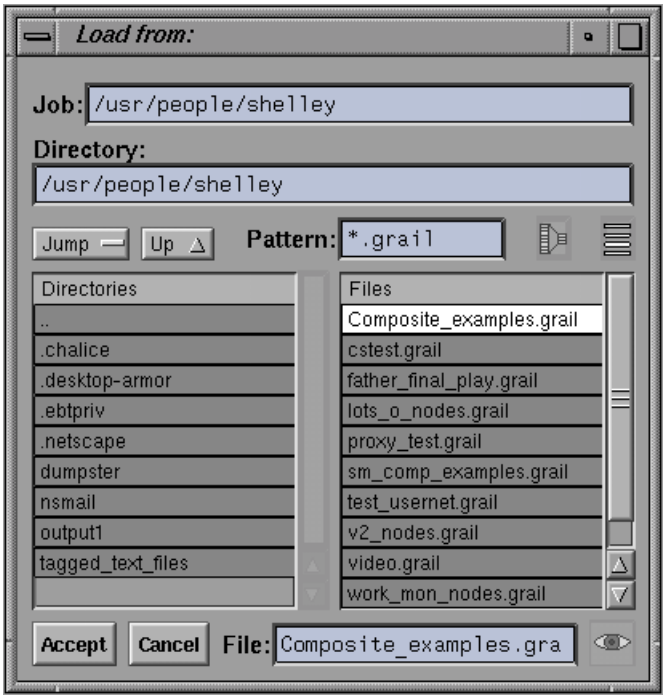
Opening an Existing File

You can call a specific file when launching Chalice from a UNIX shell or NT Command Prompt window by typing the "chalice" command followed by the filename. For example:

```
chalice filename.grail
```

Or you can use the Open command in the File menu to open a file from within Chalice:

1. **Pull down the File menu from the Chalice Main Menu and select the "Open" command.**
   Chalice displays a "Load from:" dialog box that you use to specify the file to load.



2. **Specify the directory that contains the file you wish to open.**
   By default, Chalice file dialogs display your login home directory in the Directory field. You can type the pathname of the directory you wish to access into the Directory field (be sure to press the Enter key when you are done). Alternatively, you can click on directory entries in the Directories List to navigate through your file system.

   You can also use the Jump and Up buttons to navigate through the directory hierarchy:

   - **Jump** moves you to recently accessed directories, as well as providing direct navigation to the $JOB and $HOME directories.
   - **Up** moves you one step higher in the directory structure.

The Job field will contain the directory path that has been defined for the $JOB variable. Job lets you associate files with a specific project. See "Setting Global Variables" in chapter 6 for more information.

3. **Double-click the name of the file in the Files List that you wish to open.**
   Alternatively, you can key the name of the file into the "File:" field and click the Accept button.

   Chalice loads and displays the specified grail file.

   Optional: You may click the Cancel button instead of the Accept button to cancel the load operation and dismiss the dialog.

Opening Files Created in Older Versions of Chalice

When you upgrade to the latest version of Chalice, you can open a file that was created in the previous version by following the same procedure outlined above. Chalice will convert the file before opening, as follows:

- The grail file is converted to the new version.
- The converted grail file is renamed by replacing the original ".grail" extension with the "v.X.X.grail" extension, where "X.X" represents the current version number.
- The converted, renamed grail file is saved to the same directory as the original file (the old grail file is not deleted).

Chalice loads and displays the file, along with a notifier informing you that the conversion process has taken place.

Note:

Because a new file is written to disk, you must have write permissions for the relevant directory or you will not be able to open the file.

Saving a New File

To save a new, untitled grail file:

1. **Pull down the File menu from the Chalice Main Menu and select the "Save" command.**
   Chalice displays a "Save to:" dialog box.

2. **Specify the directory in which you wish to save the file.**
   (See step 2 of "Opening an Existing File" above for more information.)

3. **Key a new name into the "File:" field.**
   Chalice assigns new files the name "untitled.grail" by default. You can change the name when you save the file.

   SGI IRIX Users:

   The name must conform to standard UNIX nomenclature. You should avoid using spaces or special characters such as the semicolon (;), backslash (\), ampersand (&), exclamation point (!), asterisk (*), or pipe (|). (For more information, refer to your IRIX documentation.)

   NT Users:

   We recommend that you follow UNIX naming conventions to ensure compatibility when exchanging files across platforms.

4. **Click the Accept button.**
   Chalice saves the file using the name and directory location you specified.

Files saved in the current version of Chalice cannot be opened in earlier versions of the software.

Using "Save As"

You can save a copy of a currently open grail file at any time by renaming it using the "Save as" command under the File menu. You must assign a unique name, unless you want to overwrite an existing file.

To rename the file:

1. **Pull down the File menu from the Chalice Main Menu and select the "Save As" command.**
   Chalice displays a "Save to:" dialog box.
2. **Follow the directions for "Saving a New File" above, starting with step 2.**

Merging Files

You can merge the contents of another file into the currently open grail file using the Merge command under the File menu.

Tip:

To avoid placing the merged nodes on top of existing nodes, be sure to scroll to a clear area of the workspace before selecting Merge from the File menu.

To merge two files:

1. **Pull down the File menu from the Chalice Main Menu and select the "Merge" command.**
   Chalice displays a "Load from:" dialog box.
2. **Select the grail file you wish to merge into the currently open file.**
   See "Opening an Existing File" for more information on navigating the directory structure to locate a specific file.
3. **Double-click the name of the file in the File List that you wish to merge.**
   Chalice places the contents of the specified file in the center of the workspace of the currently open file.

Exporting Nodes to a New File

You can export any selected node or nodes to a new file using the "Save Selected As" command in the Chalice File menu:

1. **Pull down the File menu from the Chalice Main Menu and select the "Save Selected As…" command.**
   Chalice displays an "Export to:" dialog box.
2. **Specify the directory in which you wish to save the file.**
   See step 2 of "Opening an Existing File" if you need more information.
3. **Key a name for the new file into the "File:" field.**
   See step 3 of "Saving a New File" for information on file naming conventions.

4. **Click the Accept button.**
   Chalice saves the file using the name and directory location you specified.

Quitting Chalice

1. **Pull down the File menu from the Chalice main menu and select the "Quit Chalice" command.**
   Chalice displays a notifier panel that allows you to

   - save the currently open file and quit Chalice,
   - discard any changes made to the open file during the current session and quit Chalice,
   - cancel the quit operation.

CREATING NODE NETWORKS

This section shows you how to create, connect, and delete nodes to build node networks. Each type of node performs a different task or function necessary to create a particular shot:

- You control the processing performed by a node by adjusting the set of parameters associated with that node.
- You create a flow of information from one node to the next by connecting the input of a node to the output of another.

Creating Nodes

You create nodes by selecting a node from the Node Menu and placing it in the workspace, or by selecting a node from the Node Connector Menu on an existing node. Both methods are described below:

Selecting Nodes from the Node Menu

The following procedure explains how to select a node from the Node menu.

1. **Use the left mouse button to pull down one of the Node menus.**

   The nodes are sorted into menus by category: I/O, Tool, Convert, Composite, Matte, Movers, Filters, Effects, Video, and Plug-in.

   - An individual plug-in node may appear both in the Plug-ins menu and in a category menu, as appropriate. You can identify a plug-in in any other node menu because its name appears in lighter type.
   - The last menu in the node menu bar is the Custom menu, which contains nodes or node networks you define. For more information about them, see "Creating Custom Networks" in chapter 7.

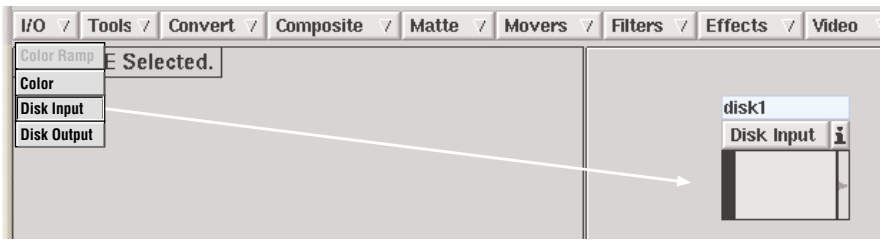
2. **Select the type of node you wish to create.**

   You can select any node, depending on the requirements of your Chalice network (e.g., Blur, Mcomp, Xpresso, etc.).

3. **Slide the cursor to the workspace and click the left mouse button.**

   As you move the cursor into the workspace, Chalice creates a small rectangular outline of the node.

   When you click the mouse, Chalice places the new node in the workspace and assigns it a unique name. (In the illustration below, the node is a Disk Input node and the default name is "disk1.")



Selecting Nodes from a Node Connector Menu

The following procedure explains how to select a node from the Node Connector Menu, which is a popup menu accessed from the input and output connectors of existing nodes. For more information about the parts of a node, see the section "Overview of the Node Layout."

This method is a convenient shortcut for node creation. It creates a new node and connects it to the previous node all in one step. You can also use this method to create a new node from the input connector of an existing node. The output of the new node will automatically be connected to the input of the node from which it was created.

1. **Use the middle mouse button to click the output connector of a node.**
   Chalice displays the Node Connector Menu for as long as you press the mouse button.

   This popup menu contains a list of every node type with an input connector to which the output of the current node can be connected. (The node menu on an output connector will not include nodes without input connectors, such as Color and Disk Input.)

2. **Select a node from the popup menu and release the mouse button.**
   Chalice creates a rectangular outline of the new node under the cursor position.

3. **Click the workspace wherever you want the new node to be placed.**
   Chalice places the new node in the workspace. The input of the new node is automatically connected to the output of the node from which it was created.

Say, for example, that you connected a Disk Input node with a foreground element to an Mcomp node, but the element needs to be color corrected before compositing. Middle-click the input connector of the Mcomp node (use the connector attached to the Disk Input) and select "Color Correct" from the Node Connector Menu.

When you place the new Color Correct node in the workspace, its output connector will be attached to the Mcomp input and its input connector will be attached to the output of the Disk Input node.

Assigning Parameter Values to Nodes

Once you place the node in the workspace, Chalice displays a set of parameter tabs for the node in the Node Parameter panel. You then assign parameter values to the node to control its functions. For more information, see "Overview of the Node Panel" and "Setting Node Parameters" in chapter 6.

Connecting Nodes

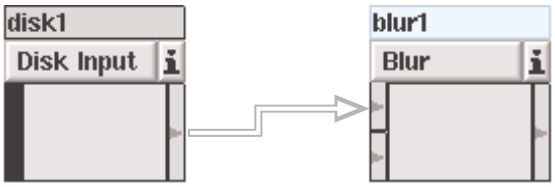
Once you have created a node and assigned parameter values to it, you then create a subsequent node and link the previous node to it. You repeat this basic operation over and over, building a network that represents a logical schematic of all the processes required to create the scene you desire.

You can think of a network as a stream of data. The stream starts at a source, for example, a Disk Input node, and the data flows downstream (to the right) through the network, passing through various nodes until it reaches a Disk Output node, which is connected to an Exec node. (For a detailed description of this process, see chapter 4, "Key Concepts.")

The following procedure describes how to connect one node to another. It assumes that you have already created more than one node.

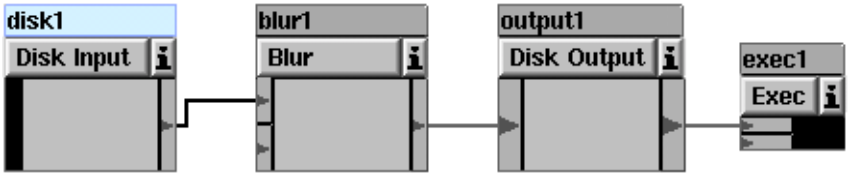
1. **To connect one node to a subsequent node in the network, click on the output connector of the first node.**

2. **Slide the mouse toward the second node.**
   Chalice begins to draw a connector line.

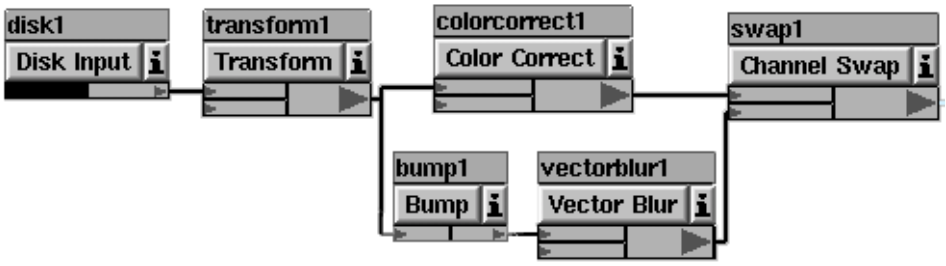


3. **Click on the input connector of the second node.**
   Chalice attaches the connector line to the input connector of the second node.

Some nodes, such as the Disk Input and Color nodes, do not have input connectors. These nodes are used only at the beginning of a network. The Exec node does not have an output connector; it can be used only at the end of a network.



You can draw more than one connector line from any output connector. This enables you to split the dataflow into two or more separate streams for processing.



Each input connector on a node can accept only one connection line. However, depending on their function, many nodes have multiple input connectors.

For example, the Over node has two input connectors, one for the foreground image and one for the background of the composite. The Mcomp node, on the other hand, which can composite a virtually unlimited number of layers, creates additional inputs as needed.

Connecting Nodes in Large Networks

When working with large, complex grail files, all nodes may not fit in the visible portion of the workspace at once.

To connect two nodes that are located far from each other in the workspace, click the output connector of the first node and use the Scroll Bars or Radar Map to move to the area of the workspace that contains the second node.

Note:

Do not click and drag in the workspace itself to move—this will delete the connector line.

Connecting Nodes Located on Different Levels

If a grail file contains Group nodes, you may need to connect a node within the group to an external node. In this case you create export flags for the node connector on the internal node. The external node is connected to the group node connector that points to the internal node. This process is described in detail in the "Grouping Nodes" section of chapter 7.

Disconnecting Nodes

To disconnect a connector line from a node, click on the node's input connector, then click again in the workspace.

Tip:

If you create a connector line from a node by mistake, press the Escape key (while the line is still "attached" to your cursor) to delete the line.

Setting Node Connection Options

You can adjust the appearance of your network node connections in two ways:

Link Straight Option

The Layout menu on the Main Menu bar features a Link Straight option. This option requests that Chalice redraw all connector lines in the network as straight lines, without right angles.

New Layout Option

The Layout menu on the Main Menu bar features a New Layout option. This option rearranges the node layout in the workspace into a grid pattern based on node hierarchy, with the Exec node (if one exists) in the upper-right corner of the workspace. All upstream nodes are arranged based on node type and connections used.

You cannot Undo this change, although you can revert to a previous autosaved version of the grail file. Be certain that you want Chalice to rearrange your layout before you set this option, especially if you have a large and complex network.

Selecting Multiple Nodes

Chalice provides the following methods for selecting multiple nodes. Whichever method you use to select nodes, the title bars of all currently selected nodes will be highlighted in blue:

- Pick All Ancestors
- Pick All Descendents
- Control-Drag Selection
- Shift-Click Selection
- Select All Nodes Command (Alt A)

Pick All Ancestors or Descendents

The Local Node Menu of every node includes these options:

- Choose **Pick All Ancestors** to select all nodes connected *upstream* from the current node in the same network.
- Choose **Pick All Descendents** to select all nodes connected *downstream* from the current node in the same network.

Control-Drag Selection

Hold down the Control key while simultaneously using the left mouse button to click and drag in the workspace. Chalice will draw a box bounding the area you define by dragging. When you release the mouse button, all nodes located within the boundary area will be selected.

Shift-Click Selection

You can hold down the Shift key and click the title bars of all nodes you want to select as a group.

Select All Nodes Command

This command can be accessed from the Chalice Edit menu, or by using the keyboard command "Alt A," to select all nodes in the workspace.

Deleting Nodes

To delete a node from the workspace, follow these steps:

1. **Click the Local Node Menu button with the left mouse button and hold.**
   Chalice displays the Local Node Menu.

2. **Select the Delete NODE option.**
   Chalice deletes the node.

Alternatively, you can delete a node by selecting it (click the node title bar) and choosing the "Delete NODE(s)" command from the Edit menu of the Main Menu. You can also press the Delete key on the keyboard to delete the selected node.

The Edit menu of the Main Menu also includes an "Undo Delete" command, which you can use to restore nodes you have just deleted.

Replacing Nodes in One Step

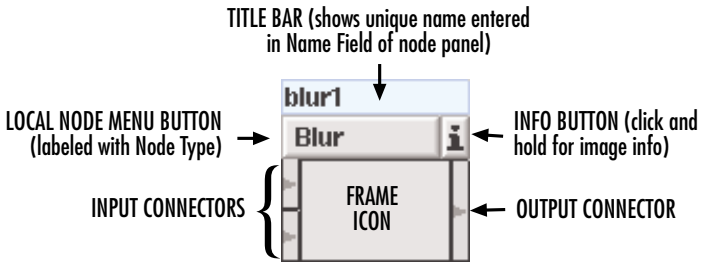
You can delete a node and replace it with another type of node in a single step by using the middle mouse button to access a replacement node menu.

1. **Click the Local Node Menu button with the middle mouse button and hold.**
   Chalice displays the menu of compatible replacement nodes for selection. If you want to replace a Disk Input node, for example, the replacement node menu will contain only the Color node (Disk Input and Color are the only two nodes without input connectors.)

2. **Select the node type you want from the those available in the menu.**
   Chalice deletes the current node and replaces it with node type you selected. All input and output connections remain intact.

OVERVIEW OF THE NODE LAYOUT

This section provides you with an overview of the format and layout of the Chalice node. Although there are many node types within Chalice, they share several common features.



The major components of the node are described below:

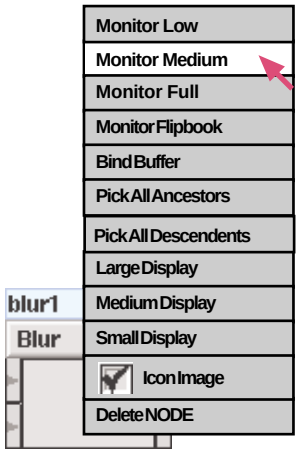
Node Title Bar

The Node Title Bar displays the name of the current node. Chalice assigns each new node a unique name (for example "dissolve1," "dissolve2," etc.). You can rename the node at any time by typing a new, unique name into the "Name:" entry box on the node panel. See also "Overview of the Node Panel" in chapter 6.

You click the title bar of any node to make it active; the title bar of the currently active node will be highlighted in blue and the node parameters for that node will appear in the node panel (on the left side of the Chalice screen).

Local Node Menu Button

Each node includes a Local Node Menu button that enables you to display a popup menu of options for that node by clicking and holding the Local Node Menu button with the right or left mouse button. The Local Node Menu button is labeled with the name of the node type (for example, Disk Input, Blur, etc.).



You can use the Local Node Menu to access a Monitor at various resolutions, to create a Monitor Flipbook, or to turn Frame Icon on and off for the node (using the "Icon Image" checkbox). The only exception is the Exec node, and any Group nodes you create, for which these specific options are not applicable.

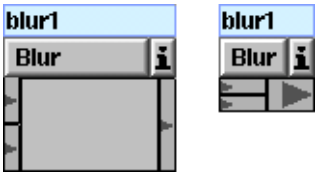
You can reset the display size of the node—see "Setting Node Display Options" for more information—or you can delete the node altogether. And you can bind the node to an image buffer, as explained in "Using the Display Buffers" in chapter 8, "Monitors and Flipbooks."

The “Pick All” options enable you to select nodes connected to the current node, as explained in the previous section on “Selecting Multiple Nodes.”

You can also use the Local Node Menu button to replace the current node with another type of node, keeping all input and output connections intact: Middle-click and hold the Local Node Menu button to access a popup menu of compatible node types.

Frame Icon

The Frame Icon displays a thumbnail image of the current frame in the image sequence at the current stage of processing in the network. (You specify the current frame number in the Frame Controller.) All nodes except Exec and Group nodes include the Frame Icon option.



Frame Icon can be turned on and off using the Local Node Menu or the Images option under the Preferences menu. For more information, see “Setting Node Display Options.”

Tip:

To update the thumbnail images for all nodes in the network, select the Update All Icons command in the Chalice Layout menu.

Input Connector(s)

You flow data into a node by linking the output connector of one node to the input connector of a subsequent node.

All nodes have one or more input connectors (except for the Color and Disk Input nodes, and any Plug-in nodes that you add to the Chalice system that do not require input connectors). For multiple-input nodes (nodes that accept more than two inputs), Chalice adds an additional input connector as soon as there are no vacancies.

For more information about using node connectors, see the section on “Creating Node Networks” elsewhere in this chapter.

Tip:

You can create new nodes directly from existing nodes by using the Node Connector Menu, which is accessed by middle-clicking on any Input or Output connector. The procedure is described in the “Creating Node Networks” section.

Output Connector

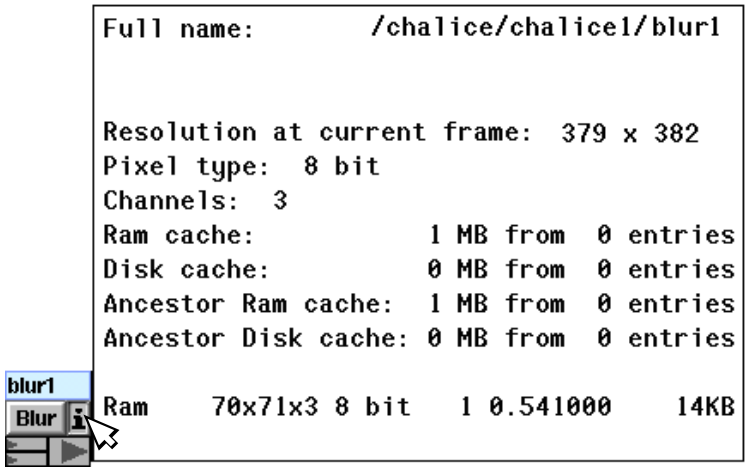
You flow data out of a node by linking the node’s output connector to the input connector of a subsequent node. All nodes have one output connector, except for the Exec node, which has none. You can connect the output connector of a node to the input connectors of more than one other node.

Note:

Group nodes can have no outputs or multiple outputs. For more information about creating and using Group node connectors, see the description of "Grouping Nodes" in chapter 7.

Info Button

Click and hold the Info button to display information about the node. See the description of the "Info Tab" in chapter 6 for details. When Chalice detects an error, the Info button on the relevant node turns red and the "i" turns into an exclamation point. Click and hold the button to display the error message.

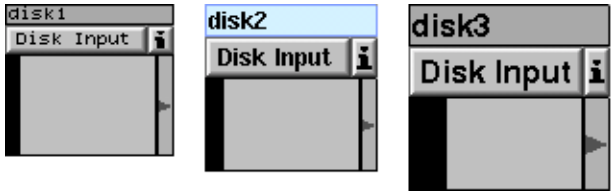


Setting Node Display Options

You can adjust the display mode of individual nodes in your network using the Local Node Menu, or of all selected nodes by using the command in the Chalice Layout menu.

Using the Local Node Menu

The Local Node Menu on each node includes options for Large Display, Medium Display, and Small Display. You can adjust the size of the current node by selecting one of these options. The default size is Medium Display. This option affects only the current node.



You can also specify whether or not the Frame Icon should be displayed for a node using the "Icon Image" option in the Local Node Menu.

Using the Layout Menu

The Layout menu includes commands that affect the display of all selected nodes in the workspace, including showing or hiding their frame icons, updating the icon displays, and changing node size, as described in chapter 10, "Main Menu Bar."

Node Display Preferences

The Node Preferences panel, which is described in chapter 11, "Setting Preferences," enables you to control the default display of new nodes, including size, icon display, and menu type.

PREVIEWING NODE OUTPUT

Chalice lets you preview the image output of a node using the following tools:

- Frame Icon
- Monitor
- Flipbook

When you click the Frame Icon of a node, it displays a thumbnail version of the current frame of your imagery as it is output from that node.

Note:

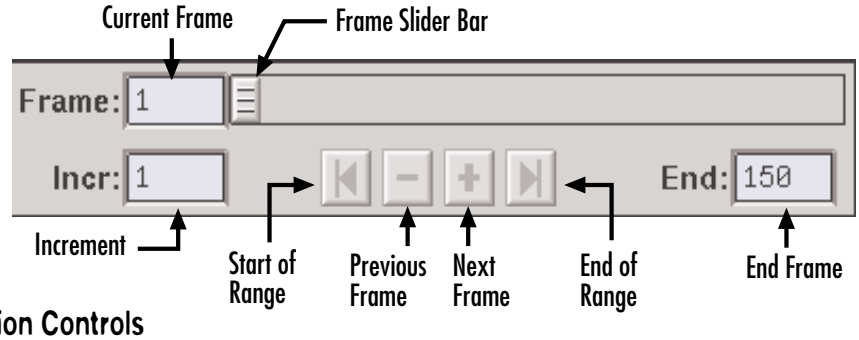
The icon image in the Frame Icon of a node will appear black, or very indistinct, if the node is processing Cineon images that have been converted using a non-default method in the Disk Input Node. For more information about working with Cineon imagery, refer to "Image Format Notes" in chapter 12, "I/O Nodes."

Of course, the frame icon image is used as a general reference and not to examine imagery in detail. To view individual frames, open a Monitor. To view a sequence, create a Flipbook. Monitors and Flipbooks are both accessed from the Local Node Menu. (For more information about them, refer to chapter 8, "Monitors and Flipbooks.")

Using the Frame Controller

The Frame Controller is used to specify the current frame to display in both the Frame Icon and the Monitor. (The Frame Controller is not used to control the playback of animated sequences; this function is handled by the controls in the Flipbook.)

You must click the Frame Icon of a node to actuate the change for the node icon. In a Monitor, you must click the Eye icon ("cook" button) to actuate the change, unless you have the Monitor Auto Update option checked.



Navigation Controls

Use one of the following methods to navigate to any frame in a sequence:

- Click the Go to Next Frame (+) or Go to Previous Frame (-) buttons.
- Click the Start of Range button to go to the first frame in the sequence or the End of Range button to go to the last frame in the sequence.
- Drag the Frame Slider Bar until the frame number you desire appears in the Current Frame field.
- Key a frame number directly into the Current Frame entry field.

Frame Navigation Hot Keys:

In any Chalice monitor, you can use the comma (,) and period (.) keys on your keyboard to navigate back or forward a frame.

Increment

This field enables you to specify how many frames to increment or decrement when you use the Next or Previous Frame buttons. By default the increment is set to 1.

End Frame Field

You use the Frame Controller to specify the total number of frames to operate on in the current sequence by typing a number that represents the last frame in the sequence into the "End" data entry field. (The first frame is assumed to be "1.")

The End Frame value is used by Chalice to set an initial sequence length for animated parameters—not to alter the actual number of frames in an input sequence.

You can revise this value at any time for any node. When you open the Space-Time Editor (the STED), the frame range in the graph will match this setting.

Changing the "End Frame" Value

When you reset the current End Frame number in the Frame Controller, a notifier panel will appear to confirm that you want to decrease the length (if you set the end frame to less than the current number of frames) or to increase the length (if you set it for more than the current number).

This notifier panel gives you the option to simply truncate or append the range, which are the default options, or to squash or stretch the range. The "squash" and "stretch" options refer not to the number of frames but to parameter data that has been animated:

- The default options, **truncate** and **append**, simply shorten or lengthen the range without affecting existing frame values. In almost all cases, you will want to use the default options.
- The **squash** and **stretch** options will move any key frames, resetting all existing animated parameters to the new frame range.

6 NODE PANELS AND PARAMETERS

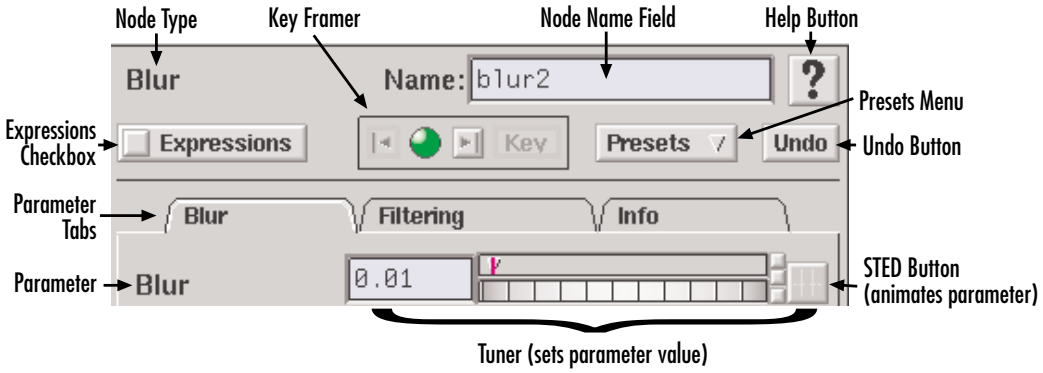
Chalice node panels provide access to all of the parameters and other controls available for each node type. The node panel is the main work area for adjusting the default parameter values. This chapter introduces the types of controls common to most nodes and explains how to use the various controls to set node parameters.

Overview of the Node Panel ..... p. 6.2
- Parameter Tabs ..... p. 6.3
- Info Tab ..... p. 6.4

Setting Node Parameters ..... p. 6.7
- Using Tuners ..... p. 6.7
- Using Data Entry Fields ..... p. 6.8
- Animating Parameter Data ..... p. 6.9
- Creating Key Frames ..... p. 6.10
- Using Preset Parameter Values ..... p. 6.11

Setting Global Variables ..... p. 6.12

OVERVIEW OF THE NODE PANEL



The area on the left side of the Chalice interface, just below the menu bars, displays the panel associated with the currently selected node. The top portion of the node parameter panel usually includes the same information, icons, and menus, regardless of the node type. These include:

Type of Node

The node type (Disk Input, Blur, etc.) is always labeled in the upper-left corner of the panel.

Node Name

Chalice assigns each new node a unique name by default (e.g., "blur1"). However, you can rename the node at any time. Each node in the current workspace level must have a unique name. (Workspace levels become relevant when you group nodes, as described in chapter 7, "Managing Large Networks.")

Node names must conform to standard UNIX nomenclature (this holds for both IRIX and NT users). Do not use spaces or special characters such as the semicolon (;), backslash (\), ampersand (&), exclamation point (!), asterisk (*), or pipe (|).

To rename a node, key the new name for the node into the "Name:" field on the node parameter tab and press the Enter key. It is best to keep node names short but descriptive.

Tip:

Use the Comment field in the Info Tab to document any pertinent information about the node for subsequent reference. (You can read the comment at any time by clicking and holding the Info button on the node.)

Help Button

The question mark icon in the upper-right corner of the node parameter panel is a context-sensitive Help button. Click on this button to access the section of Chalice Help that describes that particular node type. (For more information about the Chalice online help system, refer to the Help Menu section of chapter 10, "Main Menu Bar.")

Expressions Checkbox

Many nodes feature an Expressions checkbox that enables you to key mathematical expressions (instead of a fixed numerical value) into certain parameters. Some expressions allow you to reference data from other nodes. This feature is described in "Setting Node Parameters."

Key Framer

You use the Key Framer to mark specific frames in a sequence as key frames to animate node parameters. After you set the key frame values, the node will interpolate the "in-between"

frames—the frames that fall between key frames—according to how you have set the parameters for the node. For specific instructions on using the Key Framer, see "Setting Node Parameters."

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 in any grail file. This can save you from having to remember and reassign a complex set of parameter values that you want to reuse. For specific instructions on creating, deleting, and loading node presets, see "Using Preset Parameter Values."

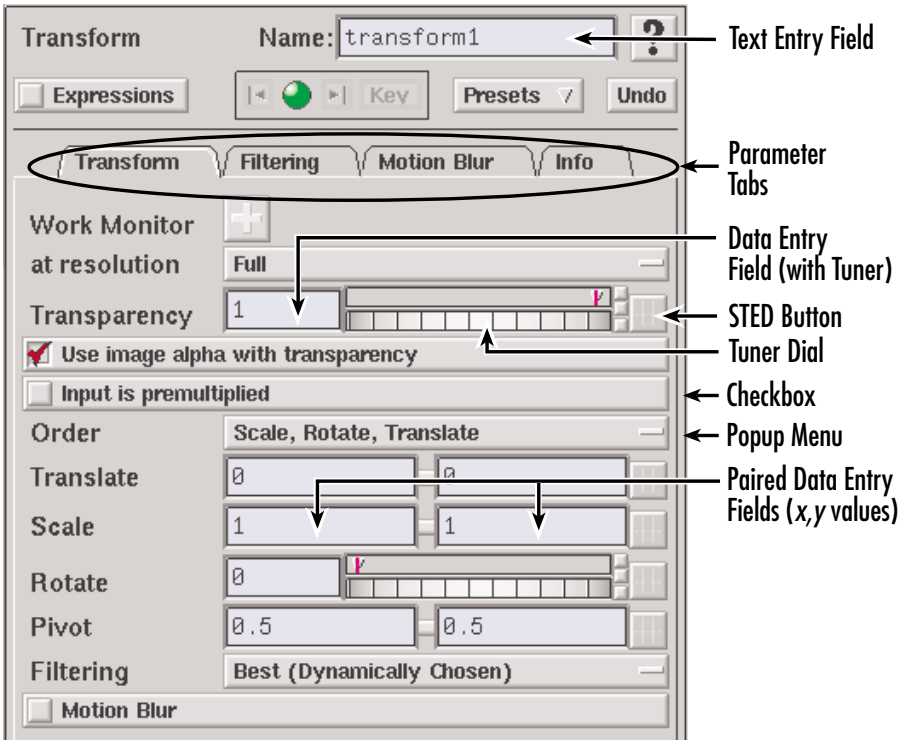
Undo button

Click the Undo button to undo the last change made to a node parameter setting.

Parameter Tabs

The lower section of the node parameter panel includes various parameter tabs that are specific to the node type. At minimum, all nodes include an Info Tab. Most nodes include more than one parameter tab, which you can select to display an additional set of parameters.

The following is an example of a typical parameter tab:



You activate a particular parameter tab in a node panel by clicking on the top of the tab. Then you set the parameters using popup menus, checkboxes, data entry fields, and tuners.

Popup Menus

Click and hold the left mouse button over a popup menu button to display the menu of available choices for that parameter. While still holding down the mouse button, move the cursor through the list of items in the menu and each item will highlight in turn. Lift up the mouse button while the cursor is over a highlighted menu item to select that option. The currently selected option is always displayed on the popup menu button.

Checkboxes

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.

Data Entry Fields

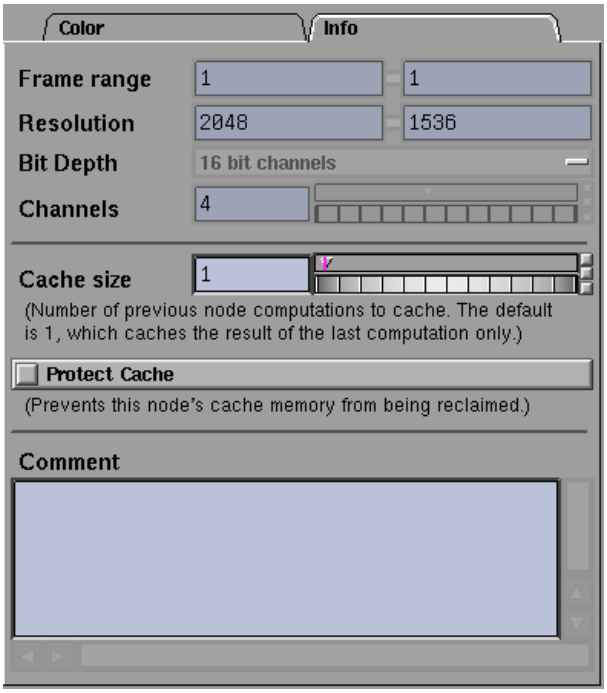
Position the cursor over the field and click the mouse button to key in a specific numerical value for a parameter. Remember to press the Enter key when you are done typing to actuate the entry. See also "Using Data Entry Fields" later in this chapter.

These fields often come in pairs, with the left-hand field representing the X axis value and the right-hand field representing the Y axis value for the parameter. When appropriate, a data entry field for a parameter will be accompanied by a tuner and/or a STED button:

- STED buttons are associated with any field and/or tuner that can be animated, as described in the section on "Animating Parameter Data." The STED button has two icon states: gray indicates that the parameter is not animated, black, that animation has been activated.
- A data entry field is often associated with a tuner dial that enables you to adjust the parameter value in the field by moving the dial rather than typing in numbers. For detailed information, refer to "Using Tuners."

Info Tab

All nodes include an Info parameter tab. The Info tab displays basic information about the node image: frame range, resolution, bit depth, and number of channels. It also includes node caching parameters and a comment field, both of which may be modified by the user.



For nodes that accept multiple inputs such as composite nodes, the Info Tab will display information relevant to the first (top) input to the node.

Resolution

The Resolution parameter displays the X and Y dimensions of the imagery, expressed as a number of pixels. The left field contains the X (width) value; the right field contains the Y (height) value. If you wish to change the spatial resolution of an image, refer to the Scale or Crop node descriptions in chapter 17, "Movement Nodes."

Bit Depth

The Bit Depth parameter displays the bit depth per channel, or pixel type, of the imagery contained in the node, such as 8- or 16-bits per channel, or floating point (32 bit) channels. If you wish to change the bit depth of an image, refer to chapter 14, "Conversion Nodes."

Channels

The Channels parameter displays the number of channels used to compose the imagery in the node. If you wish to change the number of channels in an image, use the Channel Swap node, which is described in chapter 13, "Tool Nodes."

Tip:

The information on the Info tab is also available in a handy popup display when you click and hold down the Info ("i") button on the node itself.

Cache Parameters

The Cache Size and Protect Cache parameters enable you change the default caching behavior for a specific node. Chalice always attempts to cache node calculations in case the data is requested again. When the RAM and disk caches assigned to Chalice get full, however, some data must be overwritten for new processes.

Chalice decides which node's cache is deleted first using an internal priority system based on how individual node data is accessed, such as the node last altered, a node frequently altered, a node with a monitor image in a buffer, or a node that branches (has outputs to more than one downstream node). You can, in a sense, re-weight this priority system by using the node caching parameters.

Note:

For more information, refer to "Memory Usage in Chalice" in chapter 4, and to the description of the Cache Settings panel in the Admin Menu section of chapter 10, "Main Menu Bar."

Cache Size

The Cache Size parameter allows you to adjust the number of consecutive computations that Chalice will cache. The default setting is 1, which indicates that Chalice will cache the last computation only. A setting of 2 would mean that Chalice caches the last two computations, and so forth, up to 10, which is the maximum cache setting.

In most instances it is not necessary to change the default Cache Size setting. However, you may wish to increase the cache size for a particular node if you plan to frequently access node data out of the normal order—for example, when a downstream node uses data from more than one frame (Time Convolve, Time Stretch, or the video transfer nodes, e.g.), or when you are repeatedly accessing two or more different frames downstream, as when you need to compare the effect each parameter change has on the first and last frames in a shot.

Tip:

Be careful not to set the Cache Size any larger than necessary, because high memory consumption by the node cache can slow overall performance.

Protect Cache

This parameter, when checked, specifies that Chalice assign the highest priority to preserving the data cached for this node. It is best used for a node with multiple downstream branches, or for a node whose upstream connections are unlikely to change any further. This can improve performance while you make adjustments to downstream nodes.

> **Tip:**
>
> Do not turn on the Protect Cache parameter for more than a few nodes simultaneously, as this will consume substantial amounts of memory and negatively impact system performance.

Comment

The Comment parameter enables you to document pertinent information about the node by keying a comment into the text entry field.

Reading Comments

Comments are displayed by clicking and holding the Info (i) button on the node itself.

Creating and Editing Comments

To key in a comment, click the left mouse button in the Comment field and begin typing. If you wish to enter a carriage return, hold down the Alt key on the keyboard and press the Enter key. (Otherwise, the text will stream past the visible edge of the field to the left.) The Comment field scroll bars can be used to access portions of the text that extend beyond the field frame.

Alternatively, you can use a text editor to display and edit this text: with the cursor in the text field, press **Alt-e** to bring up the default text editor for your system (e.g., Jot or Notepad).

> **Note:**
>
> Chalice also includes a Comment node, which is described in chapter 13, "Tool Nodes." Comment nodes may be used to create a conspicuous marker in the workspace to document any aspect of the shot you are creating.

SETTING NODE PARAMETERS

This section outlines the procedures to follow to use several parameter controls that are common to most nodes:

- Tuners
- Data entry fields
- STED buttons
- Expressions checkbox
- Key framer
- Node presets menu

Data entry fields, and data entry fields with an associated tuner dial (which are called "tuners" for short), are used to adjust most parameter values. The descriptions that follow include tips and shortcuts for using these parameter controls, including how to enable animation of parameter values over time.

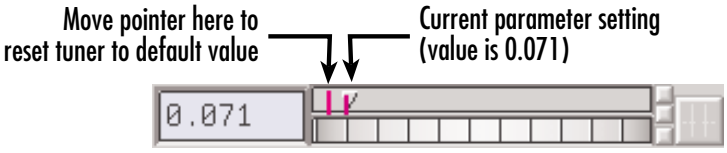
The expressions checkbox, node presets menu, and the Key Framer are all used to modify parameter data in a specific manner, as described later in this chapter.

Using Tuners

The Tuners available on many parameter tabs offer two ways to set parameter values: by adjusting the tuner dial or by keying a specific value into the tuner's data entry field. A data entry field is associated with every tuner. When you adjust the tuner, the corresponding numerical value in the data field to the left of the tuner will update accordingly.

The tuner dial has two controllers: a triangular pointer that slides back and forth across the top half of the tuner and a vernier dial on the bottom half. To adjust the tuner value with the greatest precision, click and drag the vernier dial back and forth.

The small triangular pointer indicates the current dial setting. The appearance of this pointer changes depending on the setting. To set a tuner to the default value, align the pointer with the pink vertical bar. If the current tuner setting is any other value, it is indicated by a blue vertical bar. (For some tuners, values can be set that exceed the intended range; in such cases, the pointer turns red.)



The tuner includes three small square buttons, arranged vertically, to the right of the tuner. These buttons enable you to adjust the sensitivity of the tuner.

- Each click of the top button increases the sensitivity of the tuner (usually by a factor of 10).
- Each click of the bottom button decreases the sensitivity of the tuner (usually by a factor of 10).

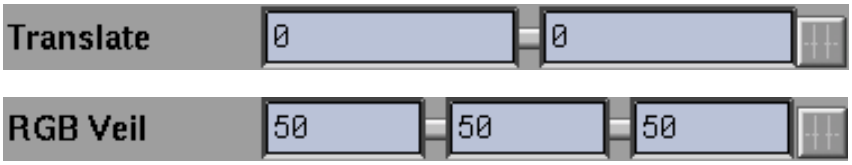
- To determine (and adjust, if necessary) the sensitivity factor of the tuner, click the middle button to replace the dial with a text field containing the current sensitivity factor (e.g., "10"). You can then key a new value into this field, if you desire. (Click the button again to replace the text field with the dial.)

Tuners usually feature an associated STED button. This indicates that the parameter values can be animated over time, as explained in the section on "Animating Parameter Data."

Using Data Entry Fields

Data Entry Fields are used throughout Chalice to enter numeric values and/or text, depending on the nature of the parameter. A field that is associated with a tuner dial will always take numerical values, while independent fields (without tuners) can be used for entering text or data.

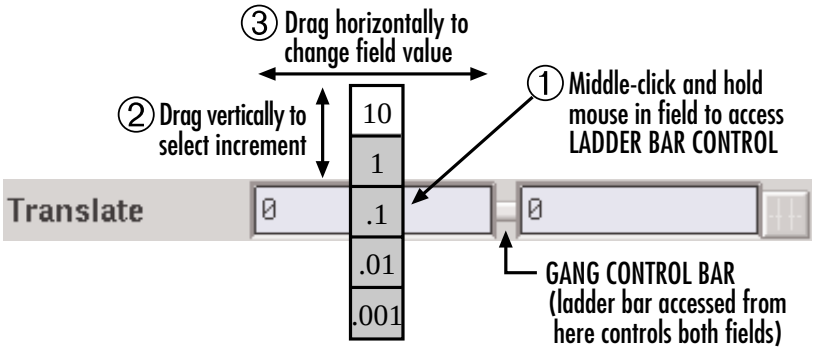
Data entry fields often appear in pairs where the left field represents an X coordinate and the right field, a Y coordinate. Fields may also come in a row of three, where (moving from left to right) they control values for the Red, Green, and Blue channels respectively.



Shortcuts for Data Entry

Middle-click and hold the mouse button in a data entry field to display a "ladder bar" control—a vertical list of increment values you can use to select the magnitude of the changes to the value in the field. First, drag the mouse up or down to select the increment value you want for the adjustment, and then drag left or right on the same plane to decrease or increase the value in the field by the selected increment.

For example, a ladder bar might offer a vertical list of 100, 10, 1, .1, and .01. If the value in the field is 1 and you drag the mouse up to the "10" box in the ladder bar, when you drag left and right at that level the value in the field will increase or decrease by 10s: from 1 to 10 to 20, etc.; or from 1 to -10, to -20, etc.



For associated data entry fields (such as a row of three fields representing the Red, Green, and Blue channels respectively), there may be a gang control you can use to adjust all values equally. Middle-click and hold any gang control bar in the field row to use the ladder bar. Then as you drag horizontally to adjust the value, the values in all three fields will increase or decrease in equal amounts.

Animating Parameter Data

Animating a parameter refers to varying its value over time. In default mode, the value you enter for a parameter applies to every frame of the input sequence. If you change frames and reset the value, the new value replaces the old for every frame.

Using STED Buttons

Whenever a STED button is associated with a field or tuner, the values for that parameter can be animated. First, you enable animation by clicking the STED button:

STED Button: Animation Enabled:

Once you have changed the STED button from gray to black, Chalice will accept a different value at each frame for the associated parameter. To disable animation of a parameter, Alt-click the STED button to return it to its default (gray) state. When you double-click the STED button, the STED editor window opens to display the parameter value as a curve across a graph representing the input frames.

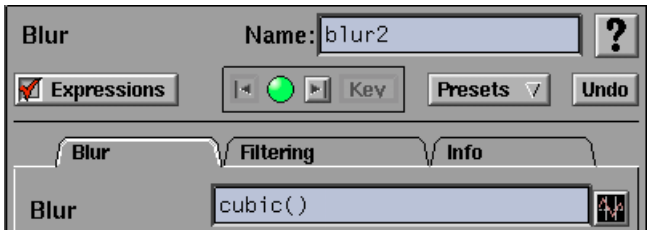
The most basic way to animate a parameter is to set a value for the first frame, and then go to the last frame and set a new value. Chalice will interpolate between these frames to create the intermediate values. For more control, however, the following methods are available, depending on the type and extent of control you need:

- You can turn on Expressions to enter an expression into the parameter that controls the value at each frame.
- You can use the Key Framer to tag specific frames as key frames.
- You can open the STED editor window to adjust values with very fine control, including the ability to apply various function curves.

Use of the Key Framer and the Expressions checkbox are described below. To use the STED editor window, however, refer to chapter 9, "Space-Time Editor (STED)," which explains the use of this powerful tool in detail.

Enabling Expressions

When you click the Expressions checkbox, any tuner dials on the parameter tab disappear and their associated data entry field lengthens to accommodate the entry of an appropriate mathematical expression into the field to control the processing performed by that particular node parameter.



Tip:

When an expression has been used to animate a parameter value, you can use the Expressions checkbox to toggle back and forth between a view of the expression itself and a view of the value produced by the expression at the current frame.

For more information on writing and using expressions, refer to Appendix B, "Chalice Expression Language," or to the applicable node descriptions.

Creating Key Frames

A key frame in Chalice is a marker that delineates the start or end point of a segment that animates a specific parameter in an image sequence. In this sense, all image sequences have at least two key frames: the first frame and the last frame.

You can adjust the values of the first and last frame to control the start and end values of the interpolation for the parameter. And you can add key frames to a sequence to split the original waveform into a series of smaller segments, each of which can be adjusted individually for finer control of the animation.

You can control the animation of parameters using the Key Framer in conjunction with the Frame Controller and the available parameter controls. (This procedure is described below.) Alternatively, you can use the Chalice Space-Time Editor (STED), which offers more options, such as the application of various functions to segments. This method is described in chapter 9, "Space-Time Editor (STED)."

Note:

To remove a segment, you must use the STED. New segments you create by adding key frames cannot be deleted using the Key Framer panel.

Key Framer Device

The Key Framer device in the top of the node panel displays a light that changes color to indicate current status: move the cursor over the button to display a Help Tag that explains the current state of the Key Framer. The "Key" button changes to its active state whenever a key frame can be set. Click the Key button to make the current frame a key frame.

Key Frame Navigation

Once you have created key frames, you can navigate directly from key frame to key frame at any time using the forward or back arrows in the Key Framer. This will update both the animated parameter values in the node panel and the monitor display, if a monitor for the node is open.

Using the Key Framer with the Frame Controller

For details on using the Frame Controller, see "Previewing Node Output" in chapter 5.

1. **To set a key frame, first you must click the STED button once to enable animation for the parameter (you do not need to open the STED window).**
   The STED button will change from gray to black to indicate that parameter animation has been enabled.
2. **Then use the Frame Controller to go to the first frame that you want to specify as a key frame.**
3. **Change the value of the parameter associated with the STED button you clicked.**
   The "Key" button on the Key Framer will become active.
4. **Click on the Key button to make the current frame a key frame.**
5. **Repeat this process until you have designated all the key frames you want.**
6. **Use the Previous and Next arrows on the Key Framer to review and adjust the key frame parameter settings, if desired.**

Using Preset Parameter Values

The Node Presets menu allows you to save and load parameter settings for a node. Once you are satisfied with the parameter values you have assigned to a particular node, you can save the parameters and then load these preset values into any node of the same type.

For example, if you have several image sequences that require the same color correction, once you are satisfied with the parameter settings for the Color Correct node you can save the settings and load them into other Color Correct nodes as you correct subsequent sequences.

Node Presets Menu

To save or load a preset, click and hold the Presets menu button on the node parameter panel to access the Presets popup menu.

Chalice will display a list including the following four options:

- User Defaults
- Factory Defaults
- Save As User Default
- Save As…

Saving Node Presets

The Presets menu provides two ways to save the current node parameter settings as defaults:

- **Save As**: This option enables you to save the current parameter settings as a "custom preset," using a unique preset name. The Save As option will not overwrite the User or Factory Default settings. When you select Save As, Chalice will display the "Enter Preset Name" dialog box. Choose an appropriate name for your current parameters and press "Accept." This new preset will appear the next time you pull down the Presets menu for any node of the same type.
- **Save As User Default**: This option saves the current parameter settings as the new User Default for the current node type. Each time you select this option, the current settings overwrite any previous User Default settings.

Restoring Factory Defaults

Once a User Default has been created for a node type, it becomes the permanent default, replacing the Chalice Factory Defaults, until the User Default setting is deleted or redefined. To delete a User Default and thus restore the Factory Defaults, hold down the Alt key while selecting User Defaults from the menu.

Loading Node Presets

The Presets menu provides three options to load preset parameter values into a node:

- Select the unique name of a **Custom Preset** (created using Save As).
- Select the current **User Defaults** settings (created using Save As User Default).
- Select **Factory Defaults** to see the Chalice default settings.

Deleting Node Presets

To delete a node preset, hold down the Alt key while selecting the preset. A dialog will ask you to confirm that you want to delete the preset.

SETTING GLOBAL VARIABLES

A global variable is a name that is associated with a value, where the value can be changed at any time. A global name, preceded by the "$" to identify it as a global, can be keyed into any appropriate data entry field and Chalice will use the current value defined for the global.

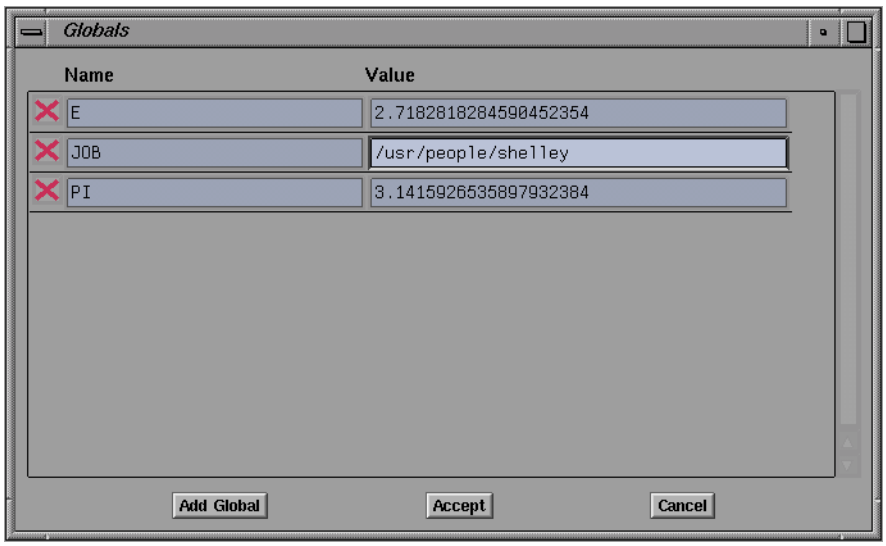
Using a global in a data entry field, rather than a specific value, has several advantages. A global name can be an effective shortcut (typing "$PI," for example, instead of "3.141592…"). And when a global represents a value that must be changed at some point, only the definition of the global value need be changed, not all the specific instances of it in numerous nodes.

Editing Globals

Chalice provides a Globals window which lists each global variable and its current definition. You can use the Globals window to add, delete, and change the definition of globals. There are three default global variables which cannot be deleted: JOB, which can be redefined, and the constants PI and E, which cannot.

To open the Globals window, select "Edit Global Variables..." from the Chalice Edit menu. This will allow you to

- determine the current values of global variables in the list.
- redefine the values of global variables in the list.
- add your own global variables to the list.
- delete any user-created global variables.



Redefining Globals

With the exception of PI and E, all global values may be changed. To change a global, simply key a new value into the "Value" entry field.

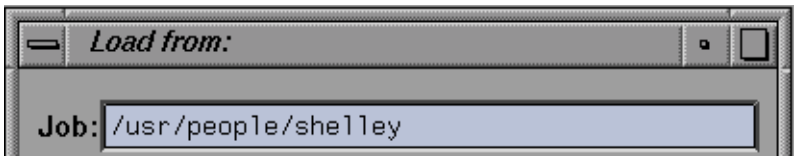
Global variable settings are grail-file-specific. That is, any changes you make to global values will not affect the global values in other existing grail files.

Note:

A global variable can be redefined temporarily when batch rendering with chal by using the -D [name definition] option (see appendix A, "Chal," for more information on rendering options).

$JOB

The default location of the Job variable is the directory from which you started Chalice. You can redefine Job to fit your current project.



Adding Globals

To create a new global, click the Add Global button in the Globals window. Chalice will add a new blank entry to the list. Then you can type a name for the new global into the "Name" field, and a definition into the "Value" field. Be sure that the name for the new global follows standard UNIX naming conventions and that all letters are uppercase.

Click the Accept button to actuate the new global and close the Globals window.

Deleting Globals

All user-created globals can be deleted. Simply click the red "X" next to the global you wish to delete to remove it from the list.

Using Globals in Data Entry Fields

To use a global variable, type the name of the global into any appropriate data entry field, preceded by "$." For example, you could type "$JOB" into the path field of a Disk Output node to represent the directory path defined in the "Value" field of the Job global.

If you later want to change the location to which image files are written by the Disk Output node, you simply redefine the Job variable by keying a new directory path into the "Value" field of the Job global in the Globals window list.

Note:

Redefining $JOB automatically triggers a rescan of all Disk Input nodes. For more information, refer to the node description in chapter 12, "I/O Nodes."

7 MANAGING LARGE NETWORKS

This manual often illustrates a particular topic with a very simplified node network, but in actual production, of course, your networks are likely to get very large and complex. This chapter is intended to highlight a number of shortcuts and other features that can make the job of building and changing a large network faster and easier.

Tip:

You may also want to refer to chapter 4, "Key Concepts," which covers memory usage in Chalice and facility-wide sharing of Chalice node data.

Shortcuts ..... p. 7.2
Finding Nodes Fast ..... p. 7.5
Grouping Nodes ..... p. 7.7
Creating Custom Networks ..... p. 7.11

SHORTCUTS

This section highlights some of the ways you can set up your file initially to save steps later, as well as a number of shortcuts for getting around in large networks and for adjusting node parameters.

Check Preference Settings

Chalice lets you change default settings for a number of features by using the panels accessed from the Preferences menu (see chapter 11, "Setting Preferences"). Say, for example, that you regularly use the Inspect panel in monitors—you can change the default monitor interface so that the Inspect panel appears automatically, rather than having to request it each time you open a monitor. You can change similar defaults, such as the fps playback rate in flipbooks and the size of new nodes. Another thing that can save a lot of time is to set the default for premultiplication status to match the type of image data you will be using most. This default controls the initial state of all premultiply buttons in nodes such as Mcomp.

If you are using proxy files, check the default monitor resolutions in Proxy Preferences. If you make sure that the default resolutions for Low and Medium monitors in Proxy Preferences match the size of your proxy inputs, you avoid having Chalice scale the proxy input image up or down a little every time you bring up a low or medium monitor.

Add Comments

One of the simplest yet most useful features is the ability to give nodes unique, descriptive names and to add comments to any node. Taking the extra time initially to add comments where appropriate can save you a lot of time in the long run. Comments are especially useful if anyone else may ever have to modify your grail file, or if you have to go back and make changes to a particular shot that you haven't looked at in days or weeks.

The easiest way to access comments is to click and hold the "i" button on the node itself. A popup menu will display any comment along with other information about the current node image, such as resolution, number of channels, and bit depth.

The "i" button can be a real timesaver, as you can go from node to node and get basic information without having to click the title bar for each node to make it active and then click the Info tab in the node panel.

Getting Around in the Workspace

The Chalice workspace can be as large as necessary to accommodate all the nodes in a network, but the portion of the workspace visible at any one time cannot. To get the maximum size visible workspace, you can toggle in and out of "Full Layout" mode using the Alt-f key combination. In Full Layout, the node panel and radar map disappear to make more room to display the network (the menu bars are still visible).

If you need to connect two nodes that are so far apart that both of them cannot be visible in the workspace at the same time, use the scroll bars or radar map to navigate from one node to the other to make the connection. (Dragging in the workspace to move will remove any connector line attached to your cursor.)

Optimizing Node Display

To make more room in the visible workspace you can also make some or all nodes smaller, turn off the icon display in the node, and collapse related nodes into groups (see the section of this chapter on Grouping Nodes).

You can change the default display of new nodes; however, you can also change any or all nodes already in the workspace whenever necessary, without resorting to the individual node display commands in the local node menu of each node.

To make the same change to multiple nodes, select them and use the appropriate command in the Layout menu. For example, you can select all nodes (using the command in the Edit menu) and choose the "hide icons" command from the Layout menu. Then you can select certain key nodes and use the "show icons" command to turn their icon displays back on. The same holds true for changing the size of nodes; just select a new size from the "change size" submenu in the Layout menu.

Turning off icon display for nodes that don't need it is especially important if you want to specify that node icons auto update (in the Node Preferences panel). This can be a great convenience, but in a large network it can consume too much memory if every node is updated unnecessarily.

Zoom Mode

Even when you have optimized node display, a large network can still extend well beyond the visible workspace. In this case, you can zoom in or out on the network by going into Zoom mode. Use the Alt-z command to toggle in and out of Zoom mode. Once you are in zoom mode, hold down the Alt key and use the middle mouse button to drag up and down in the workspace. This will zoom you in and out. Node display is simplified in Zoom mode, but nodes are still selectable.

Working with Nodes

In addition to holding down the Shift key as you select nodes, which can get tedious, there are shortcuts for selecting multiple nodes: The Edit menu contains a "Select All Nodes" command to select every node. And you can Alt-drag or Control-drag in the workspace to draw a bounding box around a group of nodes to select them. Or you can use the Local Node menu on a node to pick all upstream nodes or all downstream nodes (using the "Pick All Ancestors" and "Pick All Descendents" commands).

Some Chalice nodes accept multiple inputs. For a node with many inputs, you can right-click on each connector individually to find out which node is connected to each input. This popup display will also tell you what the connector is for, as when it is an optional input for a control image.

Node Creation

You don't need to go to the node menu bar every time you want to create a node. Middle-click and hold the input or output connector of a node to see a popup menu of all compatible nodes in alphabetical order. When you select one, you will see that a connector line has been created from the node that ends in a white box outline. Click in the workspace and this box turns into the type of node you selected from the menu, and it is already connected to the node it was created from.

You can also replace a node in one step: Middle-click the local node menu button of the node you want to replace. The same node selection menu will appear as when you middle-click a node connector. In this case, however, when you choose a node type from the menu it will replace the current node.

In large networks with dozens of input elements, you often perform the same set of operations on multiple images. To shorten this process, don't recreate the same nodes over and over. You can copy or clone an existing node or group of nodes. Or you can add a group of

nodes to the Custom menu, from which you can access them whenever you need to without having to find the original nodes and clone them.

The same principle applies to individual node parameter settings. You can create presets for any node type to quickly access a specific set of parameter settings.

Working with Node Data

Monitors

Be sure to familiarize yourself with all of the options available in Chalice monitors (see chapter 8, "Monitors and Flipbooks"). For example, you can navigate from frame to frame in a monitor using arrow keys, rather than clicking a button on the Frame Controller. If you need to navigate directly from key frame to key frame, by the way, you can use the arrow buttons on the Key Framer device in the node panel.

You can use the middle mouse button to drag the image around in the monitor. If so, you can also use the handy F9 hot key to snap the image back to center.

If you are using a work monitor, be sure to check the node documentation for shortcuts specific to that node. For instance, you use the arrow keys with modifier keys to change translation, rotation, and scaling values in small increments in the Transform work monitor.

We also recommend that you learn to use the image buffers to your advantage (they are also described in chapter 8). You can use options such as Buffer Persistence and the ability to bind a buffer to a node to save a lot of time waiting for re-cookss.

Parameters

As described in "Using Data Entry Fields" in chapter 6, there are a several shortcuts to changing parameter values in fields. You can middle-click in any field to use the ladder bar shortcut for data entry. And the associated gang control lets you use ladder bar to set multiple field values simultaneously.

Also, whenever you see the Cascade button (downward-pointing arrow) next to a column of parameter fields, you only have to enter a value in the top field and the Cascade button will copy it to the other fields in the list.

You can use the Resolution popup menu wherever it is associated with a pair of fields to select a resolution from the menu list rather than keying in numbers every time.

If you are going to be animating a lot of parameters, we recommend that you take the time to review chapter 9, "Space-Time Editor (STED)"; it has helpful shortcuts that aren't obvious otherwise.

A final tip for setting parameters in large networks is to use the data channel functions whenever appropriate—rather than entering a specific value for a parameter, you can specify that another node's parameter value be used. Then you can adjust the one node parameter and all the others will use the new value. These functions are explained in appendix B, "Chalice Expression Language." You don't have to be a computer programmer to figure out how to use many basic expressions.

FINDING NODES FAST

A grail file for a complex network can get very large. In most cases, the fastest way to find a particular node is to use the node search function, which enables you to search for nodes by name using literal strings or cshell regular expressions.

If you are looking for a node whose image is cooked into an image buffer, or you want to find nodes in a specific branch of a network, however, try these shortcuts:

Image Buffer "Find This Node" Command

Often, you want to find a node that has its image data stored in one of the nine Chalice image buffers. Each image buffer is represented by a button in a strip that is located just below the Frame Controller on the main interface.

Right click and hold any of the buffer buttons to display the buffer popup menu:

- If no data has been cooked into the buffer, the menu will read simply "Unattached."
- If the buffer does contain image data, however, the popup menu will display the name of the node, along with other basic information.

The item in the popup menu that shows the name of the node, also says "Find this node." If you select this item from the menu, the node will be selected and the workspace view will be adjusted to show the node, if it was outside the visible area.

Local Node Menu "Pick All" Commands

In a large network you will often have dozens of Disk Input nodes, each starting a separate network of operations that only meet downstream at a composite node. In this case, you often find that you need to go to the specific Disk Input that is feeding a filter node, or that you want to quickly identify the operators connected to a particular Disk Input.

You can use the "Pick All Ancestors" command on the local node menu of the filter node to highlight all upstream nodes. This trail of nodes will also highlight in the Radar Map, making it easy to find and select the Disk Input.

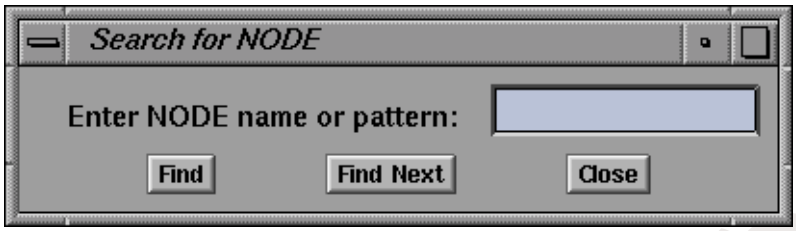
Conversely, you can use the "Pick All Descendents" command at a Disk Input node to highlight all nodes connected to it downstream.

Tip:

Using the Pick All Descendents command at each Disk Input node in turn is a good way to get an overview of the dataflow when you need to familiarize yourself with someone else's grail file.

Performing Node Searches

The Search Nodes command is accessed from the Chalice Edit menu. It opens a panel from which you perform the search.



Find

To find a node, type the name of the node in the text entry field of the Search panel, and then click the Find button. All nodes that match the search criteria will be highlighted (both in the workspace and in the radar map).

Find Next

You can click the Find Next button to highlight each match in turn. If any of the found nodes are beyond the visible workspace, the Find Next command will bring them to the center of the workspace.

Note:

The Search Nodes command works at the current workspace level only. It will not find a node that matches the search criteria if the node is inside of a Group node. To search within a Group node, push into it first and then perform the search.

Using Metacharacters

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 wildcard characters (metacharacters) in the search string:

*	matches any string of characters, including none
?	matches any single character
[ ]	matches any single character you type within the brackets

For example, typing:

image?_scaled would find nodes named *image1_scaled, image2_scaled,* etc.
image* would find all node names starting with the word image: *image, image1_scaled, image_cropped,* etc.
*image would find all node names ending with the word image: *scaled_image, cropped_image,* etc.
*[1] would find all node names ending in the number 1: *image1, image101, cropped_image1,* etc.

GROUPING NODES

Chalice provides the ability to group multiple nodes into an organizational unit, which is known as a group node. The main function of Group nodes is to simplify large, complex dataflows, making them more manageable and easier to debug.

Often, a similar series of operations must be carried out on multiple image elements. One way to simplify this task is to group together the nodes that perform the operations and clone them for each element.

Creating a Group

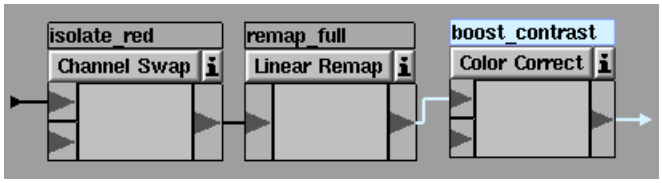
Unlike other node types, Group nodes are not created by choosing one from a Node menu. Rather, you define and create a Group node by selecting one or more nodes in the workspace and choosing the Group command (Alt g) from the Group menu.

The group node appears as another node in the workspace, however group nodes never display a frame icon (just as you cannot bring up a monitor from a group node), as they are simply containers for other nodes and do not process data in and of themselves.

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, nodes outside the group).

Push In

To view the contents of a Group node, select the Push In command (Alt +) from the Group menu. The Chalice workspace changes to display the nodes (the children) within the Group node (the parent):



Pull Out

Select the Pull Out command (Alt -) to view the next higher level—the level that contains the Group node you pushed into.

Ungroup

To ungroup a Group node, select it and choose the Ungroup command (Alt G) from the Group menu. This will eliminate the Group node, placing the nodes it contained back in the current workspace.

> **Note:**
>
> Ungrouping a Group node does not delete its contents. To delete a Group node, along with the nodes it contains, select the Group node and use the Delete Node(s) command in the Edit menu, or press the Del key.

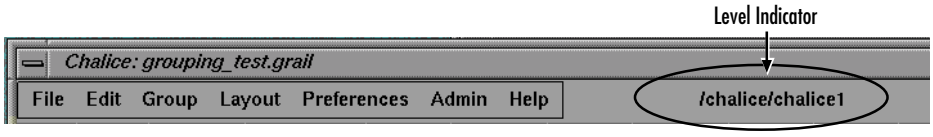
Working with Group Nodes

Key points to remember when working with Group nodes include the following:

- 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.
- Group nodes can be nested; that is, new Group nodes can be created within existing Group nodes.
- Group nodes can be copied, pasted, cloned, deleted, added to the Custom menu, and otherwise treated like other nodes.

Level Indicator

The Chalice workspace displays one level at a time. The current level is indicated on the Main Menu bar:



The "top" level (the level you are in when you open a grail file) is always "/chalice/chalice1." If you create a Group node named "matte_fix" at this level and push into it, the Level Indicator will read "/chalice/chalice1/matte_fix":

Node Names

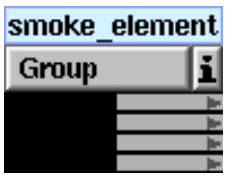
Group nodes are named just like other node types. By default, Chalice names the first Group node you create "group1" (the second would be named "group2" and so on).

Chalice requires that every node in the workspace have a unique name. More precisely, this means every node at the current level. Therefore, Chalice will allow a node named "color1" at the level of "/chalice/chalice1" and at the level "/chalice/chalice1/group1." However, if you cut or copy the "color1" node from the group1 level and paste it in the top level, Chalice will rename the node to a unique name ("color2," for example).

Exporting Connections

When you create a connection to an external node from the input or output of a child, you are "exporting" the connection. When a new Group node is created, Chalice automatically exports any connections from nodes being grouped to nodes outside the group.

Each input connector that has been exported from a child is represented by a corresponding input connector on the Group node itself. And each output connector exported from a child is represented by a corresponding output connector on the Group node. This example shows a group node with no input connections and four output connectors:



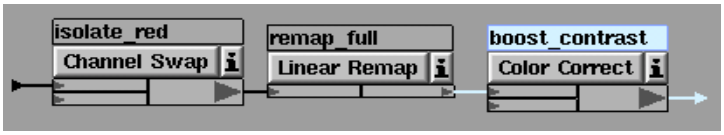
You can then use the connectors on the Group node as if they were the source connectors on the children inside the Group node: you can connect them, disconnect them, and reconnect them to different external nodes.

To export a connection, Alt-click the left mouse button on the input or output connector of a child node. Chalice creates an export flag, which is a short connecting line with a square at the end. The square indicates that the connection has been exported, but not yet connected to an external node.

If you pull out, you will see that the Group node now has a corresponding connector, which works like any other node connector.

Once you connect the input or output to a node, push back into the Group node and you will see that the corresponding export flag is now an arrow (instead of a square). This indicates that the connection has been exported and is connected to an external node.

View of Node States Within a Group Node



Terminator (square) indicates that output connector has been created on group node

Arrow indicates that output is connected to node outside of group

Note:

To "un-export" a connection, Alt-click the connector again. Chalice will remove the exported connection from the node.

Connector Info

You can use the right mouse button to click any input connector of a Group node to find out which child it leads to.

You can use the right mouse button to click the input connectors of child nodes to display connection information. This allows you to determine what external nodes a child is connected to without pulling out of a Group node.

Info Button

Click the Info button on a Group node for the following:

- **Full Name** displays the full pathname of the Group node, for example, "/chalice/chalice1/group1."
- **Inputs** displays the total number of connections available to node inputs within the Group node.
- **Outputs** displays the total number of connections available from node outputs within the Group node.
- **Children** displays the number of nodes that compose the Group node.
- **Comments** displays any text that may have been entered in the Comment field of the Info Tab.

Group Node Panel

When a Group node is selected in the workspace, a corresponding group node panel appears. This panel provides Inputs and Outputs parameter tabs, which list any or all imported or exported connections. (A Group node can have no inputs and/or outputs or multiple inputs and/or outputs.)

Inputs Tab

Chalice creates an entry in the Inputs Tab for every connection into the Group node. Each entry lists the name of the child node to which the connection leads and the number of the child node input being referenced.

Each entry also includes an Up Arrow button and a Comment field for that input:

- Use the Up Arrow buttons to change the order of the inputs on the Group node.
- Type any text you wish into the Comment field. The comment will be displayed, along with other connection information, when you use the right mouse button to click the corresponding connector on the Group node.

Outputs Tab

Chalice creates an entry in the Outputs Tab for every connection out of the Group node. Each entry lists the name of the child node from which the connection leads and the number of the node output being referenced.

Like the Inputs Tab described above, the Outputs Tab includes an Up Arrow button and a Comment field for every output connection.

Info Tab

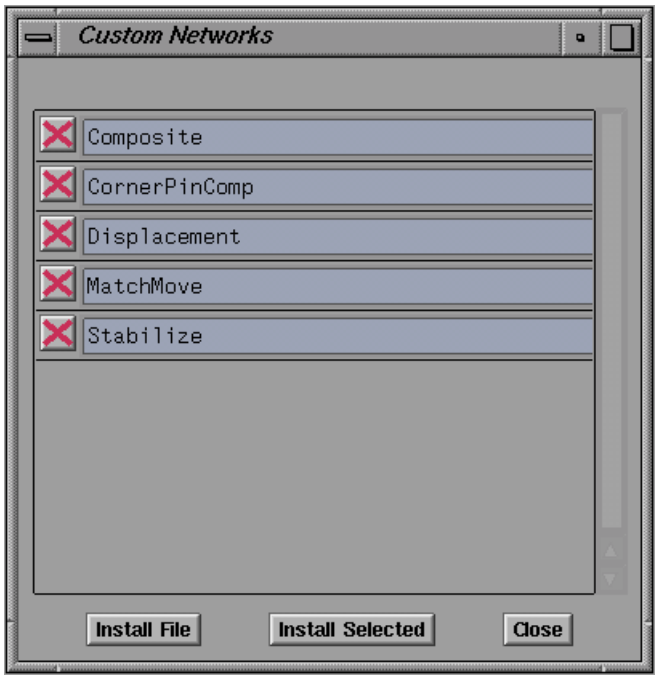
The Info tab of a Group node contains a Comment field only. You can use the Comment field to enter any information you wish about the Group node. This information will be displayed whenever you click the Info button on the Group node itself.

CREATING CUSTOM NETWORKS

In addition to the other menu categories, the Chalice Node Menu includes an additional pulldown menu: Custom. The other pulldown menus contain predefined lists of nodes, grouped by related functionality. The content of the Custom networks menu, however, is user-defined.

You add nodes, or entire node networks, to the Custom menu based on your needs. Take, for example, a situation in which you regularly use the same network of nodes to perform a common operation. You can select all of these nodes and add them to the menu as a single item. When you access this item from the menu, Chalice places the entire node network into the workspace, with all connections and parameter settings intact.

You use the Custom Networks panel to add items to or remove items from the Custom menu. This panel is accessed by selecting the "Edit Custom Networks…" item from the Chalice Edit menu.



Chalice saves any custom networks you create to the directory path specified by the CHALICE_CUSTOM_PATH environment variable.

The default location is /usr/grail/chalice2.0/support/custom. (On NT systems, replace "/usr/grail" with "C:\Program Files" or the equivalent on your workstation.)

Adding Custom Networks to the Menu

You can add any node or group of nodes in the currently open grail file to the Custom menu. You can also add an entire grail file.

To add all currently selected nodes to the menu, follow these steps:

1. **Select the node or group of nodes you wish to add to the menu.**
2. **Select the "Edit Custom Networks…" menu item from the Edit menu.**
   Chalice will display the Custom Networks window.

3. **Press the Install Selected button.**
   Chalice will display the "Enter Name" panel.

4. **Key in a name for the custom network and press the Accept button.**
   Be sure to follow standard UNIX naming conventions—see step 3 of "Saving a New File" in chapter 5 if you need more information.

   Once you press the Accept button, Chalice closes the "Enter Name" panel and adds the new item to the list in the Custom Networks window.

5. **Click the Close button in the Custom Networks window.**
   Chalice closes the window and the new custom network is now installed in the Custom menu.

To add the contents of a different grail file to the Custom menu, follow these steps:

1. **Select the "Edit Custom Networks…" menu item from the Edit menu.**
   Chalice will display the Custom Networks window.

2. **Press the Install File button.**
   Chalice will display the "Install Custom Grail File" dialog box. This is a standard Chalice file dialog.

3. **Double-click on the name of the file you wish to designate as a Custom menu item, navigating the directory structure as necessary to locate the file.**
   If you need more information about using the navigation tools in this dialog, see step 2 of "Opening an Existing File" in chapter 5.

   Chalice closes the "Install Custom Grail File" dialog box and adds the new item to the list in the Custom Networks window.

4. **Click the Close button in the Custom Networks window.**
   Chalice closes the window and the new custom network is now installed in the Custom menu.

Removing Custom Networks from the Menu

You can delete items from the Custom menu at any time by using the Custom Networks panel that is accessed from the Chalice Edit menu.

Simply press the red "X" next to any item in the list and Chalice will remove it from the list, and from the Custom menu.

Note:

If an item you remove from the menu was created from an existing grail file, the original file will not be deleted, only the copy stored with the other items in the Custom menu.

Custom Networks Shipped with Chalice

Chalice ships with a number of custom networks, including annotated grail files that containing example networks of composite operations and other tasks. Each features a different configuration and provides explanatory notes that can be accessed by clicking and holding the Info buttons on the nodes.

8 MONITORS AND FLIPBOOKS

A **Monitor** is a window that you can open at various sizes to display imagery, one frame at a time, at its current stage of processing in the network. A **Flipbook** is a window that allows you to play an animated sequence of frames at the current stage of processing.

You can access a Monitor or Flipbook from any node in the network (except for Exec and Network nodes) by choosing it from the Local Node Menu available on each node.

This chapter is divided into two parts. The introductory section of part 1 will give you an overview of the range of monitor functionality. Subsequent sections will explain the various monitor functions in detail. Part 2 includes covers creating, playing, and saving flipbooks.

Using Monitors ..... p. 8.2
- Opening a Monitor ..... p. 8.2
- Updating the Monitor Display ..... p. 8.3
- Setting Monitor Display Options ..... p. 8.3
- Monitor Modes ..... p. 8.3
- Using the Display Buffers ..... p. 8.4
- Performing A/B Wipes ..... p. 8.5
- Using ROI to Update the Monitor ..... p. 8.6
- Using the Monitor Toolbar ..... p. 8.7
- Using Monitor Presets and Display LUTs ..... p. 8.9
- Using the Inspect Panel ..... p. 8.10
- Using the Monitor Popup Menu ..... p. 8.10
- Using the Utility Monitors ..... p. 8.11

Using Flipbooks ..... p. 8.14
- Creating a Flipbook ..... p. 8.14
- Using the Flipbook Controls ..... p. 8.15
- Using the Flipbook Popup Menu ..... p. 8.16

About Chalice Work Monitors

Some nodes feature work monitors which are accessed by clicking a plus icon (+) on the node parameter panel, instead of from the Local Node Menu.

These monitors offer specific tools or displays, based on the functionality of the node. For example, the Track node has a work monitor in which you can interactively position tracking points and view tracking paths.

This chapter describes the Chalice node monitor. For more information about the functionality specific to a work monitor, refer to the description of the node from which it is accessed.

USING MONITORS

You can use Monitors to evaluate an image at the current node. You can then adjust the various node parameters accordingly, modifying the effect until you are satisfied. Monitors are an essential tool for the successful creation and debugging of an image.

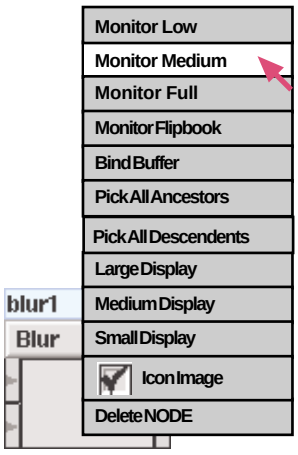
Chalice node monitors enable you to

- display multiple node monitors simultaneously
- use a single monitor to access imagery from multiple nodes
- update the monitor display as you modify parameters
- update only a selected region of the image
- display individual channels of an image
- perform A/B wipes between two images from different nodes, different frames, or before and after a modification
- access waveform, histogram, and related data for the current image
- access pixel-level data for the current image
- optimize the display for the type of image being viewed

Opening a Monitor

To open a Monitor from the Local Node Menu:

1. **Click on the Local Node Menu button for the node you wish to view.**
   Chalice displays a popup menu for as long as you hold down the mouse button:



2. **Select the size of the Monitor you wish to display.**
   You can choose Monitor Full, to display the image at its full resolution, whatever that may be for the current node. Or, you can select Monitor Medium or Monitor Low, which open the monitor at a fraction of full resolution.

   Chalice opens a Monitor window that displays the image at the current frame, as specified in the Frame Controller.

By default, the Monitor window opens with the Monitor Toolbar showing. The Toolbar enables you to access a number of display options. You can change the default Monitor interface configuration by accessing the Monitor Preferences panel from the Chalice Preferences menu. For more information, refer to chapter 11, "Setting Preferences."

The default size for Monitor Medium and Monitor Low is one-half and one-quarter of full resolution, respectively. These values can be changed in the Proxy Preferences panel.

Updating the Monitor Display

The image in any Monitor can be updated every time you make a change in a parameter setting or specify a new frame in the Frame Controller:

- You can update the Monitor manually by clicking the "Cook" button (the eye icon) on the Monitor Toolbar each time you make a change.
- You can set the Monitor to update automatically whenever you make a change by right-clicking the Cook button (the eye icon displays a green gear around it when Auto Update is enabled).

Auto Update Off: Auto Update On:

Note:

You can change the default Monitor Auto Update state in the Chalice Monitor Preferences panel.

You can also choose to update only a specified region of an image. This is a much faster way to evaluate an effect when working with large images. See "Using ROI to Update the Monitor" for instructions.

Frame Navigation Shortcut

In addition to using the navigation control buttons in the Frame Controller, you can also use the comma (,) and period (.) keys on your keyboard to go back or forward a frame at a time in any monitor.

Setting Monitor Display Options

Depending on the data format of your imagery, you may need to adjust the **Low, High,** and **Gamma** parameters (or use the **Presets/LUTs** menu to select predetermined values) in order to see the image clearly in the Monitor. For more information, refer to "Using Monitor Presets and Display LUTs."

Other display options include **A/B wipes** (see "Performing A/B Wipes"), individual **channel display**, **histogram overlays** (in Cursor Mode), and zooming. These options are accessed from the Monitor Toolbar.

The Monitor Toolbar also gives you access to four **utility monitors** for the current image: Magnify, Waveform, Scope, and Histogram.

Another useful Monitor option is the Inspect Panel, which provides **pixel-level statistics.** The Inspect Panel is accessed from the Monitor Popup Menu.

Monitor Modes

You can work with Chalice node monitors in two different modes. The default is Multiple Monitor mode, but you can switch to Single Monitor mode by turning off Multiple Monitors in the Monitor Preferences panel. The choice is a matter of personal preference, and you should use the monitor interface option that best suits your work habits.

Multiple Monitors

In Multiple Monitor mode, you can open a separate Monitor window for each node in your network. This enables you to display your imagery at different stages simultaneously, for side-

by-side evaluation and comparison. You can change parameter settings in a node and update the display for that node monitor without affecting the display in any other node monitors that you have open.

Multiple Monitor mode does allow you access multiple node images from within a single Monitor to perform A/B wipes by using the A and B Image Buffer menus on the Monitor Toolbar.

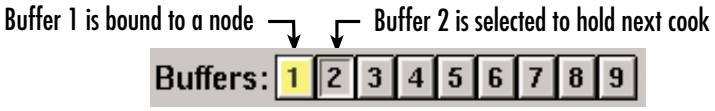
Single Monitor

Alternatively, you can work in Single Monitor mode to view all your imagery in the same Monitor window. In Single Monitor mode, you select the image you want to display in the Monitor from the A Image Buffer menu on the Monitor Toolbar.

You can store up to nine different node images in the Chalice display buffers, as long as Buffer Persistence is turned on (checked) in the Chalice Preferences menu. The next section explains how to use the display buffers.

Using the Display Buffers

When you open a node monitor, you process, or "cook," the current image at that stage of modification in the network. When you cook an image it is stored in an image display buffer. Chalice provides multiple display buffers for flexibility in viewing imagery at different nodes and at different stages of modification.



Right-click and hold any Display Buffer button to access a popup menu containing information and options:

- If the buffer is empty, the popup will read "Unattached."
- If an image has been cooked into the buffer, the popup will display the name of the node and information about the image: size; number of channels; bit-depth; and frame number.
- If the buffer is bound to a node (the buffer button will be yellow), a third option, "Unbind" will appear in the menu. Select "Unbind" to unbind the buffer.

Tip:

You can select the node name item in the Display Buffer popup menu and Chalice will highlight that node in the workspace. If the node is offscreen, Chalice will bring it into view.

Displaying Buffered Images

You can display the contents of any buffer in the currently active Monitor. Click and hold the A Image Buffer button on the Monitor Toolbar to access the Image Buffer menu.

The Image Buffer menu lists all of the Chalice display buffers by number ("buffer1," for example), along with their contents (the node name and frame number cooked into the buffer; "disk1 (frame4)," e.g.). If a buffer does not currently contain any image data, it is listed as "empty."

You can select any buffer that is not empty and the corresponding image will appear in the Monitor, replacing the previous image. The title bar of the Monitor will also update to reflect the change.

Tip:

You can add an image to a buffer without opening a Monitor.

Hold down the Alt key while selecting a Monitor option from the Local Node Menu. Chalice will cook the image without opening a Monitor for that node and store it in the selected buffer.

You can specify what images are stored in these buffers to control how your imagery is displayed and you can specify certain aspects of buffer behavior by using the options described in the next section.

Assigning Images to Buffers

By default, Chalice assigns the next available image buffer each time you open a Monitor. To assign an image to a specific buffer, click the corresponding buffer button (located directly below the Frame Controller) to select it. Then open a Monitor for the node image you wish to display.

The behavior of image buffers will vary, depending on how you have set the Buffer Auto Increment and Buffer Persistence options in the Chalice Preferences menu:

Buffer Auto Increment

The Buffer Auto Increment option specifies that the image for each node monitor that is accessed be stored in the next available buffer in line, rather than overwriting the contents of the last buffer used. (You can still overwrite a buffer manually by clicking on the buffer button to select it and then cooking a new node.)

Buffer Persistence

The Buffer Persistence option specifies that an image remain stored in a buffer, even when it is not currently being displayed in a Monitor. This enables you to access the contents of a buffer again without re-cooking the image.

The default state for this item in the Chalice Layout menu is off, because Buffer Persistence uses more memory. However, you may wish to turn persistence on, especially in Single Monitor mode, to use the A and B Image Buffer menus in the Toolbar to best advantage.

Bind Buffer

The Bind Buffer option, which is accessed from the Local Node Menu, allows you to associate a particular buffer with a node. When a buffer is bound to a node, only the image for that node can be cooked into the buffer. The buffer button turns yellow when the buffer is bound, and it cannot be selected.

To unbind a buffer, right-click and hold the buffer button to access the Display Buffer popup menu; select “Unbind” from the list. (You also unbind a buffer when you delete the node to which it was bound.)

Performing A/B Wipes

You can perform A/B wipes within any Monitor window, in Multiple or Single Monitor mode. You can wipe between any two node images as long as they are the same resolution and bit-depth. (Refer to “Wipes Using History,” below, to wipe between cooks of the same node image.) Follow these steps:

1. **Depress the A/B Wipe button on the Monitor Toolbar to activate wipe mode.**
   The image being displayed when the Wipe button is depressed becomes the “A” image for the wipe.

The Monitor will remain in wipe mode until you click the A/B Wipe button again to deactivate it.

2. **Click and hold the B Image Buffer button to display a menu of available images and select the one you want to use for the "B" image of the wipe.**
   Pointers appear at the top and bottom of the display to indicate the wipe border. The Monitor title bar now displays the names of both image buffers.

   Note:

   If the image you wish use for the B image of the wipe is not stored in a buffer, hold down the Alt key while selecting a Monitor from the local node menu of the node containing the image you wish to view.

   This will cook the image into a buffer without opening another monitor. You can then select it from the B image buffer menu.

3. **Drag a pointer left or right to wipe.**
   Click directly on either pointer and hold while dragging. The A image will be revealed from the left side of the monitor image area; the B image, from the right.

4. **If desired, select another image from the A and/or B image buffer menus to replace one or both of the images currently being used in the wipe.**

Wipes Using History

You can select the History item in the B Image Buffer menu to wipe between two cooks of the same node image. This allows you to gauge the effect of a change to a parameter setting or to compare two different frames in a sequence. Each node has its own History buffer.

To wipe between cooks, follow the steps for performing an A/B wipe above. However, for step 2, select "History" as the B image. Then either change a parameter setting for the node selected as the A image or go to a new frame, and press the Cook button before continuing to step 3.

Using ROI to Update the Monitor

"ROI" stands for "region of interest." The ROI option allows you to define a region of an image in the Monitor and cook only that region. This can save time when tweaking parameter values because Chalice does not have to recompute the entire image each time.

Tip:

Hold down the Control key while selecting a Monitor from the Local Node Menu to open a Monitor without processing the image (it will appear black), and then use ROI to cook a selected region only.

ROI is *not* applicable to such operations as scaling, which by their nature affect the entire image. ROI affects the current Monitor display only—not final processing of the imagery.

Follow these steps to use ROI in the currently active monitor:

1. **Press the ROI button on the Monitor Toolbar to select it.**
   The ROI icon will change to reflect its active state.

ROI Off: ROI On:

(Alternatively, you can select the "Enable ROI" option in the Monitor Popup menu. The Monitor Popup Menu is accessed by clicking and holding the right mouse button anywhere in the image.)

2. **Hold down the Alt key and simultaneously drag the cursor across a portion of the image using the left mouse button.**
   Chalice will draw a rectangle bounding the area you define by dragging. When you release the mouse button, this area becomes the region of interest.

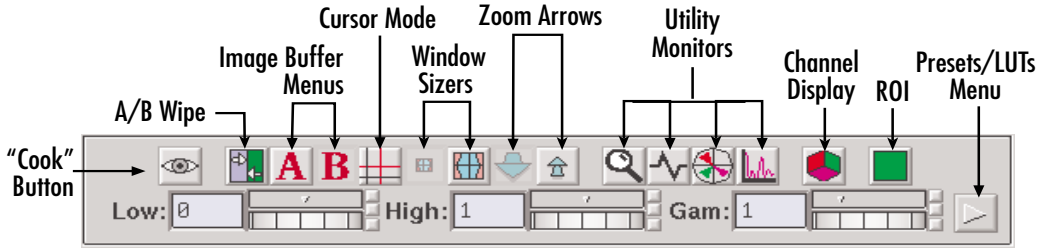
   You can redefine the region at any time by repeating this step on a different image area.

3. **Modify a parameter setting for the node and press the cook button.**
   (If Monitor Auto Update is on, you will not need to press the cook button.) Only the area of the monitor display that you have defined as the region of interest will update to reflect the change.

As long as the current Monitor remains open, all subsequent changes to parameter settings will be reflected only in the region of interest. You can change frames in the current Monitor display without disabling ROI. Only the portion of the new frame that falls within the previously defined area will update.

Using the Monitor Toolbar

The Monitor Toolbar offers a number of options for previewing and updating images in the Monitor.



"Cook" Button

The eye icon is the Cook button, which you click to process, or "cook," the image to update the display after any change to parameter values or after changing frames. The eye icon turns red while the system is processing.

If the Monitor Auto Update option is turned on, the image re-cooks automatically. You can toggle Auto Update on and off in any monitor by right-clicking on the Cook button.

A/B Wipe Button

When depressed, the A/B Wipe button enables wipe mode, in which you can compare two separate images or compare an image before and after modification. See "Performing A/B Wipes" for a complete description of the process.

A Image Buffer Menu Button

Click and hold this button to access a popup menu listing the contents of the Chalice display buffers. Select a buffer from the list to display the image it contains in the Monitor window.

B Image Buffer Menu Button

This button becomes active when A/B Wipe mode is enabled. Click and hold to access a popup menu listing the contents of the Chalice display buffers and the "History" buffer for the node. (The History buffer is used to wipe between cooks of the same node image.) Select a buffer from the list to use as the B image in the wipe.

Cursor Mode Button

Click this button to enable Cursor Mode. In cursor mode, a histogram display appears over the image in the Monitor, along with a movable crosshairs device. Two graphs display the color values of all pixels along the X and Y coordinates marked by the crosshairs. You can click anywhere in the image to reset the crosshairs position and update the graphs.

(To view a histogram of the color values for the entire image, click the Histogram button on the Monitor Toolbar to display the Histogram window.)

Window Size Buttons

Click the Window Size buttons to reduce or enlarge the size of the Monitor window.

Zoom Buttons

Click the Zoom buttons to zoom in or zoom out.

Tip:

You can click and drag the image, using the middle mouse button, to move it around in the Monitor window. This is particularly useful when you have zoomed in on an image.

Perhaps even more useful is the F9 hot key, which re-centers the image in the monitor frame if it has been dragged out of place and returns the image to the original scale.

Utility Monitor Buttons

Click any of these four buttons to bring up the associated utility monitor—Magnify, Waveform, Scope or Histogram—for the current image. Refer to "Using the Utility Monitors."

Channel Display Menu Button

Click and hold the Channel Display button to access a menu listing all channels in the current image. You can select any individual channel for display, or you can choose RGB display to view the full-color image.

ROI Button

Click the ROI button to toggle the region-of-interest option on and off. Refer to "Using ROI to Update the Monitor" for instructions.

Low, High, Gamma Parameters

The Low, High, and Gamma parameters, located on the bottom row of the Toolbar, enable you to adjust the Low, High, and Gamma color frequency distribution values for the current display.

In most cases you will only need to reset the High value, if any. The optimal value for any specific data type is probably already available as a preset in the Presets/LUTs menu (see below), which updates these parameters automatically when you select a preset, or overrides them altogether when a display LUT is selected.

Note:

These parameters adjust only the Monitor display; the actual image data is not modified.

Using Monitor Presets and Display LUTs

Click and hold the Presets/LUTs button to see a menu of presets and LUTs to choose from for the current monitor display:

- **Monitor presets** are predefined values for the Low, High, and Gamma parameters that are optimal for the type of image you are currently viewing. (When a LUT is selected, the Low, High, and Gamma parameters are deactivated.)
- **Display LUTs** are predefined lookup tables (LUTs) optimized for the display of certain types of imagery, including Cineon 10-bit log, Cineon that has been inserted into 16-bit space (so-called "10-bit linear"), as well as 16-bit and floating point CG image data.

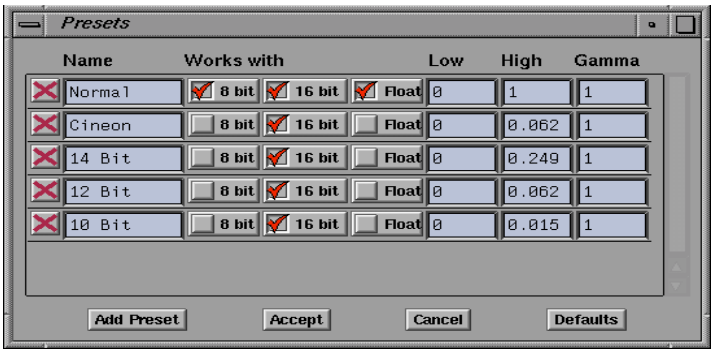
The Presets/LUTs menu is user-expandable. You can edit the preset values or add your own monitor presets to the menu. And you can add your own lookup tables for 8-bit, 16-bit, and floating point data to the menu.

Adding LUTs

To add a LUT to the menu, place it in the chalice2.0/support/luts directory. Example C source code for creating display LUTs is located in chalice2.0/support/src/luts.

Adding Presets

To add a preset to the menu, or to reset the presets to their default values, select the Edit Presets option from the Presets/LUTs menu. Chalice will display the Presets panel, which lists all of the available monitor presets:



Follow these guidelines to use the Presets panel:

- To change the default values of a monitor preset, key a new value into the appropriate data entry field. You can also redefine the image types to which the preset can be applied—8-bit, 16-bit, or floating point images—by clicking the appropriate checkbox. When you are satisfied, click the Accept button to actuate the changes and close the window.
- To add a preset, click the Add Preset button. Chalice will create a new entry in the list. Key a name into the Name field. Key in Low, High, and Gamma parameters values. Specify the types of files (8 bit, 16 bit, or Float) with which you wish the Preset to function by clicking the relevant checkboxes. Then click the Accept button.

- Delete a Preset by clicking the X icon at the left of the panel.
- Click the Defaults button to reset all Monitor Presets to their system default values.

Using the Inspect Panel

The Inspect panel is accessed by checking the Inspect box in the Monitor Popup Menu (right-click and hold anywhere in the monitor image to display this menu).

Tip:

You can use the Monitor Preferences panel to have the Inspect panel display by default when you open a monitor.

The Inspect panel provides information about the image size and about the pixel currently positioned under the cursor, including its location in XY coordinates and color values.

Tip:

You can use the Magnify monitor to easily identify specific pixels for examination by the Inspect panel.

By default, the Inspect panel displays the color values for the current pixel as RGBA. A menu on the Inspect panel enables you to switch the display from RGBA to HSL or HSV.

Using the Monitor Popup Menu

The Monitor displays a popup menu when you click the right mouse button anywhere in the Monitor image. This menu enables you to control Monitor interface options by using checkboxes to select or deselect menu items.

Full Interface

This option displays the Menus, Toolbar, and Inspect panel, which are described below.

Menus

This option adds a File-Display-Windows-Help menu bar at the top of the Monitor:

- The **File menu** contains two items: Image Save… and Quit. The Quit command closes the monitor. The Image Save… option brings up a "Save Image File" dialog in which you can save the current frame to disk as a TIFF image.
- The **Display menu** is a duplicate of the Monitor popup menu.
- The **Windows menu** is used to open the four monitor utility windows, Magnify, Waveform, Scope, and Histogram, which are also accessible from the Toolbar.
- The **Help menu** lets you access the Chalice Online Help system.

Enable ROI

This option enables region-of-interest updates. Refer to "Using ROI to Update the Monitor."

Toolbar

This option displays the Toolbar. Refer to "Using the Monitor Toolbar."

Inspect
This option enables you to display or hide the Monitor Inspect panel, which is described above.

Double Buffer (IRIX Only)
This option enables double buffering for smoother image display in the currently open Monitor. (Double buffering is not available on NT-based machines.)

Anamorphic
This option enables you to view anamorphic imagery "unsqueezed"; that is, as it would appear when projected. If you view non-anamorphic imagery in this mode, the image will appear stretched in the X dimension to achieve the 2.40:1 aspect ratio.

Quit (ALT x)
This command closes the Monitor window.

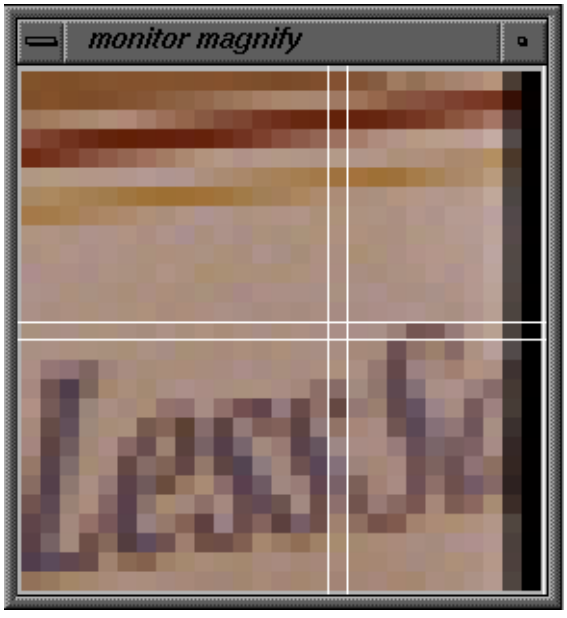
Using the Utility Monitors
Four utility monitors are accessible from within the Chalice Monitor window by clicking the appropriate button on the Monitor Toolbar:

Magnify: Waveform: Scope: Histogram:

Each utility monitor is described below.

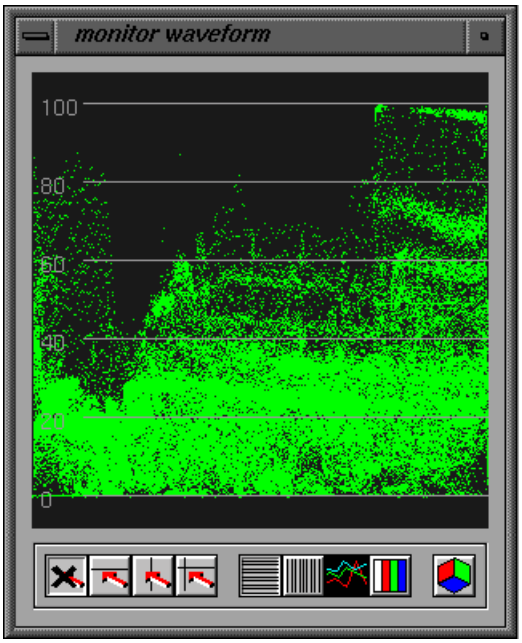
Magnify Monitor
The Magnify window displays a pixel-level close-up of a portion of the image. As you move the cursor over the Monitor, the display in the Magnify window also shifts to display the area surrounding the current location of the cursor.

The Magnify window features a double-crosshairs device that allows you to center on any pixel. You can use the Magnify window in conjunction with the Monitor Inspect panel to obtain detailed data about the position and color values of specific pixels. (See also "Using the Inspect Panel" earlier in this chapter.)



Waveform Monitor

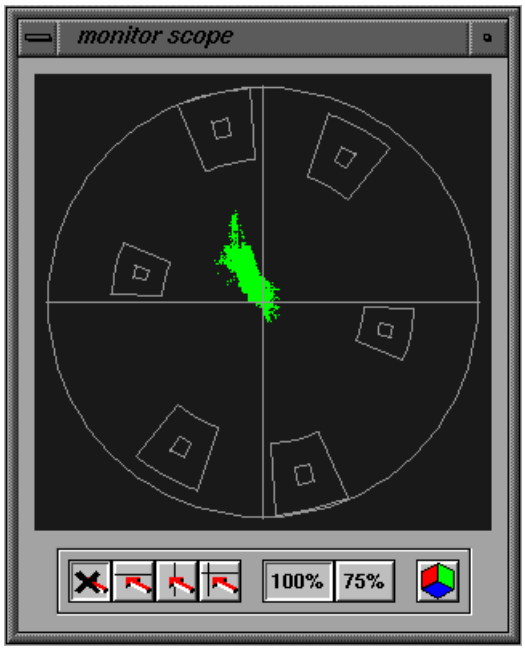
The Waveform window enables you to examine the image data in composite or parade (YRGB) waveform displays. (Chalice also provides a vectorscope display in the Scope Monitor.)



Scope Monitor

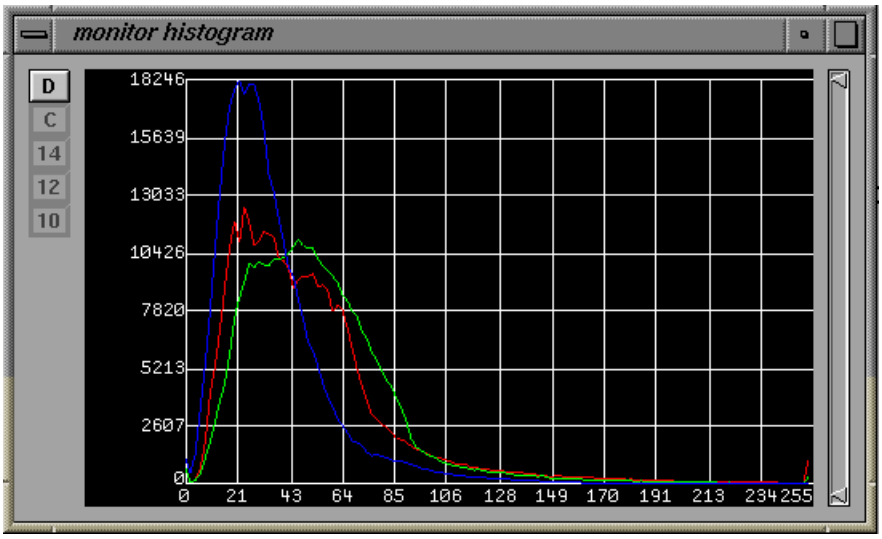
The Scope window provides a vectorscope display of the image data on a 360° circular scale. Hue and saturation parameters are best viewed in the Scope.

Hue data is represented in the Scope by the angle of the vector. The degree of saturation is represented by the amplitude of the vector, with vectors near the outer edge indicating high saturation and those near the center indicating low saturation.



Histogram Monitor

The Histogram window displays a frequency distribution graph of color values for the current image. You can adjust the perspective by using the vertical sliders on the right. For 16-bit images, you can use the buttons on the left to change the graph display value range from the default (D) to Cineon (C), 14-bit, 12-bit, or 10-bit.



USING FLIPBOOKS

A Flipbook enables you to create and view the animation of a sequence of images. You can access a Flipbook from any node in the network (except for Exec and Network nodes).

You can use the Flipbook to evaluate an animated effect by viewing the imagery at the current node. You can then adjust the various node parameters accordingly, modifying the effect, and creating new flipbooks if necessary, until you are satisfied.

You can also save a Flipbook to disk for later playback.

Creating a Flipbook

To create a Flipbook, follow these steps:

1. **Click and hold the Local Node Menu button for the node you wish to view.**
   Chalice displays the Local Node Menu.

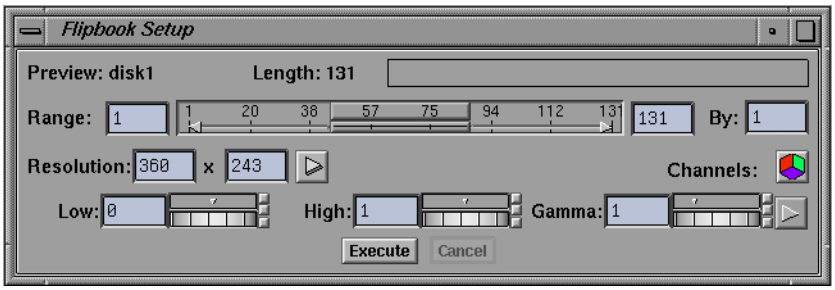
2. **Select Monitor Flipbook.**
   Chalice displays the Flipbook Setup window (see below).

3. **Press the Execute button in the Flipbook Setup window.**
   Chalice begins to process the sequence and opens the Flipbook window.

   Once processing is complete, the Flipbook Setup window disappears and the Flipbook becomes available for use.

Flipbook Setup Window

The Flipbook Setup window enables you to specify various display options before creating a flipbook.



Range

The Range parameter allows you to set the frame range and increment to be used for the flipbook by typing start frame, end frame, and increment values into the appropriate data entry boxes. By default, the Flipbook Setup window displays the total number of frames set in the Frame Controller and an increment of 1.

Presets/LUTs Menus

You can set the Low, High, and Gamma parameters directly by keying the values in the data entry boxes, or you can choose an appropriate setting or LUT from the popup menu.

Resolution

You can adjust the resolution of the flipbook by selecting a size from the Resolution menu, which includes these options: Icon, Low2 (half of low), Low, Medium, Full, 2X, 3X, and 4X. Alternatively, you can type specific pixel dimensions in the “Res:” data entry boxes.

Channel Display Button

You can choose to create a flipbook of the RGB image data, which is the default setting, or of any individual channel in the image by selecting an available option from the Channel Display popup menu.

Execute

Click the Execute button when you are ready to begin processing the frames. Chalice will open a Monitor Flipbook window and a progress indicator in the Flipbook Setup window will display the current frame being processed.

Once Chalice begins processing the flipbook, the Cancel button becomes active. Click the Cancel button at any time to stop the operation.

> **Note:**
>
> To create a flipbook of more than 90 to 100 frames on an IRIX system, you will probably need to increase the number of shared memory segments from the default setting as shipped by SGI. For more information, see "Enhancing Operation of Chalice on IRIX Systems" in chapter 3.

Once the flipbook is complete, the Flipbook Setup window will close. Use the controls in the Flipbook Monitor window to view the flipbook.

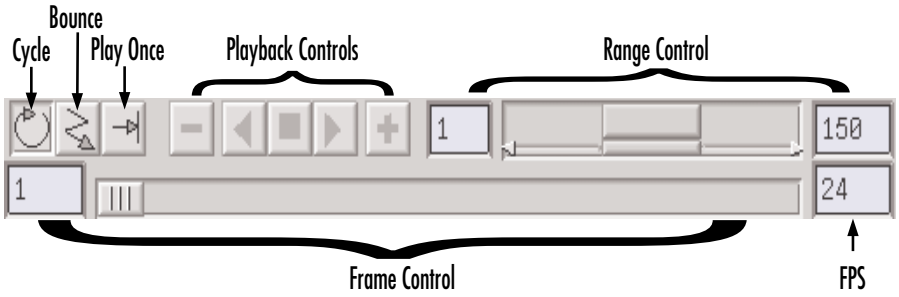
Using the Flipbook Controls

The Flipbook provides playback controls and the same Toolbar as the standard node Monitor. However, in the Flipbook only the Window Size, Zoom, and Cursor Mode buttons on the Toolbar are active. (If Cursor Mode is selected, the graphs animate as the Flipbook plays.)

For more information about these controls, see "Using the Monitor Toolbar."

You can right-click anywhere in the image area of the Flipbook to access the Flipbook popup menu, which enables you to save flipbooks to disk, load audio files, and change the flipbook interface configuration.

Beneath the Flipbook Toolbar is the Flipbook Frame Controller:



Looping Controls

Used in conjunction with the Play button, these three buttons control the way in which the Flipbook will play back a sequence of images:

- **Cycle** causes the Flipbook to repeatedly loop through a sequence. For example, a sequence of 1, 2, 3 will repeat as 1, 2, 3, 1, 2, 3, 1, 2, 3, etc.
- **Bounce** causes the Flipbook to repeatedly play forward and back through a sequence. For example, a sequence of 1, 2, 3 will repeat as 1, 2, 3, 2, 1, 2, 3, 2, 1, etc.
- **Play Once** causes the Flipbook to play the sequence once, then stop.

Playback Controls

- The Minus (-) button steps back through the sequence frame by frame.
- The Reverse arrow plays the sequence in reverse order, from last frame to first.
- The Stop (rectangle) button stops playback of the sequence.
- The Forward arrow button plays the sequence.
- The Plus (+) button steps forward through the sequence frame by frame.

Range Control

The Range Control slider enables you to specify the range of frames you wish to play back. Click and drag the arrows on the slider to specify the starting and ending frames of the desired range, or key the desired frame numbers into the data entry boxes.

Frame Control

The Frame Control slider enables you to move back and forth through a sequence by dragging the slider bar. The current frame number is indicated in the associated data field.

Frames-Per-Second

The fps indicator in the lower-right corner displays the playback speed in frames per second. You can key a new playback speed into the field for the current monitor, or change the default fps value in the Flipbook Preferences panel. (To learn how to access this panel, refer to the following section.)

Using the Flipbook Popup Menu

Right-click and hold anywhere in the image area to display the Flipbook popup menu, which provides the following checkbox and dialog-access controls:

Toolbar

This checkbox controls the display of the Flipbook Toolbar, which is described in the previous section, “Using the Flipbook Controls.”

Frame Control

This checkbox controls the display of the Frame Controller panel, which is described in the previous section, “Using the Flipbook Controls.”

Audio

This checkbox controls the display of the Audio Controller panel. This panel is used to play back an AIFF file that has been loaded into the flipbook using the Audio Load menu item (see below).

The Audio Controller panel includes a Speaker icon, which you click to enable the audio, which will then play when the flipbook plays. The panel also features a pair of data entry fields that you use to control audio sync by linking a frame number to an audio sequence mark, in seconds. By default, frame 1 is linked to 0 audio seconds.

Image Save...

Select this menu item to access the “Save Image File” dialog, in which you can save the currently displayed frame of the flipbook to disk.

Movie Save...

Select this menu item to access the “Save Movie File” dialog, in which you can save the flipbook to disk as an FRP file. You can play the file at any time by selecting “Play FRP Movie File…” from the Chalice File menu.

(For more information about the FRP file format, refer to the "Disk Output Node" description in chapter 12, "I/O Nodes.")

Audio Load...

Select this menu item to access the "Choose Audio File" dialog, in which you can select an AIFF format audio file for playback with the flipbook image sequence.

Audio On

This checkbox performs the same function as clicking the Speaker icon on the Audio Controller panel.

Double Buffer (IRIX Only)

This checkbox enables you to specify double buffering for smoother image display in the currently open flipbook.

Anamorphic

This checkbox enables you to play an anamorphic image sequence "unsqueezed"; that is, as it would appear when projected. If you check this option for non-anamorphic sequences, the frames will appear stretched in the X dimension to achieve the 2.40:1 aspect ratio.

Preferences...

Select this menu item to bring up the Flipbook Preferences panel to change flipbook default settings. This is the same panel that is accessed from the Chalice Preferences menu. The Flipbook Preferences panel is described in chapter 11, "Setting Preferences."

9 SPACE-TIME EDITOR (STED)

Chalice includes a channel and temporal data editing utility called the Space-Time Editor (STED) that enables you to perform complex editing of parameter values, individually or collectively. The STED is a powerful and versatile tool that can help you create precisely the image you desire.

The STED displays parameter data as a waveform; that is, it plots the values for a parameter across the range of frames in the current image sequence. You use the STED to control the interpolation of this parameter data.

The STED will display a waveform for each channel or other parameter type in a group. (A group is usually associated with a single node.) The waveforms appear on a scrollable and adjustable two-dimensional graph:

- The horizontal axis of the graph represents time, expressed in equal increments of frame numbers.
- The vertical axis represents image values, which will vary depending on the type of operation.

Depending on the node or parameter for which you opened the STED window, each waveform may represent any one of the following:

- the entire image
- a single image channel
- the X and Y coordinates of particular image pixels

Waveforms can be manipulated individually or in groups. A waveform can be split into segments by inserting key frames at each point you want a new segment to start.

You adjust the value and position of key frames and apply different functions to segments to control the interpolation of parameter data.

STED Buttons

You open the STED window by double-clicking on a STED button. These buttons appear on numerous parameter tabs throughout the Chalice interface.

The first time you click a STED button, Chalice makes the associated parameter data available to the Space-Time Editor, and changes the appearance of the STED button icon from a gray matrix to a waveform over a black background:

STED Button (inactive): STED Button (enabled):

The second time you click the STED button, Chalice opens a STED window, which enables you to animate values for a particular node or parameter.

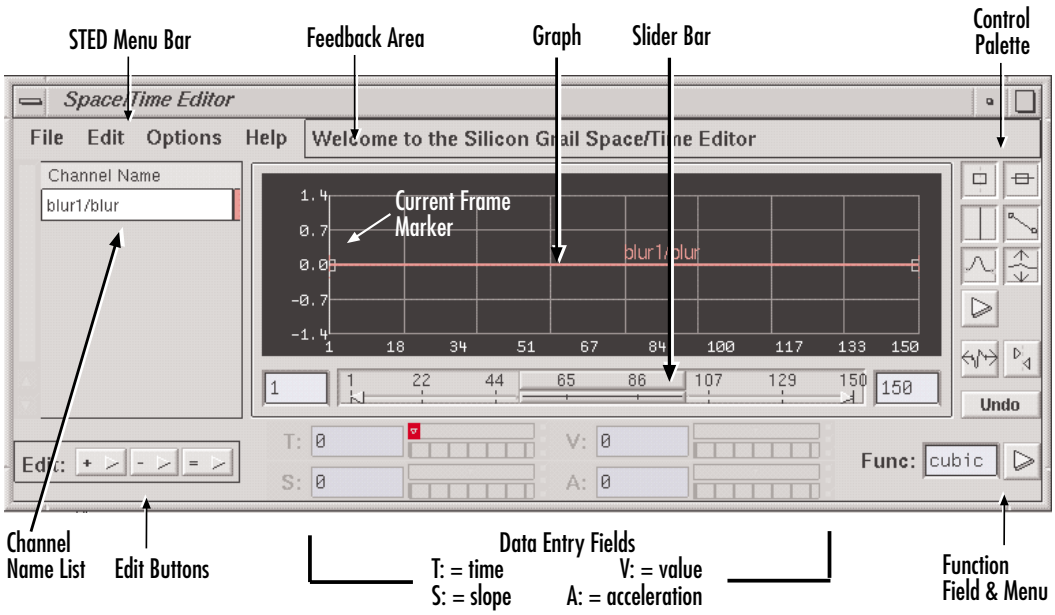
Note:

The STED button for a parameter must be in its active state, or that data will not be available for display and editing in the STED window.

You can Alt-click any active STED button to return it to its inactive state.

STED Window

The STED window displays a graph surrounded by a number of buttons, menus, and data entry fields that enable you to manipulate the waveforms in the graph in various ways.



Resizing the STED Window

You can enlarge or reduce the size of the STED window by clicking the left mouse button and dragging any corner or side handle on the STED window.

Tip:

The larger the STED window is, the easier it will be to examine, select, and adjust points and segments in the graph.

Scrolling in the Graph

To scroll vertically through the STED graph, click the middle mouse button anywhere in the graph and drag up or down.

To scroll horizontally, use the Slider Bar at the bottom of the graph. The bar on the slider is divided into two parts; click the bottom half of the bar and drag in the direction you want to scroll.

Adjusting the Display Range in the Graph

You can increase or decrease the range of values displayed in the vertical axis of the graph by holding down the middle mouse button and dragging left (to decrease) or right (to increase).

The Slider Bar at the bottom of the graph allows you to adjust the range of values displayed in the horizontal axis of the graph. Click and drag the arrows on the slider to adjust the range, or key new values into the start and end fields located on the left and right ends of the slider.

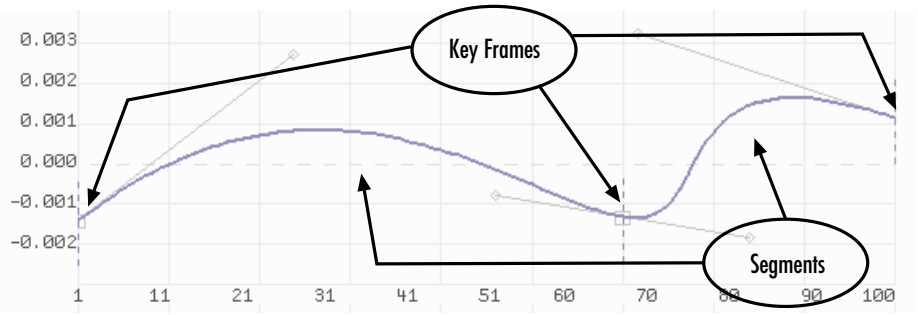
Tip:

Activate (depress) the Vertical Adapt and Horizontal Adapt buttons on the Control Palette, or choose the equivalent commands from the STED Options menu, and the graph display will adjust automatically to reflect the current range of values and frames:

- Vertical Adapt adjusts the display to encompass the current value range in the vertical axis of the STED graph.
- Horizontal Adapt adjusts the display to encompass the current frame range in the horizontal axis.

STED Graph

The STED graph displays a waveform to represent each active parameter in the Channel Name list. Each waveform can be divided into additional segments, and each segment can be edited individually.



Adding Segments

Every waveform is bounded by two key frames, the first frame and the last frame, which allow you to adjust the start and end values for the waveform. You can divide a waveform into additional segments by adding key frames.

To create a new segment or segments, hold down the Alt key and click at every point on the line where you want a new segment to start. Chalice will mark the beginning of each new segment (and the end of the previous segment) by creating a key frame at that point.

Each time you Alt-click on a waveform, Chalice will create a new key frame at the same position on all other waveforms in the graph. To isolate a single parameter for modification, turn off the display of any other waveforms by clicking on their names in the Channel Name list.

You can then adjust the horizontal position of the key frame, to change the length of the segment, and the vertical position of the key frame, to change the start or end value for the interpolation of the segment. (For more information, see “Editing Segments” below.)

Deleting Segments

To remove a segment, click on it to select it and press the Delete key, or select Delete from the STED Edit menu. (Alternatively, you can select the time mark to the left of the segment rather than the segment itself.)

Chalice will delete the segment along with the key frame on its left (this is the key frame that controlled the start value for the deleted segment). The segment before it then stretches to fill the gap.

If the segment you delete is the first or last segment in the waveform, however, the entire waveform will be shortened accordingly.

Editing Segments

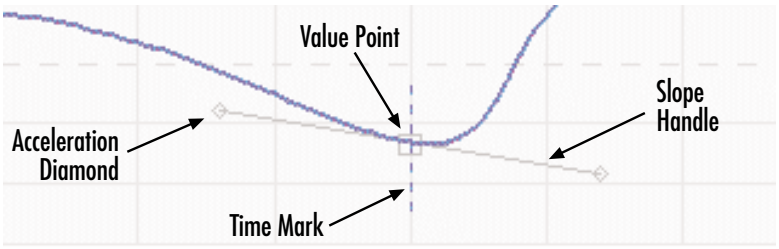
The following characteristics of segments can be modified:

- start values
- end values

- function
- slope
- acceleration

The start and end values of a segment are defined by the key frames on either side. Different functions can be applied to a segment to control the interpolation from the start value to the end value. For some functions you can control the slope of the curve and the rate of acceleration.

Each key frame has at least two control elements: the value point and the time mark. For segments with functions that apply slope and acceleration values, key frames also provide slope handles and acceleration points to control these characteristics of the curve.



Value Point

The Value Point is a square that controls the value represented in the vertical axis of the graph. Click and drag the value point up or down to adjust the value.

Alternatively, you can select the point to highlight the corresponding data entry field below the graph (it is labeled "V:") and key in a specific numeric value.

Time Mark

The Time Mark is a vertical dashed line intersecting the value point. Click and drag the time mark left or right to adjust the frame number.

Alternatively, you can select the line to highlight the corresponding data entry field below the graph (it is labeled "T:") and key in a specific frame number.

Slope Handle

A Slope Handle is a white bar intersecting the key frames of function segments that use slopes. You can click and drag a slope handle radially around the key frame to adjust the angle of the slope.

Alternatively, you can select the slope handle to highlight the corresponding data entry field below the graph (it is labeled "S:") and key in a specific value.

Acceleration Diamond

An Acceleration Diamond is a diamond-shaped point attached to each end of a slope handle for functions that use acceleration. You can click the diamond at either end of the handle and drag it in or out (the handle will shrink or stretch) to change the rate of acceleration.

Alternatively, you can select the acceleration diamond to highlight the corresponding data entry field below the graph (it is labeled "A:") and key in a specific value.

Tip:

You can select more than one item in the graph at a time and apply the same modification to all selected items in one step. To select multiple items, hold down the Shift key while clicking on each item you want to select.

For example, you could select several value points and drag them all vertically in the graph to apply the same rate of change to all of them. Or, you could give the selected points identical values by keying a specific number into the "V" data entry field below the graph.

Untying Segments

By default, the key frame separating two segments is tied; that is, its value point defines the end value of the segment on the left and the start value of the segment on the right. Consequently, any changes you make to the value apply to both segments.

However, you can untie the segments so that each can be edited separately. To untie two segments, select the value point between them and click the Tie/Untie button on the Control Palette (or select the Tie/Untie command from the STED Edit menu).

You can also untie the slope and acceleration values of two segments. To untie the slope value, select the slope handle that intersects the key frame between the segments. Then click the Tie/Untie button.

To untie the acceleration value, select the diamond on either end of the slope handle and click the Tie/Untie button.

- When tied, the value point of the key frame is displayed as a square; when untied, the square changes to two triangles, each of which can be selected and modified separately.
- When a slope handle is tied, it is a solid line intersecting the key frame point; when untied, it changes to a pair of dashed lines radiating from the key frame point, each of which can be selected and modified separately.
- When an acceleration point is tied, it appears as a diamond shape; when untied, it changes to an "x" shape. Each "x" can be selected and modified separately.

Channel Name List

The Channel Name list on the left side of the STED window displays the current list of parameters, each of which corresponds to a different waveform in the graph. Each item has a unique name, which always includes the node name and parameter name. For example, "blur1/blur" signifies the blur parameter of a node named "blur1."

Any item in the Channel Name list may be deactivated. If you click an active item in the list, the name is grayed out and the corresponding waveform disappears from the graph. If you click a grayed-out name in the list, the item becomes active again and its waveform reappears in the graph.

Edit Buttons

The Edit buttons enable you to add or remove parameters from the Channel Name List. (Remember that the STED window is capable of displaying parameter data for any node, not just the currently active node, as long as the STED button associated with the parameter is in its active state.)

The contents of the Edit menus will vary, depending on the current state of your grail file. Click any of the Edit buttons to display a popup menu of available parameters, grouped by node. For example, "chalice1/crop1/*" would represent all active parameters in a node named "crop1."

"Plus" Menu

The (+) button displays a popup menu of all active parameter data. Select any item in the menu to add it to the current list of items in the Channel Name List. Select the "*/*/*" option to add all items in the menu to the list.

"Minus" Menu

The (-) button displays a popup menu of all items currently in the Channel Name List. Select any item to remove it from the list. Select the "*/*/*" option to remove all items from the list.

"Equals" Menu

The (=) button displays a popup menu of all active parameter data in any node. Select an item from this menu to replace all items currently in the Channel Name list with the selected parameter data. Select the "*/*/*" option to replace the current Channel Name List with all items listed in the popup menu.

Data Entry Fields

The data entry fields at the bottom of the STED window display current values for selected segments and key frames in the graph. You can adjust these values by keying new values into any active field, or by using the associated tuner, to make precise numerical adjustments.

"T:" Field

When active, this field displays the currently selected value for the horizontal axis of the graph (the time mark), which usually represents a frame number.

"V:" Field

When active, this field displays the currently selected value for the vertical axis of the graph (the value point).

"S:" Field

When active, this field displays the slope value of the currently selected function curve. Slope applies only to the cubic, Bezier, and quintic functions.

"A:" Field

When active, this field displays the acceleration value for the currently selected function curve. Acceleration applies only to the Bezier and quintic functions.

Function (Func) Menu

The Function menu in the lower-right corner of the STED window allows you to select an interpolation function to apply to the currently selected segment. The menu includes the following commonly used functions:

- constant(): Constant value of the in point.
- linear(): Straight line interpolation from 0 to 1.
- ease(): Values ease in smoothly starting at 1 and ease out smoothly at 0.

- easein(): Ease in smoothly at the start.
- easeout(): Ease out smoothly at the end.
- cubic(): Interpolates the in and out points using the slopes. Results in a cubic polynomial.
- bezier(): Interpolates the in and out points by fitting a Bezier curve using the slopes and accelerations.
- quintic(): Interpolates the in and out points by using slopes and accelerations. Results in a quintic (degree 5) polynomial.
- gammaf(): Interpolates the in and out points by fitting a gamma curve.

Function (Func) Field

The Function Field in the lower-right corner of the STED window displays the interpolation function currently applied to the selected segment.

The Function field enables you to modify a function, or apply a function not included in the Function menu, by keying an appropriate expression into the "Func" field and pressing the Enter key.

For example, you can reverse any function by multiplying by -1. To reverse a linear function, modify it as follows:

```
linear() * -1
```

To create a cosine curve, try using this expression:

```
cos($F * 8)
```

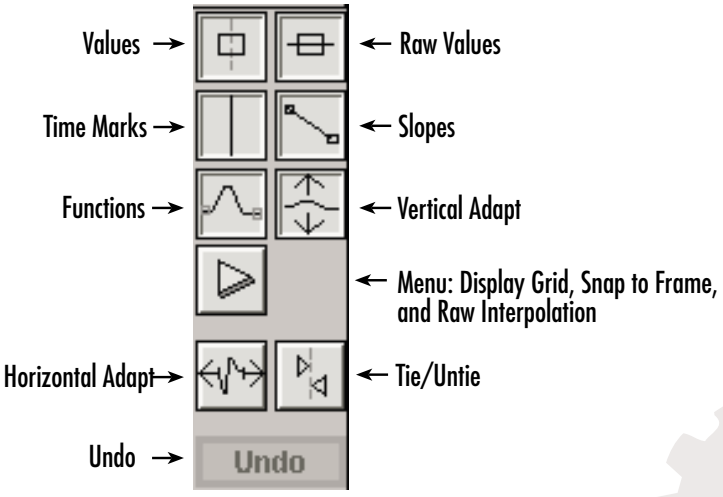
To view any segment as raw data, use this expression:

```
raw()
```

For more information about using expressions, see appendix B, "Chalice Expression Language."

Control Palette Buttons

The right side of the STED window features a palette of buttons that allow you to control display options for the STED graph. (These options are also available in the STED Options and STED Edit menus.)



All options are on by default, with the exception of Horizontal Adapt, Tie/Untie and Undo, which by the nature of their operation do not have a default state.

In instances where the graph contains numerous, closely spaced waveforms, or waveforms with many key frames, it may be helpful to turn off the display of one or more options to facilitate the selection of other options.

Values

This button controls the display of value points for key frames in the graph. When the Values button is active, all value points in the graph are visible. Any point can be selected and modified.

If you turn Values off, the value points disappear from the graph and cannot be selected or modified.

Raw Values

This button controls the display of raw value data points in the graph. This option is only applicable to segments in the graph that are displaying raw (uninterpolated) data.

When Raw Values is on, a square representing the value in the vertical axis is displayed across the range of the horizontal axis. Each raw value point may be adjusted individually.

If you turn Raw Values off, the value points disappear from the graph and the data cannot be modified.

Time Marks

Like the Values button, this button controls the display of time marks for key frames in the graph. When the Time Marks button is active, all time marks in the graph are visible. Any time mark line can be selected and modified.

If you turn Time Marks off, the time marks disappear from the graph and cannot be selected or modified.

Slopes

This button controls the display of slope handles for function segments that use slopes. When the Slopes button is active, all slope handles in the graph are visible. Any handle can be selected and adjusted.

If you turn Slopes off, the handles disappear and slopes cannot be selected or modified.

Functions

This button turns functions on or off. When the Functions button is on, all segments can be selected and new functions can be applied to them.

If you turn Functions off, segments in the graph are still visible, but they cannot be selected and you cannot apply a new function to any segment.

Vertical Adapt

This command adjusts the vertical scale factor to display the entire range of current parameter data values for the vertical axis.

When the Vertical Adapt button is on, the vertical scale of the graph will adapt to the new value range automatically after every modification.

You can click the middle mouse button and drag back and forth in the graph to override the current Vertical Adapt setting and adjust the vertical scale manually.

Horizontal Adapt

This command adjusts the horizontal scale factor to display the entire range of current parameter data values for the horizontal axis.

You click the Horizontal Adapt button after a modification to adapt the horizontal scale of the graph to the new value range.

Alternatively, you can use the Slider Bar at the bottom of the graph to adjust the horizontal scale manually.

Tie/Untie (ALT t)

This button enables you to tie or untie two segments at a specific key frame. When the Tie/Untie button is clicked, it always toggles the selected item to a new state—if it is tied, Chalice will untie it, and vice versa.

Menu

Press the Menu button on the Control Palette to display a popup menu with three options. A checkmark next to an option indicates that it is on (all three are on by default).

Display Grid toggles the background grid display in the graph on and off.

Snap to Frame ensures that any key frame you reposition by dragging its time mark snaps to the nearest exact frame number when you release the mouse button.

Raw Interpolation applies only to raw data segments. This option adds linear interpolation between samples at each frame.

Undo (ALT u)

You can click the Undo button whenever it is available to undo the last change you made in the STED window.

STED Menu Bar

The STED features a menu bar with four options: File, Edit, Options, and Help.

File Menu

The STED File menu enables you to export and import parameter data and to quit the STED window.

Export (ALT e)

The Export command opens a standard Chalice "Save To" dialog box that allows you to save current data for all parameters in the Channel Name list.

By default, the Save To dialog displays the name "untitled.ch" in the File field. You can remove the ".ch" extension when you rename the file and Chalice will still be able to load the data, but it is recommended that you use this extension to easily identify the file type. You can save this data in any directory. You can use the Export command to save changes to parameters for which a file has already been created. Each time you perform an export operation, the information is displayed in the feedback area at the top of the STED window.

Export Active Channel...

Export Active Channel performs the same function as Export, but only those parameters in the Channel Name list that have not been grayed out will be exported. This option allows you to quickly deselect parameters that you do not want to export, or do not want to export to the same file.

Export Raw (ALT w)

This option is the same as Export, except that instead of exporting the functions which make up the curve for each channel, the actual values at each frame are written out.

These values are written out as ASCII text, which can be read by any standard text editor or passed to other programs. The file contains a line for each frame that is in the selected range, and on each line is a tab-separated value for each channel that was exported.

Export Active Raw (ALT W)

This option is the same as Export Raw, except that only those channels which are active in the Channel Name list are exported.

Import (ALT i)

Use the Import command to apply data (function or raw) from an external file to the parameters in the Channel Name list. When you execute the Import command, Chalice displays a standard "Load from" panel that enables you to select a file. If the data is raw, then the active channels (that is, the channels the data is being imported into) are converted to raw(), and values are inserted for each frame that is active.

The number of channels in the file does not have to match the number of active channels, and the number of frames in the file does not have to match the active frame range. In such a case, Chalice will act as follows, notifying the user in each case:

- If the input file has more frames than the current range, it only uses what it needs and ignores the rest.
- If the input file has fewer frames than the current range, it uses what it has.
- If the input file has more channels than are active, it only updates the active ones, in order, and discards the rest.
- If the input file has fewer channels than are active, it updates what it can.

Quit STED (ALT q)

The Quit STED command closes the STED window and quits the Space-Time Editor.

Edit Menu

The STED Edit menu includes the Undo and Tie/Untie commands, which are described in the section on the Control Palette, as well as commands for copying channel values and pasting them to another channel and for deleting selected channel segments:

Copy Channel (ALT c)

The Copy Channel command copies the selected channel.

The Copy Channel command is only available when a single channel in the STED is selected. If the STED contains other active channels, deselect them by clicking on the associated channel name in the list. Deselected channels will be grayed out.

Paste Channel (ALT v)

The Paste Channel command replaces the values of the selected channel with the channel values that you last copied using the Copy Channel command (see above). Only one channel in the STED can be available to use the Paste Channel command (any other channels in the list must be deactivated; that is, grayed out).

Delete Selected (DEL)

The Delete Selected command deletes selected segments in the graph. Alternatively, you can press the Delete key on your keyboard.

Options Menu

The STED Options menu allows you to control various display options in the STED window. This menu is an alternate means of accessing the same options available using the Control Palette. For more information, see the description of the Control Palette earlier in this section, where these options are listed and defined.

The one exception is Long Label Names:

Long Label Names

This option, which is active by default, displays both the channel name and the node name, in the form "node/channel" (e.g., "blur1/blur"). Turn Long Label Names off to display only the channel names (e.g., "blur").

Help Menu

The STED Help menu contains two items:

General (ALT h)

The General command opens the Chalice Online Help System to this chapter.

Version (ALT V)

The Version command displays the current version number of the Space-Time Editor in the feedback area at the top of the STED window.

10 MAIN MENU BAR

The Chalice Main Menu Bar provides access to commands and options for working with grail files and the nodes that compose them.

The Chalice Main Menu bar includes:

File Menu ..... p. 10.2
Edit Menu ..... p. 10.4
Group Menu ..... p. 10.6
Layout Menu ..... p. 10.7
Preferences Menu ..... p. 10.9
Admin Menu ..... p. 10.10
Help Menu ..... p. 10.12

FILE MENU

The File menu commands allow you to load and save grail files or movie files or to quit Chalice. (See also "Working with Grail Files" in chapter 5, "Building Chalice Networks.")

New (ALT n)

This command creates a new untitled grail file. A notifier panel will give you the option to save or discard changes to the currently open file.

Open... (ALT o)

This command opens a "Load from:" dialog that enables you to open an existing grail file.

Merge... (ALT a)

This command opens a "Load from:" dialog box that enables you to select a grail file to merge with the currently open file. The merge operation copies the contents of the selected file and places them in the current file, with all connections intact.

> **Tip:**
>
> Chalice will place the nodes to be merged in the center of the workspace. To avoid placing these nodes atop the existing nodes in the current grail file, move to a clear area of the workspace before selecting Merge from the menu.

Save (ALT s)

This command opens a "Save to:" dialog box that enables you to save a new, untitled grail file to disk. For previously saved grail files, this command saves the current version of the open grail file.

Save As... (ALT S)

This command opens a "Save to:" dialog box that enables you to key in a new name for the current grail file and save it to disk.

Save Selected As... (ALT E)

This command opens an "Export to:" dialog box that enables you to save all currently selected nodes to a new grail file.

Revert... (ALT R)

This command opens a "Load from:" dialog box that enables you to revert to a previously autosaved version of the currently open file.

The names of autosaved files are based on the name of the current file, and include the date, time, and count of the autosave to enable you to select the autosave version you want to use. The date format is mmddyyyy (note that it is year-2000 compliant), followed by the time of the autosave (hhmmss).

Refer to the "Autosave Preferences" section of chapter 11 for a description of the Autosave panel, which enables you to change various default settings.

Play FRP Movie File...

This command opens a dialog box that enables you to select a Chalice FRP file for playback in a flipbook. FRP files are created when you save a flipbook to disk, or when you output images in FRP format.

The Play Movie Files dialog box works like any other Chalice file chooser dialog. You navigate to the directory containing the .FRP file you want to play and double-click on the filename. Chalice will bring up a monitor flipbook with the file loaded into it.

MRU File List

The File menu contains a list of up to four of the most recently used grail files. As a shortcut, you can select a file from this list to open, rather than using the Open command and going through the "Load from:" dialog box to locate the file.

Quit Chalice (ALT q)

This command quits Chalice. A notifier panel enables you to save or discard the current version of the open file before quitting or to cancel the Quit command.

EDIT MENU

The Edit menu enables you to cut, copy, paste, clone, and delete the selected node or nodes in the workspace. To select more than one node at a time:

- hold down the Shift key while clicking on the title bars of the nodes you wish to select.
- hold down the Control (or Alt) key and drag a bounding box around the nodes you wish to select.
- choose the Select All Nodes command from this menu.
- choose "Pick All Ancestors" or "Pick All Descendents" from the Local Node Menu.

The Edit menu also enables you to access dialogs to edit global variables, change the contents of the Custom node menu, access the STED channel editor window, or to search for nodes by name.

Cut (ALT X)

This command removes all selected nodes from the workspace and places them in the Chalice clipboard, a special buffer used for the Cut, Copy, and Paste operations. Connections among the selected nodes are retained when they are cut; however, all external connections—connections to non-selected nodes—are lost.

Copy (ALT C)

This command creates a copy of all selected nodes and places them in the Chalice clipboard, a special buffer used for the Cut, Copy, and Paste operations. The copied nodes retain any connections they have to each other; however, all external connections are lost.

Paste (ALT V)

This command places the contents of the Chalice clipboard in the workspace. The clipboard is a buffer containing all nodes selected during the last (most recent) Cut or Copy operation.

All nodes in the workspace must have unique names; therefore, Chalice will append a digit to the name of a pasted node, if necessary. For example, if a node named "color" is copied and pasted, Chalice will name the pasted node "color1" (unless, of course, a node named "color1" already exists; in that case Chalice will increment the number by the first available digit that creates a unique name, such as "color2").

Clone (ALT C)

This command copies all selected nodes and places them in the workspace. Unlike nodes created by a Copy operation, cloned nodes retain all the external inputs of the nodes from which they were created.

> **Note:**
>
> The Clone operation uses a separate buffer from Cut, Copy, and Paste. You can clone a node or group of nodes without replacing any nodes currently in the clipboard as a result of the most recent Cut or Copy operation.

Select All Nodes (ALT A)

This command selects all nodes in the workspace.

Delete Node(s) (DEL)

This command deletes the currently selected node or nodes. You can undo this command by using the "Undo Delete" command.

Undo Delete (ALT u)

This command restores the most-recently deleted node (or nodes, if you deleted multiple nodes simultaneously) to the workspace.

Delete All... (ALT D)

This command deletes all nodes from the workspace. A notifier panel will ask you to confirm or cancel the operation. You can use "Undo Delete" to restore all the nodes, or use the "Revert" command in the File menu to restore an earlier version of the file, if available.

Delete All removes all nodes from the current level of the workspace only. However, this means that at the top level (/chalice/chalice1), the entire contents of the grail file is deleted. On the other hand, if this operation is performed at a lower level (/chalice/chalice1/group1, e.g.), that is, if you have pushed into a network node, all the nodes at that level (inside that network node) are deleted.

STED (Channel Editor) (ALT e)

This command opens a Space-Time Editor (STED) window that enables you to edit channel and temporal data. Only previously edited channels are available for loading and editing.

You can also open STED windows using STED buttons in specific node parameter panels. For more information, refer to chapter 9, "Space-Time Editor (STED)."

Edit Global Variables...

This command opens a window that contains a list of global variables, each of which associates a name and a value. The Globals window is used to create new global variables or to change the values of existing global variables. For more information, refer to "Setting Global Variables" in chapter 6, "Node Panels and Parameters."

Search Node(s)...

This command opens a panel that enables you to search for nodes by name using literal strings or cshell regular expressions. The Search command is especially helpful when working with large, complex node networks. For more information about how to get full use out of the Chalice search function, refer to "Finding Nodes Fast" in chapter 7, "Managing Large Networks."

Custom Networks...

This command opens a window that lists all items currently available in the Custom node menu, and enables you to add custom networks to or delete them from the list. For more information about creating and using custom networks, see "Creating Custom Networks" in chapter 7.

GROUP MENU

The Group menu allows you to collapse nodes into a group, represented by a single group-node icon, as described in the Grouping Nodes section of chapter 7, "Managing Large Networks."

Group (ALT g)

The Group command replaces all selected nodes with a Group node, which contains the selected nodes. The Group node retains any input and output connections to nodes outside the group.

Ungroup (ALT G)

This command "ungroups" a selected Group node; that is, the Group node is replaced by whatever nodes it contained. All internal connections (connections among the nodes in the Group node) and external connections (connections to nodes outside the group) are retained after the Ungroup operation.

If necessary, Chalice will append a digit to the names of nodes after ungrouping to ensure that all nodes in the current workspace have unique names.

Push In (ALT +)

This command allows you to view and manipulate the contents of a selected Group node. In effect, the Push In command takes you "down" one level. The workspace and Radar Map views change to display the nodes contained in the Network node you have pushed into.

Note:

The Level Indicator on the Main Menu bar always displays the current level of the workspace.

Pull Out (ALT -)

This command pulls you out of the current Group node view by taking you "up" one level.

LAYOUT MENU

The Layout menu enables you to specify a number of display options for the workspace, such as how connections between nodes are drawn, whether to operate in zoom mode, or whether to enable audio. These settings affect the entire existing layout, as well as any new nodes that are created. This menu also provides commands to control the display of selected nodes in the layout, such as to hide or show node icons. If a menu item is checked, it is enabled.

Monitors to Front (ALT W)

This command is useful on IRIX systems in those instances when monitor windows become hidden behind the Chalice main interface. Invoke "Monitors to Front" to bring all open monitors, as well as other Chalice windows such as preference panels, to the front of the interface.

Update All Icons (ALT U)

This command will instruct Chalice to update the thumbnail view of the node image for every applicable node in the workspace (that is, for every node with its frame icon displayed).

Every node that is displaying an image which does not reflect the current node data will have a red box drawn around the frame icon. This box disappears once the icon image is updated.

Show Icons of Selected Nodes (ALT N)

This command turns on, or shows, the frame icon display of every node currently selected.

Hide Icons of Selected Nodes (ALT H)

This command turns off, or hides, the frame icon display for all currently selected nodes.

Tip:

There are two advantages to hiding selected node icons. One is that such nodes take up less room in the workspace which, in a large network, allows you to see more nodes at once. The other reason is that you can save processing time when updating icon displays if unnecessary node icons do not have to be updated.

Change Size of Selected Nodes

This command enables you to set all selected nodes to a specific size: Small, Medium, or Large. (To change the size of any new nodes you create, reset the default node size in the Node Preferences panel, which is described in chapter 11, "Setting Preferences.")

Double Buffer (IRIX Only)

Check this box to turn on double buffering for smoother display of nodes on workstations running IRIX. (Uncheck this option to turn double-buffering off.) The default state is "on."

Link Straight (ALT I)

Check this box to redraw the connecting links between nodes as straight lines. (Uncheck to redraw the links using right angles, which is the default.)

Full Layout (ALT F)

Check this box to expand the viewable workspace. The node panel, radar view, and all other interface items below the menu bars will disappear and the workspace will expand to fill the screen. (Uncheck to return to the default layout.)

Zoom Layout (ALT Z)

This command puts the workspace into zoom mode to provide a macroscopic view of the grail network. The Zoom command is especially useful when working with large grail networks that cannot otherwise be viewed in their entirety at one time.

In zoom mode, each node is represented as a simplified icon labeled with the node name (but if you zoom out far enough, the node name will disappear). Use the mouse to control the zoom function as follows:

- **Zooming In and Out:** Alt-middle-click (hold down the Alt key and click the middle mouse button) and drag upward in the workspace to zoom in (to get a closer view) or downward in the workspace to zoom out (to see more of the network).
- **Directional Navigation:** To move around the network in zoom mode without changing the level of magnification, click the left, middle, or right mouse button and drag the workspace in any direction.

Sounds

This box, which is checked (on) by default, turns on audio prompts and system alerts to notify you of monitor updates, errors, etc.

New Layout (ALT I)

This command rearranges nodes in the workspace into a grid based on node hierarchy. It places the Exec node (if one exists) in the upper-right corner of the workspace. All upstream nodes are arranged in columns based on node and connection types.

You cannot undo this command. Be certain that you want Chalice to rearrange your layout before you issue this command. You can always drag nodes back to their original location; however, with a large, complex layout, this can be time-consuming. You can also revert to the last autosaved version of the file; see “Revert” in the File Menu.

PREFERENCES MENU

The Preferences menu enables you to access a series of panels to change the default settings for the following:

- **Monitor Preferences**: Change defaults for monitor interface, such as menu bar display, display LUT to use, monitor modes, etc.
- **Flipbook Preferences**: Change default interface and playback control settings.
- **Node Preferences**: Change default size, icon display, and menu display for new nodes.
- **Proxy Preferences**: For Chalice monitors, redefine what fraction of full size is represented by Monitor Medium and Monitor Low.
- **Data Format Preferences**: Specify default assumptions Chalice should make about the image data you use most often, such as whether it is premultiplied.
- **Autosave Preferences**: Turn function on or off; change default frequency of autosaves and location of autosaved files.

> **Note:**
>
> For detailed descriptions of each panel, refer to chapter 11, "Setting Preferences."

The Preferences menu also provides options for workspace display as well as buffering options in node monitors, which are described in the following paragraphs.

Continuous Updates

This option modifies the operation of two display update functions: Icon Auto Update and Monitor Auto Update. When the Continuous Updates box is checked, Chalice recalculates and updates the affected displays continuously, after each modification or user action.

For instance, if you are viewing a monitor with Monitor Auto Update on, and you use a tuner to adjust a node parameter, the Continuous Updates option will update the display not only after each change to the parameter setting, but also during each incremental move of the tuner, without waiting for you to release the mouse button.

> **Note:**
>
> This option substantially slows performance, especially when working with large networks, which is why it is unchecked by default.

Follow Cooks (ALT F)

With this box checked, Chalice will scroll the workspace view automatically to follow the progress of the current "cook" operation across the grail network. Follow Cooks is initiated when validating or executing a network from the Exec node, as well as when the cook button is pressed in a monitor window.

Buffer Auto Increment (ALT I)

Chalice has multiple buffers for storing images that are displayed in node monitors. This option, which is checked by default, specifies that each new image that is processed for display be stored in the next buffer in line, rather than overwriting the contents of the last buffer used. For more information about image buffers, see chapter 8, "Monitors and Flipbooks."

Buffer Persistence (ALT P)

The Buffer Persistence option specifies that an image remain stored in a buffer, even when it is not currently being referenced; that is, when it is not currently being displayed in a node monitor. This enables you to access the contents of the buffer again without re-cooking the image. However, this option uses more memory and the default state is unchecked (off).

ADMIN MENU

The Admin menu enables you to obtain licensing information, change cache settings, and to rescan your storage device to verify all Disk Input node data.

Show My License Info...

This command displays a panel of information about the license used for the current copy of Chalice you are running on your workstation.

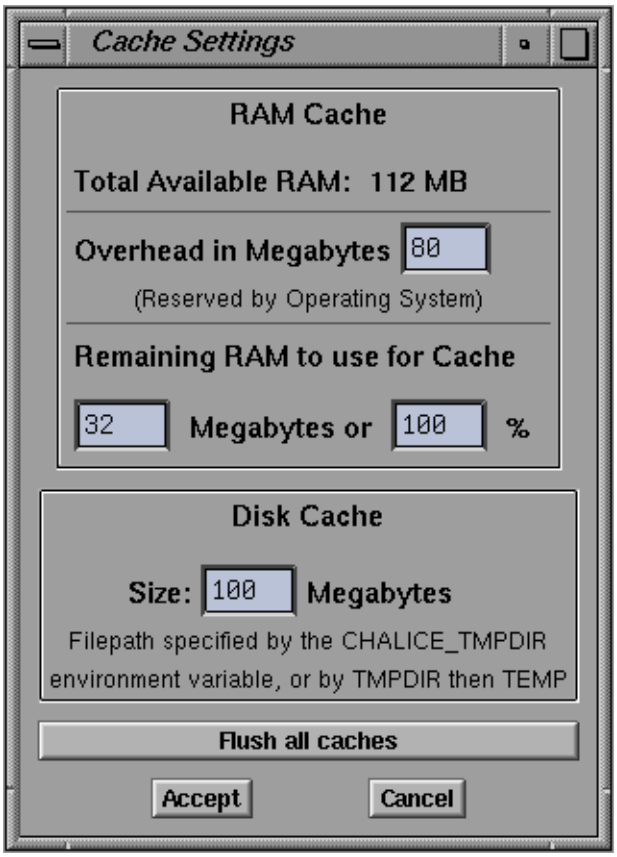
Show Network License Info...

This command displays a panel of information about the licenses currently being used and currently available on your network.

Cache Settings...

This command is used to access the Cache Settings panel, which enables you to control how much RAM and disk space are allotted to Chalice for caching. Chalice will run much faster when caching is used effectively, because it enables the software to store computations for re-use rather than recomputing data every time it is requested.

For an overview, refer to “Memory Usage in Chalice” in chapter 4.



Chalice can cache image data in RAM or on your hard disk. RAM caching is the first choice, because it is significantly faster. If Chalice runs out of RAM, it will then resort to disk caching. If both RAM and disk storage are exhausted, the data will have to be recomputed the next time it is requested.

During the execution of a final render, Chalice attempts to cache only the data needed to complete the render successfully without recomputing, because final renders are typically higher resolution and therefore more memory intensive. If Chalice runs out of both RAM and disk cache space during the execution of a final render, it will cease to render the shot and you will have to reset the RAM and disk cache values in the Cache Options panel to complete it.

RAM Cache

The RAM Cache parameters display the total RAM available on your system, and enable you to change the default value for how many megabytes of that total should be reserved for the operating system. The remaining RAM is displayed in a pair of data entry fields, which you can change using either megabytes or a percentage of the total remaining RAM.

Disk Cache

This Disk Cache parameters indicate the amount of disk space, in megabytes, that Chalice can assume is available for caching. The specified disk space must be available on the disk to which the environment variable CHALICE_TMPDIR points. If that variable has not been set, Chalice will check the TEMP or TMP variable.

Flush All Caches

Press this button to flush all data from all the caches for all nodes, after which all image data will have to be recomputed.

Rescan All Disk Input Nodes

This command will verify all input sequences (both full resolution and proxy resolution files) specified in every Disk Input node in the current grail file.

This is essentially the same disk rescanning operation that is performed to update individual input sequences when you press the “Rescan Disk” button for a file in a Disk Input node. The Rescan All command saves you from having to press every Rescan Disk button in every Disk Input node by hand, however, Rescan All may take a very long time to process, depending on the number of inputs to the grail file.

HELP MENU

The Help menu provides access to the Chalice help systems and enables you to report problems with the Chalice software or documentation.

Online Manual (ALT h)

This command accesses the Chalice Online Help System, an online version of the Chalice User Guide. Use the extensive hyperlinked contents and index to help you find the information you need.

> **Note:**
>
> Context-specific help for each node is also available by clicking the Help button located on every node panel.

Writing Plug-ins...

This command accesses extensive documentation for Chalice plug-in developers. Like the General help, the Plug-in help is in html format.

Help Tags

Check this option to turn on the "Help Tag" feature, which displays context-specific help text in popup displays when you move the cursor over certain parameter buttons and other program features. The Help Tag option is selected by default.

Version (ALT V)

This command displays the current version number of Chalice in the System Feedback area at the bottom of the Chalice interface.

> **Tip:**
>
> Be sure to check the current version number of the Chalice software you are using before contacting Silicon Grail with any questions or problems.

Report Problems (ALT p)

This command opens a window in which you can report problems you may encounter while using Chalice.

Fill in the information as indicated in the reporting window. Save and exit the window when you are done and Chalice will automatically email the information to Silicon Grail. Exit without saving to cancel.

> **Note:**
>
> Jot is the default editor used on IRIX for the reporting system. If you would prefer to use a different text editor, reset your EDITOR environment.

11 SETTING PREFERENCES

The Chalice Preferences menu enables you to change the default settings in Chalice. This menu is described generally in chapter 10, "Main Menu Bar." However, the Preferences menu also enables you to access a series of panels in which you can change the defaults for various categories of functionality in Chalice. These panels are explained in detail in this chapter:

Monitor Preferences ..... p. 11.2
Flipbook Preferences ..... p. 11.4
Node Preferences ..... p. 11.6
Proxy Preferences ..... p. 11.7
Data Format Preferences ..... p. 11.8
Autosave Preferences ..... p. 11.10

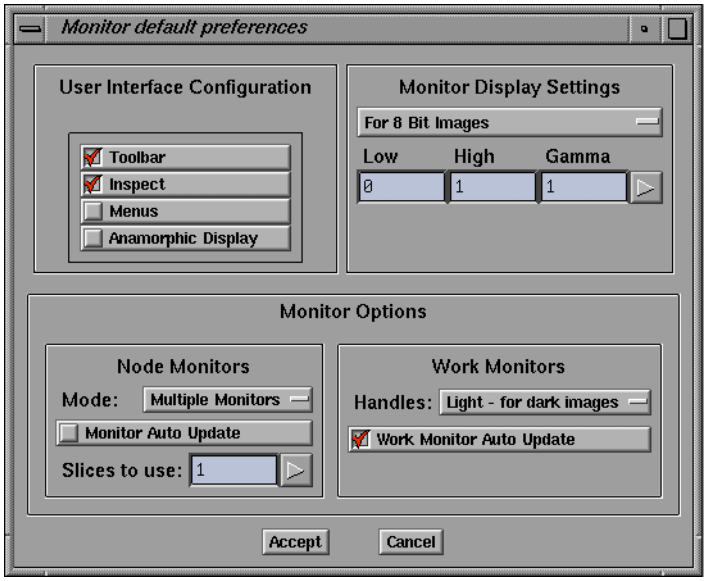
Saving Changes to Preference Settings

To save new settings in preferences panels for the current session and subsequent sessions (when you restart Chalice), click the **Accept** button. To dismiss the panel without changing any settings, click the **Cancel** button.

MONITOR PREFERENCES

You access the Monitor Preferences panel from the Chalice Preferences menu. This panel enables you to set default states for Chalice node monitors and work monitors.

- The upper left section, **User Interface Configuration,** allows you to control the default display of optional toolbars, menus, and panels.
- The upper right section, **Monitor Display Settings,** enables you to set the defaults for displaying 8-bit, 16-bit, and floating point imagery.
- The lower section, **Monitor Options,** enables you to set preferences pertaining to monitor operation, for both node monitors and work monitors.



User Interface Configuration

This section features checkboxes that enable you to change the default configuration of the user interface for Chalice monitors. Simply check the boxes of the monitor interface elements that you want to activate by default.

For example, you can specify that monitors always open with the Inspect panel displayed by checking the “Inspect” box.

Monitor Display Settings

This section enables you to reset the default values that Chalice applies to the **High**, **Low**, and **Gamma** display parameters, which appear across the bottom of the monitor toolbar.

A menu allows you to set the defaults for displaying 8-bit, 16-bit, and floating point images. You can key values directly into the Low, High, and Gamma parameters, or you can use the associated popup menu either to select preset values for these parameters, or to select a LUT that overrides these parameters.

If you are setting the defaults for 16-bit images, an additional parameter, **Icon Display Shift**, will appear. This parameter applies only to node icon image display in the node’s Frame Icon, and only to nodes containing 16-bit images.

The default value of 8 is best for viewing most 16-bit imagery in icon view. The exception is Cineon Linear imagery, which will appear indistinct or black.

Note:

Cineon Linear refers to images converted from log space using the optional "insert" method in the Disk Input node, which does not stretch the data across the entire range of the 16-bit colorspace.) Resetting the Shift value to 4 is best for displaying Cineon Linear in the Frame Icon; however, other 16-bit linear imagery will appear blown out.

Monitor Options

This section enables you to set defaults for various monitor options.

Node Monitors

Node monitors are the standard Chalice monitors that are accessed by selecting Monitor Full, Medium, or Low from the local node menu of a node.

Monitor Mode

This menu sets the default mode for monitor operation: Multiple Monitor or Single Monitor mode. In Multiple Monitor mode, which is the factory default, you can keep multiple monitor windows open simultaneously to display imagery at various stages of processing.

Select Single Monitor mode to view all your imagery in one monitor, if you prefer. Refer to chapter 8, "Monitors and Flipbooks," for a complete description of how to use monitors in each mode.

Monitor Auto Update

Check to specify Auto Update as the default state for any node monitors that you open. When Auto Update is on in a monitor, the image is updated each time you change a parameter setting.

Note:

You can override the current setting in any monitor by right-clicking on the cook button (eye icon) to toggle to the opposite state.

Slices to Use

This field enables you to reset the default monitor slicing level, which is 1 (that is, no slicing). Slicing can improve the speed of rendering and updating large monitor images by processing the image data in slices, where each image slice appears in the monitor as it is computed.

The popup menu associated with this field enables you to select a recommended number for various image resolutions, or you can key in a specific number of slices.

Work Monitors

Work monitors feature enhanced, node-specific functionality. They are accessed by clicking a plus icon in the node panel of those nodes that offer work monitors.

Monitor Handles

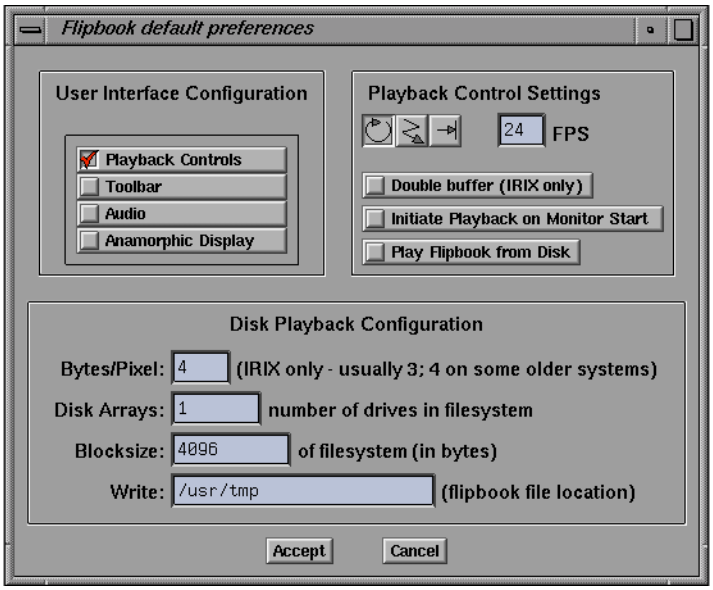
A number of work monitors provide drawing, cropping, and other tools superimposed over the monitor image. The Monitor Handles menu enables you to improve the visibility of these tools based on the overall brightness of your imagery. You can select "dark handles, for light images" or "light handles, for dark images."

Work Monitor Auto Update

Check this option to specify Auto Update as the default state for any work monitors that you open. When Auto Update is on in a work monitor, the image is updated each time you change a parameter setting.

FLIPBOOK PREFERENCES

You access the Flipbook Preferences panel from the Chalice Preferences menu. This panel enables you to set default states for Chalice flipbooks.



User Interface Configuration

This section features checkboxes that enable you to change the default configuration of the optional toolbars, menus, and panels in the flipbook interface. Simply check the boxes of the interface elements that you want to activate by default.

Playback Control Settings

This section enables you to change default settings for the following playback control options in flipbooks:

- the frame rate (fps);
- the type of repeat to use, if any;
- whether to use double buffering (IRIX only);
- whether Chalice should initiate playback automatically as soon as the flipbook is available; and
- whether flipbook data should be written to disk for playback, instead of to RAM, as the flipbook is created.

Disk Playback Configuration

These parameters are used to change defaults for saving flipbooks to disk to enable the fastest possible playback performance.

Bytes/Pixel

The “Bytes/Pixel (3 or 4)” parameter is set automatically by Chalice, based on the type of hardware being used. For example, some older SGI machines will run flipbooks faster when set to 4; newer systems, when set to 3. To ensure that a flipbook created on a newer SGI system would run well on an older system, however, you might reset this value accordingly.

Disk Arrays

When you are striping data across a disk array, it is essential that you enter the number of drives into this parameter field. Chalice will use this information to segment the data properly for faster playback.

Blocksize

The default value in this parameter field is 4096, which is the IRIX default. The correct block size is essential to enable Chalice to write flipbooks to disk in a form that will provide optimal playback speed. If your disks have been formatted at a different block size, change this parameter to match.

Write Location

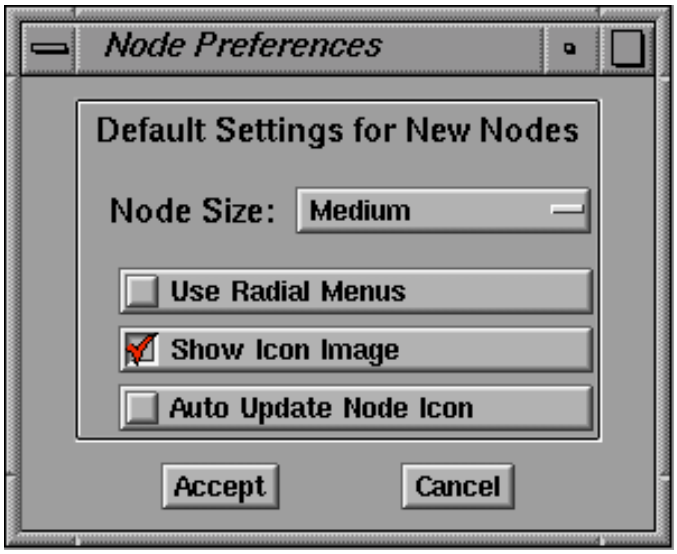
Use this parameter field to change the default location to which flipbooks will be written.

NODE PREFERENCES

This panel enables you to reset the following defaults for new nodes.

Note:

These node preferences may be changed for any individual node using the Local Node Menu button on the node itself without affecting the default settings.



Node Size

Select Small, Medium, or Large for the size of new nodes; the default is Medium.

Use Radial Menus

Check this box to make the Radial Menus option the default for displaying node popup menus. When checked, any menus accessed from a node in the workspace will appear in a radial layout. Otherwise, Chalice will display a traditional, single-column menu.

Show Icon Image

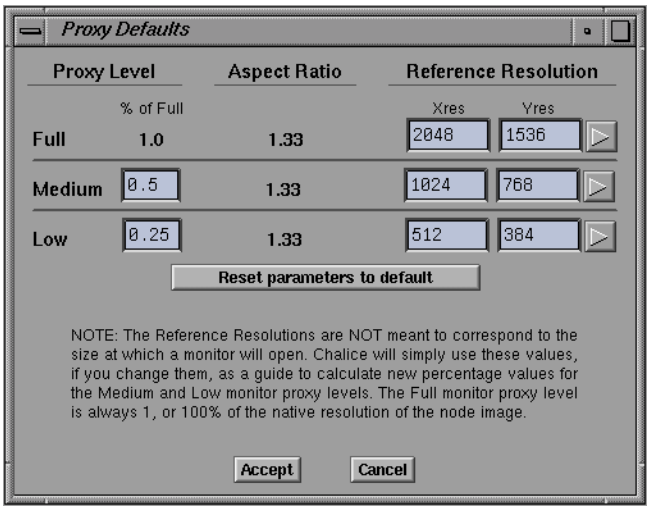
This item enables you to specify whether you want to create new nodes with the frame icon visible. The frame icon displays a thumbnail version of the image that would appear in the node monitor. Turn it off to create new nodes without a frame icon to save room in the workspace.

Auto Update Node Icon

Check this option to update the thumbnail images in the frame icons for any affected node each time you change a parameter setting.

PROXY PREFERENCES

This panel enables you to change the definition of “medium “ and “low” resolution when opening monitors in Chalice. Full resolution always represents 100 percent of the image size at a particular node. Full resolution cannot be redefined. Medium and Low represent fractions of 100 percent, and these fractions can be changed in the Proxy Defaults panel.



The size of Medium and Low monitors is usually changed when you are using file proxies for some or all of your source imagery. To take best advantage of the availability of file proxies, the medium and low monitor resolutions should be redefined, if necessary, to match the file proxy sizes so that Chalice does not have to scale the proxy inputs up or down a little to match the monitor size requested. (Refer to “Using Proxies in Chalice” in chapter 12 for more information.)

Proxy Levels

The factory default is that Medium represents 50 percent of full, and Low, 25 percent of full. Two methods are provided for changing these values:

- you can enter a new fraction (in the range of 0 to 1) into the Medium and/or Low Proxy Level fields, or
- you can use the Reference Resolution parameters to have Chalice calculate a new fraction for you.

Reference Resolutions

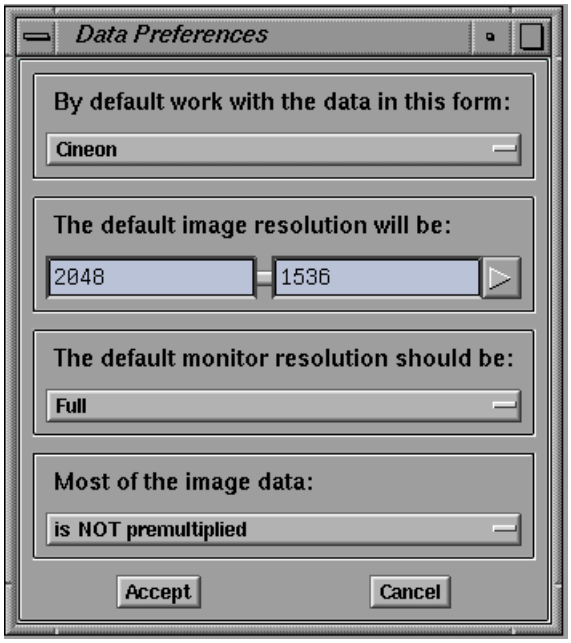
These parameters are provided for convenience, when you know that you want a monitor proxy to match a specific resolution, such as NTSC (720 x 486). Rather than getting out a calculator to figure out what fraction 720 x 486 is of, say, 2048 x 1536, you simply select the appropriate resolutions from the popup menu, or key in exact pixel values for the X and Y fields. Chalice updates the Proxy Level parameters for Medium and Low.

Aspect Ratios

In addition, the aspect ratio of each resolution is displayed so that you can avoid using resolutions with different aspect ratios, if need be.

DATA FORMAT PREFERENCES

This menu item enables you to access the Data Preferences panel, in which you can set default values to match the type of data you will be working with most often, so that the nodes you create will reflect these preferences in their parameter settings.



For example, if you specify a default image resolution of 640 x 486, any new Color node you create will adopt this default size. If you specify that most of the image data is premultiplied, any Mcomp node you place will use this as the default setting in each layer.

The Data Preferences panel enables you to specify the default values to use for

- the form of the data (i.e., Cineon, 16-bit linear, 8-bit, etc.)
- image resolution
- work monitor resolution
- the premultiplied status of image data

By default work with the data in this form:

A popup menu enables you to select one of the following forms of data as the default form to work with: Cineon, 16-bit linear, 8-bit linear, 8-bit NTSC, 8-bit PAL, float linear, or float Cineon.

For example, there are three different 8-bit linear options that you can specify as the default data form: 8-bit linear, 8-bit NTSC, and 8-bit PAL. The difference among them is the luminance function that will be pre-selected in any nodes that use Luminance.

This means that if 8-bit linear is the selected default form and you place a Channel Swap node, the Luminance menu in the Channel Swap node panel will be preset to Film; if 8-bit NTSC is the default, however, the Luminance menu would be preset to NTSC.

The default image resolution will be:
You can enter a numerical value for each dimension in the data entry fields to specify a default resolution for image data, or you can use the associated popup menu to choose a resolution from the resolution list.

For example, if you select 2K film resolution as the default, the Resolution parameter of any new Color node you create will default to 2048 x 1536.

The default monitor resolution will be:
This menu enables you to change the default resolution in the "At Resolution" parameter menu that accompanies every Work Monitor button. The default is to open work monitors at Full resolution, however you can select Medium or Low instead.

Most of the image data:
This popup menu enables you to specify the default alpha status of RGBA image data: **premultiplied**, **not premultiplied**, or **Ultimatte**.

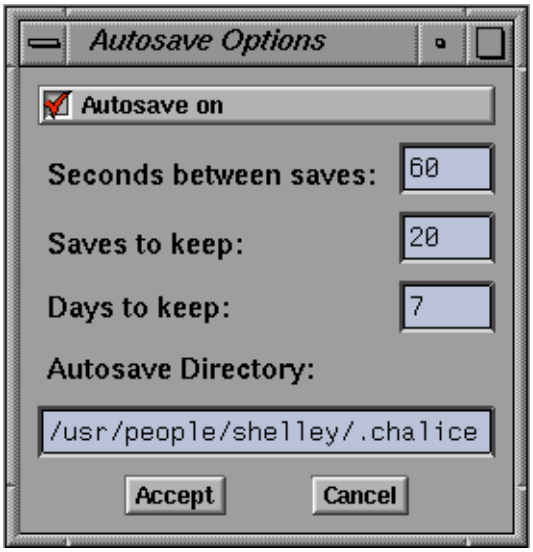
Every Mcomp, Zcomp, Atop, Inside, Outside, or Over node that you create will use the selected option as the default state for every layer. (Mcomp, Zcomp, and Over offer all three options; however, Atop, Inside, and Outside only offer the choice between premultiplied and not premultiplied.)

For more information about the significance of premultiplication to compositing, see "About Premultiplication" in chapter 15, "Composite Nodes."

AUTOSAVE PREFERENCES

Autosave is a Chalice feature that automatically saves a copy of the currently open grail file at regular intervals. The Autosave panel enables you to turn autosave on and off as well as specify how your grail files will be autosaved.

To access an autosaved grail file, use the Revert command. Revert can be found under the Chalice File Menu, which is described in chapter 10, "Main Menu Bar."



Autosave On

When the **Autosave on** box is checked, the autosave feature is enabled.

Seconds Between Saves

You can use the **Seconds between saves** data entry field to change the default setting, which will save the current file every 60 seconds.

Saves to Keep

Autosave creates a cache of incremental saves of the current file, up to the maximum number that you specify. The default is 20 autosave files. To change the maximum, type a new value into the **Saves to keep** data entry field.

After the maximum number of saves is reached, Chalice deletes the oldest version when a new file is added.

Days to Keep

Autosaved files are *not* deleted automatically whenever you quit Chalice. To avoid the accumulation of unnecessary files on your hard disk, however, Chalice will delete all autosaved files older than the value specified in the **Days to keep** field. The default value is 7 days.

Tip:

To have Chalice delete all autosaved files whenever you quit Chalice, enter 0 as the value in the "Days to keep" field.

Autosave Directory

The files are saved into the directory specified in the **Autosave Directory** data entry field. The default location is $HOME/.chalice; you can specify a different directory by typing a new pathname into the field.

12 I/O NODES

The I/O nodes are the Chalice input and output nodes, which are used to import and create imagery and to render modified images to disk. Chalice networks generally begin with one or more Disk Input or Color nodes and end with a Disk Output node followed by an Exec node.

Unlike other nodes, the Disk Input and Color nodes do not have input connectors. They are source nodes which import or create imagery. The Disk Output node has a single input connector.

In addition to descriptions of the I/O nodes, this chapter also includes information about the image file formats that can be imported and exported by Chalice, and how to use proxy files in Chalice:

Image File Formats Supported by Chalice ..... p. 12.2
Using Proxies in Chalice ..... p. 12.5

The I/O nodes menu includes:

Color Node ..... p. 12.9
Disk Input Node ..... p. 12.12
Disk Output Node ..... p. 12.23

Parameter Tabs in the Disk Input and Disk Output Node Panels

The Disk Input and Disk Output node panels include more tabs than can be displayed across the node panel at one time. To select a tab that is not currently visible, access the Tabs popup menu, which lists every parameter tab available for the node.

Click and hold down the mouse button over one of the Tabs icons (located on the left and right sides of the panel) to display the Tabs menu. Select a tab from the menu list and release the mouse button.

IMAGE FILE FORMATS SUPPORTED BY CHALICE

The current version of Chalice imports and exports the file formats in the list below. However, you can add support for other image formats by writing a Chalice plug-in. (HTML documentation for plug-in developers is available from within Chalice by selecting "Writing Plug-ins…" from the Help menu.)

File Format	Default Extension	IRIX	NT
AVI Video for Windows	avi		✔
Adobe Photoshop 3.X	psd	✔	✔
Alias	als	✔	✔
Alias Camera Depth	depth	✔	✔
Alias Eclipse		✔	
Chalice Direct Image	cdi	✔	✔
DPX	dpx	✔	✔
Fido	cin	✔	✔
FRP	frp	✔	✔
GWH&A WHIP	pic	✔	✔
Iff/ILBM	iff	✔	✔
Jpeg/Jfif	jpeg	✔	✔
Mac PICT	pict	✔	✔
Maya IFF	miff	✔	✔
MS Windows Bitmap	bmp	✔	✔
Movie	mov	✔	
Quantel	qtl	✔	✔
Raw	raw	✔	✔
RLL	rll	✔	✔
SGI Image	rgb	✔	✔
Side Effects	pic	✔	✔
SoftImage	pic	✔	✔
SUN Raster file	img	✔	✔
Targa	tga	✔	✔
Tiff	tiff	✔	✔
Wavefront RLA/RLB	rla	✔	✔
Xwd	xwd	✔	✔
YUV	yuv	✔	✔
ZPIC	zpic	✔	✔

Image Format Notes

This section provides important information about certain image file formats which should be reviewed before importing and exporting these file types.

The Disk Input and Disk Output nodes feature specialized tabs for controlling aspects of the importation and exportation of certain image formats. In addition to DPX, Fido, FRP, and the IRIX-compatible movie formats described below, tabs are available for importing Maya, YUV (at NTSC, PAL, or HD resolution), and ZPIC as well as for setting compression options when writing Jpeg and Tiff files to disk.

Always review the default parameter settings in the format tabs for Maya, YUV, and ZPIC when importing these file types:

- Only the RGBA or the Z channel data of a 5-channel **Maya** file can be imported into any one Disk Input node.
- Unless the default is changed, **YUV** files will be converted to 8-bit data upon import.
- A resolution must be specified for **ZPIC** files.

A formatting tab will only be activated when the associated format has been specified for import or rendering (otherwise the tab is grayed out). The parameters in these tabs are described in the Disk Input and Disk Output node descriptions later in this chapter.

Cineon (DPX/Fido) Formats

Chalice imports and exports Kodak Cineon 10-bit log imagery in DPX or Fido format. By default, the Disk Input node automatically remaps Cineon files to 16-bit linear colorspace, after which they can be treated like any other 16-bit imagery.

However, you also have the option to choose other conversion methods in the Fido and DPX formatting tabs in the Disk Input node. You can convert the log data to floating point linear, or you can choose to "insert" the log data into 16-bit or floating point linear to avoid clamping the highs. Such data is referred to as "Cineon Linear." You also have the option not to convert the log data at all.

Note:

The available import/export conversion options are described in the "DPX and Fido Parameter Tabs" section of the Disk Input node description. It is highly recommended that all users working with Cineon files review this information. Even if you are experienced with log-to-linear conversion issues, it will still be profitable to review how such conversions are handled in Chalice.

For a general overview of the subject, refer to Image Representation in Chalice in chapter 4, "Key Concepts."

Viewing "Cineon Linear"

Although it is not the default option, Cineon Linear is a highly recommended conversion option for 10-bit log files to preserve the integrity of the image data. However, Cineon Linear will appear indistinct in node monitors using the default monitor display settings.

To view a Cineon Linear image, open the node Monitor and select one of the Cineon display LUTs or presets from the Presets/LUTs menu. Remember that these options alter monitor display parameters only, not the node data.

Cineon in the Frame Icon

If the image data in a node is native Cineon 10-bit log data or Cineon Linear (which is created by choosing one of the insert methods to convert the files to linear), the Frame Icon image will appear indistinct or black. The default Chalice setting for viewing node icons in the workspace is not optimized to display this type of data because the ideal display shift for Cineon Linear would make other types of linear data appear overexposed.

For more information, refer to chapter 8, "Monitors and Flipbooks."

Adobe Photoshop Files

Chalice cannot read native Photoshop files created in versions newer than Photoshop 3.X, because Adobe instituted layering with version 4. However, you can flatten a Photoshop 4.X (or later) file, save it as a tiff, and import the tiff into Chalice.

Chalice Direct Image Format

Chalice enables you to import and export images directly in Chalice's internal file format. Although Chalice Direct Image (CDI) format is not likely to be the final output format for your shot, it does provide a very fast way to import and export frames of imagery that have been modified in Chalice.

Because CDI files do not employ compression, they can be quite large and require a great deal of disk space. Therefore, you may wish to avoid using this format for film resolution images.

Chalice FRP Files

FRP, which stands for "Fast Realtime Playback," is an internal format that Chalice uses to create single-file image sequences for realtime playback (hardware permitting). An FRP file can be created not only by specifying the format for rendering in a Disk Output node, but also directly from a flipbook, in which case the current flipbook sequence will be rendered to disk as an FRP.

To play an FRP, it is not loaded into a Disk Input node. Instead, it can be accessed from within Chalice at any time by using the "Play FRP Movie" command in the Chalice File menu, which will load the sequence into a Chalice flipbook for playback.

Movie Formats

The type of movie file format that can be imported and exported by Chalice depends, in part, on the system software your workstation is running:

- Chalice users running NT can import and export AVI files.
- Users on IRIX systems can import and export SGI, QuickTime, or MPEG.

USING PROXIES IN CHALICE

All Chalice Disk Input nodes provide the option to create and/or 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.

Another reason to use proxies is if the full resolution files are not available yet—for example, if all of the background frames haven't been digitized or if some of the finals of the CG elements haven't finished rendering.

"On-the-fly" proxy views have always been available in Chalice by bringing up a monitor image at a fraction of full size. In such cases, Chalice accesses the full resolution image and scales it for viewing in the monitor. However, it can be considerably faster to access lower-resolution files. Also, being able to scale to different resolutions in Chalice monitors does not address a situation in which the full resolution files are not currently available.

There are two main components to using the Chalice proxy system:

- creating or specifying proxy files for an input or inputs
- selecting a proxy level to work in globally

The actual proxy files are specified (or created) using the Disk Input node. The Chalice main interface also offers a global switch called "Proxy Input" that is used to specify the proxy level to work in for the grail file as a whole.

> **Note:**
>
> Unless you enable proxies (by checking the "activate" box in the Proxy tab), Chalice operates exactly as if there were no file-proxy capability. The full resolution source imagery is always accessed, and scaled as needed to fit the requested monitor size.

Other points to consider to get the most out of working with proxies include

- how proxies affect monitor display,
- matching default monitor resolutions to proxy file resolutions, and
- keeping proxy files in sync with changes to full resolution inputs.

Activating File Proxies

1. **Always start by specifying the "Full Path" parameter of the Input tab.**
   Full Path files are the full resolution image files that will be used to render the final shot.

2. **Then enable proxies in the Proxy tab and specify a medium and/or low resolution proxy.**
   You can point to an existing file sequence to use, or you can press the Create Files button for that proxy level to create them.

   Proxy files must match the full resolution source files in bit depth and number of channels; otherwise an error is generated. This prevents modifications being made to proxy files that would be inappropriate if later applied to imagery of a different bit depth or number of channels.

3. **If the full path imagery is not available, change the Full Resolution parameter in the Proxy tab from "Use Source File" to "Specify Size" to specify what the full resolution and bit depth will be.**

Then proceed to point to the image files to be used for the proxies. (Obviously, if Full Path image files have not been specified in the Input tab, you will not be able to use the "create proxy" function for that imagery.)

Disabling a Proxy Input

You can disable proxies for a Disk Input at any time by deselecting the "Enable Proxy Files" box in the Proxy tab. When you disable proxies, the Full Resolution menu on the Proxy tab is automatically set to "Use Source File," because a full path sequence must be specified in the Input tab in that case.

Setting the Global Proxy Input Level

A global Proxy Input switch enables you to set the current proxy level to use: Full, Medium, or Low. This switch specifies which of the files in a Disk Input node will be accessed by the network, constrained by the variables discussed next.

Unless a proxy file has been specified for a source image in the Disk Input node, the full path image data is always used in the network. This is true whatever the global Proxy Input setting may be, making the switch irrelevant in this case. For example, if you bring up a Monitor Medium or Low, the source image is scaled down to match the requested size.

If proxy inputs are specified, Chalice will access them when it is more efficient—regardless of the Proxy Input level selected. For example, say that the global Proxy Input switch is set to Full and you open a node monitor at medium resolution. If the source Disk Input node has proxy files enabled, Chalice will use the file closest to the requested monitor size (scaling the image up or down a little if necessary to match the monitor size).

Note:

This means that you must keep all files specified in a Disk Input node in sync. Changes to full resolution inputs require that proxy files be updated or disabled.

If the global Proxy Input level is set to Low and you request a Monitor Full, Chalice will take the low resolution proxy input, if specified in the source Disk Input node, and scale it up, so that the resulting monitor image will look blocky. (To see the full resolution source image, the Proxy Input switch should be set back to Full.)

How Source Imagery Is Accessed from a Disk Input Node

(NOTE: this chart assumes that file proxies are enabled and both medium and low proxy files have been specified for the relevant Disk Input node.)

Monitor Size Requested:	Global "Proxy Input" Setting:		
	High	Medium	Low
Monitor Full	full resolution source image	medium proxy source image, *scaled up to full resolution*	low proxy source image, *scaled up to full resolution*
Monitor Medium	medium proxy source image	medium proxy source image	low proxy source image, *scaled up to medium resolution*
Monitor Low	low proxy source image	low proxy source image	low proxy source image

Defining Monitor Resolutions to Match Proxy Files

As described previously, Chalice will access the specified source file with the resolution closest to the requested monitor resolution and then scale it up or down if necessary to display it at the requested size. To avoid such scaling, you may want to redefine Monitor Medium and Monitor Low to match the proxy file resolutions for your current project.

Monitor Full is always 100 percent of the full resolution source image, but Medium and Low are both fractions of full resolution, and these fractions can be changed in the Proxy Preferences panel (which is accessed from the Chalice Preferences menu). By default, Medium is defined as half and Low as one-quarter of whatever full resolution is for the current node.

Say, for example, that you are working with 4K imagery and that your medium proxy files are 2K. However, your low resolution proxies are not 1K but typical NTSC resolution (720 x 486). Using the default Monitor Proxy values, a request for a Monitor Medium will display the 2K files (which are exactly half of full resolution), but a Monitor Low will access the video resolution files and scale them up to 1K (since Low is defined as a quarter of full resolution).

In this case, you can use the Proxy Preferences panel to redefine Monitor Low to be the same fraction of full resolution that a 720 x 486 image is of 4K imagery. (This fraction happens to be 0.175781. Chalice will do the calculation for you if you select a reference resolution in the preferences panel.)

Note:

This setting affects all node monitors opened at low resolution. Using the example of Monitor Low defined as the fraction 0.175781, bringing up a Monitor Low on a CG element that is 1K at full resolution will display an image that is 17.5781 percent of 1K resolution.

Other Tips for Working with Proxies

Proxy files are commonly used by specifying them in the Disk Input nodes, keeping the global Proxy Input switch at the default setting of High, and using Monitor Medium to access medium resolution proxies and Monitor Low to access low resolution proxies.

In this way you control access to the source images by selecting a monitor resolution. With the global Proxy Input switch set to Full, you can still work faster by opening a Monitor Medium for an input with a Medium proxy. When more precision is needed, you can access the full resolution file by choosing Monitor Full.

When working with proxies, it is also important to use fractions rather than pixels, where applicable, as the basis for spatial measurement values. For example, if you scale an image by specifying a percentage of full resolution in the Scale node parameters, this value will be equally valid for a proxy or a full resolution image.

Using Rescan Disk to Update Proxies

Proxy files must be kept in sync with any changes made to the full resolution source files. You can create new proxy files to replace the old ones by using the Create Files option in the Disk Input node. However, the updated proxy files you want to use may already have been created by someone else. In that case, use the Rescan Disk option for the proxy input to force Chalice to reread the disk and update accordingly. (Rescan Disk can also be used to update the full resolution input if necessary.) The rescanning process can take some time, depending on the sequence.

Chalice also provides a “Rescan All Disk Input Nodes” command in the Admin menu. This will rescan all Disk Inputs without having to go into each source node individually to press the “Rescan Disk” button. (Of course, the “Rescan All” command can take a long time to execute, depending on the contents of your grail file).

COLOR NODE

The Color node enables you to create either a solid color field or a gradient color ramp using the values you specify. The output of the Color node can be manipulated just like any other image data. It occupies no space on the hard disk unless you save it as an image file.

The Color node enables you to

- create horizontal ramps, vertical ramps, or solid colors
- use either RGB or HSV color space to specify colors
- specify the alpha channel value
- specify the resolution, bit depth, and number of channels for the output

Tip:

It is often convenient to save specific Color node settings as a preset by using the "Save As" option in the Presets menu, which is located just above the Info tab on the Node Panel. For more information, refer to "Using Preset Parameter Values" in chapter 6.

Using the Color Node

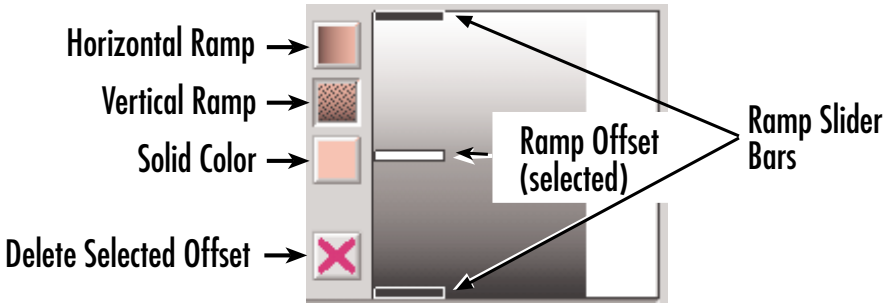
Start by selecting a solid field of color or a gradient (a horizontal or vertical ramp). Then select the colorspace you want to work in: HSV or RGB.

Next, use the color parameters (the HSV or RGB color bars) to specify the output color. You can also adjust the alpha channel value in the Alpha tuner.

Finally, specify the Resolution, Channels, and Bit Depth per channel of the output.

Creating a Ramp

When you select a gradient, ramp slider bars appear at opposite edges of the color box. You can adjust the color (and alpha) parameters for each bar individually. Click on a ramp bar to select it (the bar will highlight).



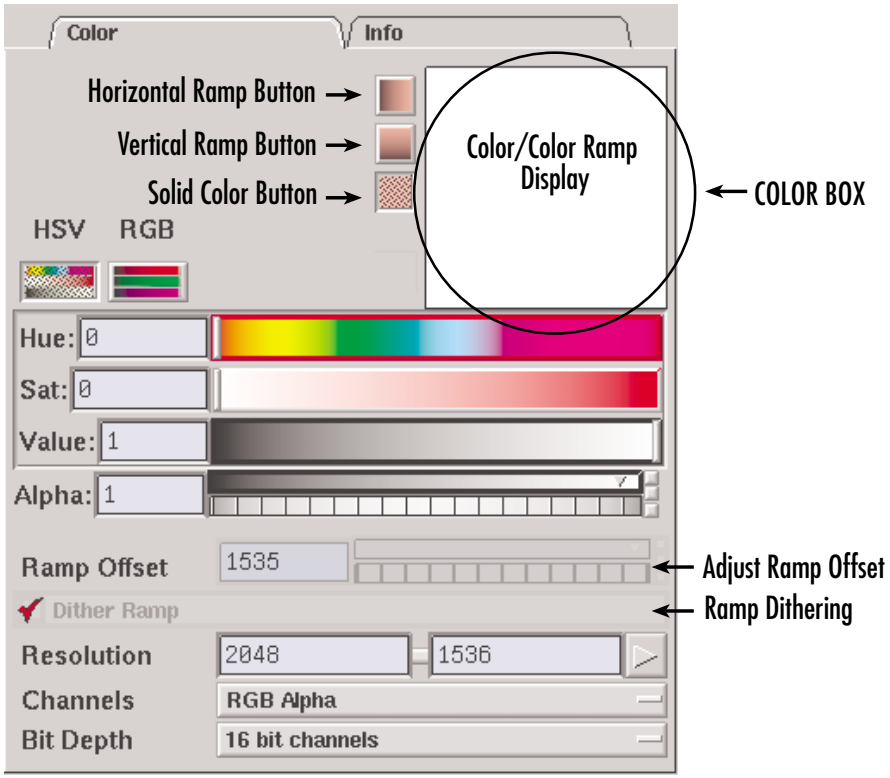
Ramp Offsets

You can create ramp offset slider bars by clicking in the color box. Each time you click, Chalice creates a new offset bar. To delete an offset, select the bar and then click the red X icon. (This icon only appears when a ramp offset is selected.)

You can drag these bars vertically or horizontally within the color box to manipulate the color frequency distribution, or you can use the Offset parameter. When an offset bar is selected, the Ramp Offset parameter will become active to enable you enter a precise numerical value (in pixels) for the offset.

Color Parameter Tab

The Color tab provides parameter controls for the color channel values of the color or color ramp, as well as the output channels, resolution, and bit depth.



Color Ramp/Solid Color Buttons

The two Color Ramp buttons and the Solid Color button enable you to specify a horizontal color ramp, a vertical color ramp, or a solid color. For example, if you need to create a solid background color for later use in bluescreen or greenscreen compositing, select the solid color button.

Color Box

This displays how the color or color ramp will appear in the output. If the Alpha value is adjusted, this channel will also be displayed (as a grayscale strip) in the color box.

When a ramp has been selected, the color box can be used to create ramp offsets: just click within the color box to create an offset bar, which can be adjusted by dragging it with the mouse or by using the Offset parameter.

HSV/RGB Colorspace Selectors

If you select **HSV** colorspace, the Color parameter tab displays parameters for Hue, Saturation, and Value. 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.

If you select **RGB** colorspace, the Color parameter tab displays parameters for Red, Green, and Blue. The RGB parameters are set in a range of 0–1, which represents the percentage of the total contribution of each color to use. This makes these parameters consistent whichever bit depth you choose for output from the node.

Alpha

The Alpha parameter enables you to control the alpha channel value. The Alpha channel retains monochrome matte data and determines the extent of coverage of an element at a given pixel.

Ramp Offset

You add an offset to a ramp by clicking in the color box to create a ramp offset slider bar. The Ramp Offset parameter enables you to precisely adjust the position of the selected offset by keying in a numerical value or using the tuner to set the value, rather than by dragging the slider bars in the color box. The offset from the edge is expressed as an X (for horizontal ramp) or Y (for vertical ramp) pixel coordinate.

Dither Ramp

This checkbox becomes active when a horizontal or vertical ramp is selected and 8-bit channels have been specified for the output. In such cases, Chalice does the internal calculations in floating point math and then truncates to 8-bit, which can cause banding when a small color change is being stretched over a large area.

The Dither Ramp option adds random noise to the floating point value, before truncating, thus hiding the banding in noise. It is on by default, but if you should want to turn off dithering, simply uncheck the box.

Resolution

This parameter enables you to specify the width and height of the Color node output. You can key a pair of XY values into the data entry fields, or you can use the popup menu to access a list of many video, HD, film, and large format resolutions. When you select a resolution from the list, the corresponding XY values are entered into the Resolution parameter fields.

Channels

This parameter enables you to select which channels to output:

- RGB
- RGB Alpha
- Alpha only

Bit Depth

You can specify the bit depth per channel of the output by selecting an option from the Bit Depth menu (the default is 8-bit):

- 8-bit channels
- 16-bit channels
- floating point channels

DISK INPUT NODE

The Disk Input node enables you to load an image sequence from your hard disk or from a remote directory on a shared volume. Unlike most other Chalice nodes, you cannot feed data from another node into the Disk Input node. Rather, it is common to begin a Chalice network with a Disk Input node.

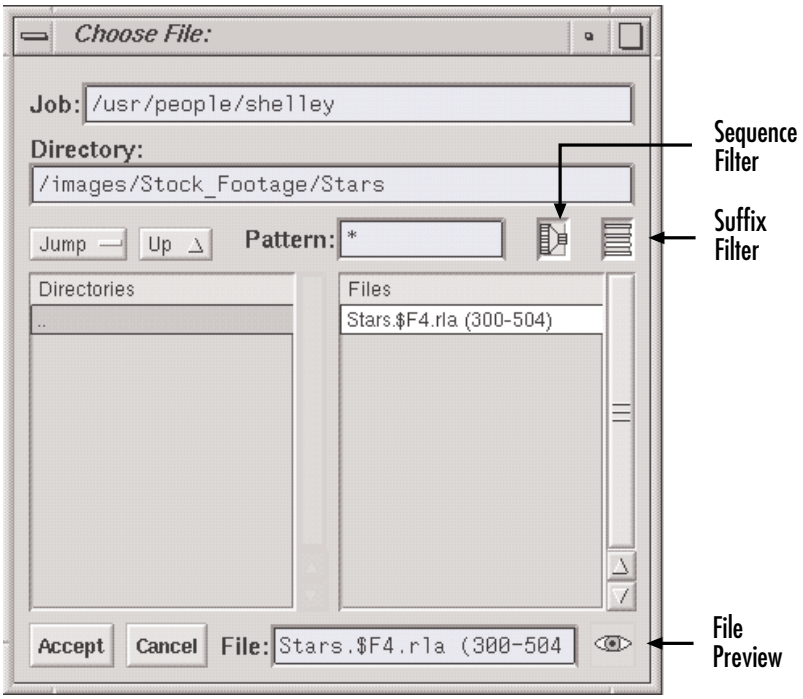
IMPORTING IMAGE FILES

You specify the imagery to import in the Input parameter tab of the Disk Input node by using either of the following methods:

- Type the complete pathname of the file to be imported directly in the Full Path field.
- Click the plus button to display a Choose File dialog box.

Using the File Chooser

The Choose File dialog box is a standard Chalice file dialog. You can key the name of the directory you wish to access into the Directory field or click the Directories entries to navigate through the various levels of your file system. For more information on using the navigational features of this dialog, see Opening an Existing File in chapter 5, "Building Chalice Networks."



The File Chooser dialog features buttons that facilitate locating and importing image files:

- Click the **File Preview button** to bring up a monitor display of the currently selected image file in the Files list. File Preview is used to display single image files, so sequence filtering (see below) must be turned off to select an individual file for previewing.

- Turn on **Suffix Filtering** to display only those files in the Files list that have legitimate image-file format extensions. Refer to the beginning of this chapter for a complete list of Image File Formats Supported by Chalice.
- Turn on **Sequence Filtering** to display one entry per sequence in the Files list. Take, for example, a three-second Cineon shot where each frame is a separate image file named as follows: "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 (1-72)." Turn sequence filtering off (the Sequence Filter button is a toggle) to display all of the files individually.

Note:

You will frequently encounter the **"$F" variable**, as in "filename.$F4.cin," e.g. 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 identifying the frames as Cineon-type files.

Once you locate the file you wish to import, 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 Choose File dialog will close and the selected images will be loaded into Chalice.

The Disk Input node panel will appear. The node panel will contain information specific to the imagery that has been loaded. If you click the Frame Icon on the Disk Input node in the workspace, Chalice will display a thumbnail image (an iconic representation of the image) in the Frame Icon.

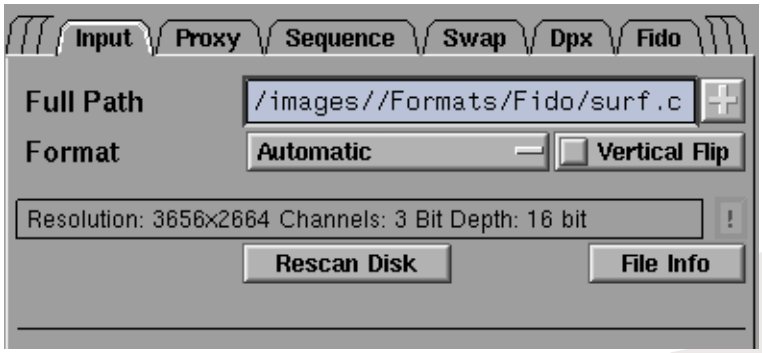
Parameter Tabs on the Disk Input Node Panel

The Disk Input node features a number of special-purpose parameter tabs (like all Chalice node panels, it also includes an Info Tab, which is described in chapter 6, "Node Panels and Parameters"):

- The first four tabs are used to specify files to import (**Input tab**), to work with proxy files (**Proxy tab**), to resequence frames (**Sequence tab**), and to add, delete, or swap channels (**Swap tab**).
- The rest of the tabs on the Disk Input node panel enable you to set options for specific file formats: **DPX, Fido, Maya, YUV,** and **ZPIC**. The parameters for these tabs are only available when the source frames are of the corresponding file format.

Input Parameter Tab

The Input tab is used to specify the full resolution image sequence to be imported.



Full Path

The plus button associated with the Full Path parameter opens a Choose File dialog box that enables you to specify the directory location and files you wish to load. Alternatively, you can enter the complete pathname into the Full Path data entry field.

Format

The Format button displays a popup menu of file formats that can be imported by Chalice. The default choice is "Automatic," which indicates that Chalice will examine the image files to determine their format. When it cannot determine the file type, an error message will appear in the Status field of the node panel. In such a case, use the Format menu to tell Chalice the file type of the image you are attempting to import.

Vertical Flip

Certain file formats invert the orientation of images. Click the Vertical Flip checkbox to invert the imagery. (This box can be checked or unchecked at any time to change the orientation of the image.)

Rescan Disk

When you press the Rescan Disk button, Chalice will rescan the source disk to verify that the image file data specified in the Path parameter is up-to-date. If not, Chalice will update the parameter.

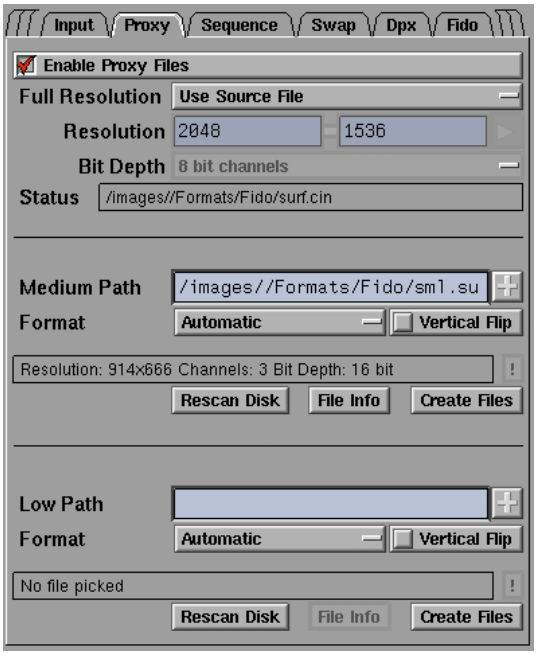
Note:

Chalice will perform a rescan automatically whenever the pathname in a Disk Input node is altered, or if a global used in the pathname ($JOB, e.g.) is redefined.

File Info

The File Info parameter enables you to view a list of every frame in the sequence. For each frame it displays filename, width in pixels, height in pixels, number of channels, bit-depth, file size in Megabytes, and file format. If there are gaps in the sequence, this view will identify the missing frames.

Proxy Parameter Tab



The Proxy tab is only used 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. If necessary, you can create proxy files directly from the Proxy tab, as described below under "Medium Path."

The parameters on the Proxy tab are described below. For a comprehensive overview of the subject, however, see Using Proxies in Chalice at the beginning of this chapter.

Enable Proxy Files

This checkbox is the master control for enabling the use of proxies in a Disk Input node. When you check the box to enable the use of proxy files, the rest of the parameters on the Proxy tab become available. These parameters enable you to specify information about the full resolution files, if they are unavailable, and to specify or create medium and low resolution proxies.

Full Resolution

You always start by pointing the Disk Input node to the full resolution files using the Full Path parameter on the Input tab. Relevant information about the Full Path image sequence is displayed here in a status bar.

If, however, the full resolution imagery is unavailable, you can change the Full Resolution menu choice from the default, "Use Source File," to "Specify Size." This will activate the associated Resolution and Bit Depth parameters, which enable you to specify this information for the full resolution files that will eventually be used for the final shot.

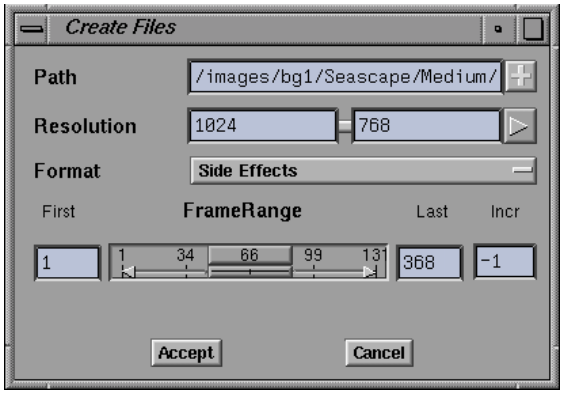
Medium Path

The Medium Path parameters enable you specify the files to be used as medium resolution proxies. These parameters work exactly like those located on the Input tab (refer to the Input Parameter Tab description), including file path, flipping the input, rescanning the disk to update the image file information, and displaying the file info. In addition, however, Medium path includes a "Create Files" option:

Create Files

In some cases, proxy files will already exist on disk or other storage device. In other cases, however, you have to create the proxy files yourself. The Create Files option enables you to create proxy versions of the full resolution imagery directly from this tab. (It is not necessary to connect the input to a Disk Output node and an Exec node to render them.)

When you press the Create Files button, Chalice displays a file creation dialog in which you specify the proxy pathname, resolution, and format, as well as the range of frames to create.



Low Path

The Low Path parameters enable you specify image files to be used as low resolution proxies. The procedure for specifying or creating low resolution proxies is exactly the same as that for Medium proxies, as described directly above.

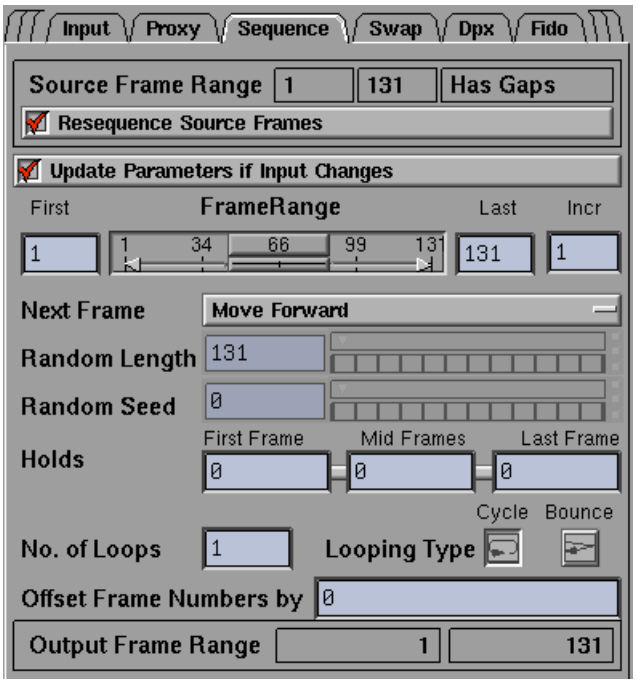
Sequence Parameter Tab

The Sequence tab of the Disk Input node has essentially the same functionality as the Sequence node, and you can always modify a sequence at any point in a grail network by using the Sequence node. However, this tab enables you to resequence frames immediately upon input, without resorting to another node.

In the Sequence tab, you can

- specify the range of frames in the selected sequence to import.
- control the direction in which the frames are accessed.
- hold frames and add loops.
- renumber frames as output from the Disk Input node.

The Sequence tab parameters are disabled by default. If you enable the tab by checking the Resequence Source Frames box, all the settings you specify in this tab become applicable until or unless the tab is disabled.



Note:

The **Source Frame Range** display (top of tab) always reflects the frames as accessed on disk. The **Output Frame Range** (bottom of tab) display updates to indicate any change in the range as a result of modifying the Sequence parameters.

Resequence Source Frames

This parameter is the master control that enables resequencing. Unless it is checked, none of the other Sequence tab parameters will be available for modification and any previous modifications will be ignored.

Update Parameters if Input Changes

By default, if the input sequence changes the Sequence tab parameters will update to reflect any changes in the sequencing of the source imagery. In some cases, however, you might not want Chalice to make this adjustment automatically. If so, simply uncheck this parameter and the Sequence tab will keep your settings intact.

Frame Range

The Frame Range parameters enable you to specify new starting and ending frames to access and to skip frames by changing the Increment value.

Tip:

When you load a sequence into the Disk Input node, the Source Frame Range display at the top of the tab indicates whether or not the sequence has gaps. If there are gaps, click the "File Info" button on the Input tab to get detailed information about each frame in the source sequence.

Next Frame

This popup menu enables you to change the order in which frames are accessed from "Move Forward," which is the default, to "Move Backward," if you want to reverse the order in which the specified source frames are accessed and, therefore, output from the Disk Input node to the rest of the network.

You can also instruct Chalice to "Pick Random Frames." The randomize option is commonly used for source frames consisting of noise or grain patterns. When Pick Random Frames is selected, the following parameters become available:

- **Random Length** enables you to enter the total length in frames that the sequence should be.
- **Random Seed** enables you to enter a seed value for the randomize operation.

Holds

First Frame will repeat the first frame in the specified sequence by the value entered in the field; **Mid Frames** will repeat every frame between the first and last frames by the value entered; and **Last Frame** will repeat the last frame. To double every frame, for example, you would enter a hold value of 1 in all three Hold fields.

No. of Loops

This parameter enables you to specify the number of iterations of the sequence. If you specify more than one iteration, specify how the sequence should be repeated using the Looping Type parameter (see next).

Looping Type

Cycle will repeat the sequence in order. For example, a sequence of three frames will repeat as 1, 2, 3; 1, 2, 3; 1, 2, 3; etc.

Bounce will repeat the sequence first forward, then backward. For example, a sequence of three frames will repeat as 1, 2, 3, 2, 1, 2, 3, 2,1, etc.

Offset Frame Numbers By

This parameter enables you to renumber the frames output from the node by creating an offset from the first frame number. The "Output Frame Range" display, located just below this parameter, updates automatically as the offset value is adjusted.

To create an offset, key an integer value into the data entry field. (By default, the offset is 0.) The value you enter is added to the current starting number: positive values increase the starting number, while negative values decrease it.

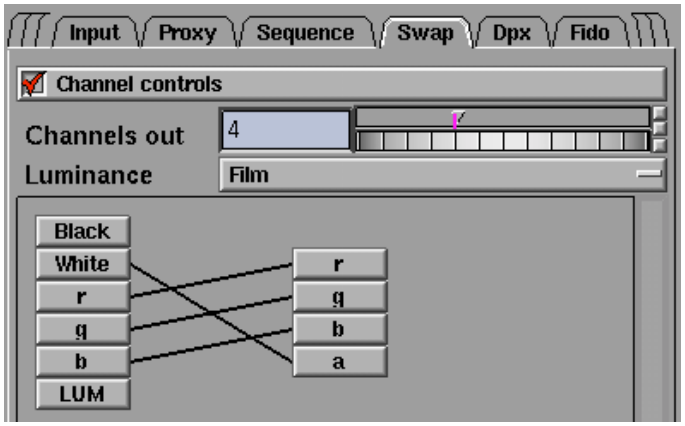
Say, for example, that you load an eleven-frame sequence numbered "file.0040.cin" to "file.0050.cin." By setting a negative offset of 39 (that is, -39), you would remap the sequence such that frame "0040" becomes accessible at frame 1 within Chalice, instead of at frame 40. Frame "0041" would remap to frame 2, and so on.

Note:

Adjusting the frame offset will not instruct Chalice to ignore gaps in the source sequence. An error message indicating that a frame file on disk is missing will still be generated; only the frame number at which the error occurs is offset.

Swap Parameter Tab

The Swap tab enables you to modify channel data in your Disk Input image. You can add channels, delete channels, and rearrange the channel configuration. (This is essentially the same functionality that is available in the Channel Swap node.) A common use of this feature is adding an alpha channel to an RGB image.



Channel Controls

Check this box to enable the channel configuration parameters to receive input. If this box is not checked, you will not be able to alter the channel data.

Channels Out

The Channels Out parameter allows you to alter the number of channels in the image. You can key in a number directly or use the tuner dial. An image must contain at least one channel and cannot exceed ten channels.

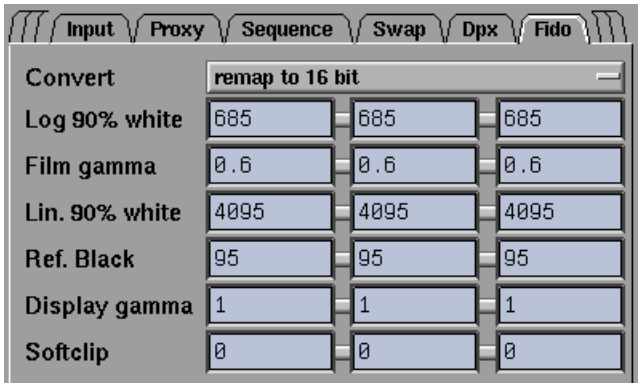
To rearrange the configuration of the channels, click and draw a connector line from the output channel (on the right) to the input channel (on the left).

Luminance

This parameter allows you to specify the type of luminance (Film, NTSC, or PAL) Chalice will calculate if luminance has been selected as an output channel. For more information on channels and luminance, see the Channel Swap Node description in chapter 13, "Tool Nodes."

DPX and Fido Parameter Tabs

The DPX Parameter Tab and the Fido Parameter Tab both feature the same set of parameters to facilitate the conversion of these Cineon 10-bit log formats to 16-bit or floating point linear space. (If you need to convert log files to an 8-bit linear data representation, use the Bit Reduce Node.) Therefore, both tabs are described here, illustrated by the Fido Parameter Tab.



Convert

By default, the Disk Input node converts Cineon log files to 16-bit linear using the "remap to 16-bit" option. The Convert parameter enables you to select another conversion option from the popup menu.

You can choose not to convert the data by selecting "None" from the conversion menu, as when it is necessary to demote the file to 8-bit space using the Bit Reduce node or when you want to perform an operation on the log data.

Or, the Cineon log files can be linearized to 16-bit or floating point colorspace by using one of two methods:

- Use the Cineon Softclip algorithm to **remap** the log space to 16-bit (the default) or floating point.
- Use the strict Cineon Linear conversion to **insert** the log data into 16-bit or float space.

Using "Remap to"

The default conversion is **Remap to 16-bit**, which stretches the log data in a constant curve to fill the 16-bit space. The Remap to 16-bit and **Remap to Float** options use the Softclip conversion method, which clamps values above Log White.

Note:

Depending on your imagery, you may want to adjust the Softclip parameter for the Remap options. When the Softclip parameter setting is left at its default of 0, the Log 90% White parameter value is mapped to 65535 in 16-bit, and to 1 in float.

Positive Softclip values create a breakpoint below 90% White, above which the slope of the conversion curve is "softened" to reduce clamping of the highs.

For a complete description of the Softclip conversion process, refer to the Bit Reduce Node description in chapter 14, or to appendix C, "Softclip Conversion."

Using "Insert into"

The **Insert into 16-bit** option converts Cineon log files to "Cineon Linear." The insert method is a strict conversion that does not stretch the data to fill the 16-bit space. Instead, the Log 90% White value (the default is 685) corresponds to the Linear 90% White value, which has a default value of 4095. Log values above 685 are not clamped when converted.

Note:

Setting the Linear 90% White value at 4095 is why Cineon Linear imagery will look indistinct or black in node monitors unless the default monitor display values are changed. To view Cineon Linear, select one of the Cineon display LUTs or the Cineon Monitor Preset (see chapter 8, "Monitors and Flipbooks," if you need more information about monitor display parameters).

The **Insert into Float** option linearizes the log data by inserting it into floating point space without clamping float values at 1 (as the Remap to Float option does).

Conversion Parameters

The following chart shows which parameters are applicable to each option in the Convert menu:

Parameters Applicable to Each Convert Option	Convert to Linear Options				
	None	Insert into 16-bit	Insert into Float	Remap to 16-bit	Remap to Float
Log 90% White		685	685	685	685
Film Gamma		0.6	0.6	0.6	0.6
Linear 90% White		4095			
Reference Black				95	95
Display Gamma				1.0	1.0
Softclip				0	0

All four conversion methods use the Log 90% White and Film Gamma parameters. The two Remap options use the Reference Black, Display Gamma, and Softclip parameters. Only Insert into 16-bit uses the Linear 90% White parameter. The values in the chart represent default parameter settings.

Note:

All of the default values for Cineon conversion are based on specifications published by the Cinesite Digital Film Center and Kodak Motion Picture & Television Imaging. These documents are available for downloading in Acrobat format from the Silicon Grail website (www.sgrail.com).

Log 90% White

The Log 90% White parameter enables you to modify the default RGB log values of 685 for the 90% white card.

Film Gamma

The Film Gamma parameter enables you to modify the default RGB conversion values (0.6) for film gamma. (The slope of the straight-line portion of the characteristic curve for typical negative film is known as the film gamma.)

Linear 90% White

The Linear 90% White parameter enables you to modify the default RGB linear values of 4095 for peak white (used in the default conversion to 16-bit linear color space).

Reference Black

The Reference Black parameter enables you to modify the default RGB log values of 95.

Display Gamma

The Display Gamma value is used in the Softclip conversion algorithm. The default value of 1.0 should not need modification in most cases.

Softclip

The Softclip parameter enables you to modify the default RGB values to reduce the effects of harsh clamping at the high end when remapping log data. The default value is 0 (no softclip). The Softclip parameter can be adjusted in the range of 0–100.

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. The slope of the distribution curve above the breakpoint is changed to remap the highs more gradually.

Tip:

Middle-click and hold one of the Ladder Bars for any parameter to adjust all three fields (R, G, and B) simultaneously. Ladder bars are the small horizontal bars connecting the RGB fields.

Move the mouse vertically to select an increment value from the Ladder popup, and then move it horizontally at that rung to change the values in the RGB fields by the chosen increment.

Maya Parameter Tab

The Maya parameter tab is provided for users who need to import five-channel (RGBAZ) Maya IFF (.miff) imagery. These Maya image files store the RGBA data as 8- or 16-bits per channel and the Z-depth (ZBUF channel) data as floating point.

To resolve the conflict such a mixed-depth image file would cause in Chalice, you must use the **Read Data** parameter to specify whether the RGBA channels or the ZBUF channel should be imported into the Disk Input node.

Tip:

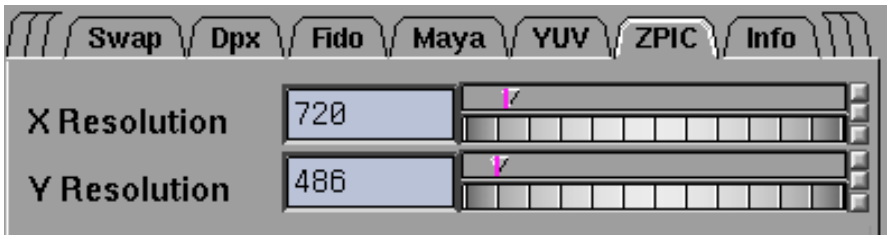
To access all five channels of data, select the same file sequence in two different Disk Input nodes and specify RGBA in the Maya tab of one node, and ZBUF in the Maya tab of the other.

YUV Parameter Tab

When YUV colorspace data is imported into Chalice, it is converted to floating point RGB data. However, because of the way YUV data is encoded, the conversion to RGB can result in negative numbers. Therefore, if the floating point RGB data is converted to 8-bit, any negative values will be lost. The YUV parameter tab enables you to specify how this data will be represented within Chalice.

Click the **Represent As** menu button and select one of the following options: 8-bit, 16-bit, or Float. The default is to represent YUV files as 8-bit per channel data.

ZPIC Parameter Tab



ZPIC is the SoftImage file format for z-depth channel data. Unfortunately, ZPIC files do not contain the necessary information in the header to enable Chalice to determine their resolution. The **X Resolution** and **Y Resolution** parameters in the ZPIC tab enable you to tell Chalice the resolution of the ZPIC file being imported.

Be aware that in some cases black space is omitted from the ZPIC file when it is written, which can have the effect of changing the resolution from what it was originally in SoftImage. The size entered in the Resolution parameters must be the size actually saved by SoftImage. Hopefully, the associated RGB file resolution can be referenced for the ZPIC file.

DISK OUTPUT NODE

The Disk Output node enables you to specify how a sequence of images will be written to your hard disk or other storage device.

When you are ready to write a sequence to disk, you connect the output of the final process node for the sequence to the input connector of a Disk Output node. The output connector of the Disk Output node is in turn connected to an Exec node for actuation (refer to the Exec Node description in chapter 13, "Tool Nodes").

EXPORTING IMAGE FILES

The Disk Output node accepts one input and includes parameters that allow you to specify name, directory, file format, and file conversion and compression options. These options can be specified for both final output and for any temps you may want to render.

Rendering Temps and Finals

During the creation of an effects shot, it is often necessary to create temporary composites ("temp comps"). Composited frames may be needed for evaluation purposes or for reference by another artist before the final shot is ready to be rendered.

Temp comps are usually lower resolution than the finals will be, and at times it may be preferable to render them in a different file format. The Disk Output node enables you to specify the name, directory, file format and compression options to use for temps separately from the finals of the same sequence. The file characteristics of finals are specified in the Finals Output tab, while temps, if desired, are specified in the Temps Output tab, which contains the same set of parameters.

Note:

It is the Exec node that controls which type of file is output, temps or finals. No temps will be rendered unless you specify that option in the Exec node. The Exec node also enables you to lower the resolution of the temps output to save rendering time and/or disk space.

Filename and Path Conventions

By default, the Finals Path entry field contains the following directory path and filename:

$JOB/$OP/$F4.tiff

- "$JOB" is a global variable (see Setting Global Variables in chapter 6) that defaults to the directory from which you started Chalice.
- The "$OP" variable represents the current node name (for example, "output1").
- The "$F4" variable sequentially numbers each frame file with a four-digit number, starting with "0001" (the last frame of a 200-frame sequence would be "0200").
- The ".tiff" is the file format extension. The default file format for the Disk Output node is TIFF (see the Format parameter description below). If you select another format from the Format popup menu, Chalice will replace the .tiff extension in the File field with the appropriate extension for the chosen file format.

Therefore, if you write a 200-frame sequence to disk using a Disk Output node named "output1" and use the File parameter default of "$JOB/$OP/$F4.tiff," Chalice will create 200 files named "0001.tiff" through "0200.tiff" in a directory named "output1" in the directory from which you started Chalice.

Saving Files in Movie Formats:

If you output a sequence as any type of movie file, 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.

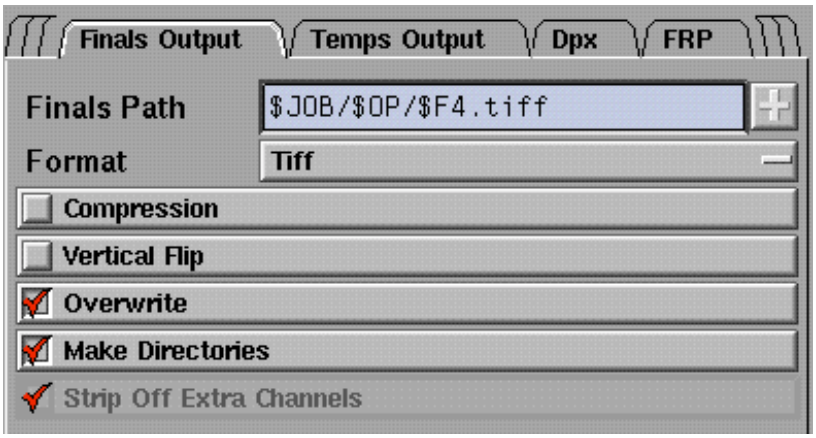
Parameter Tabs on the Disk Output Node Panel

The Disk Output parameter tabs include Finals Output and Temps Output tabs for specifying characteristics of the final files you want to output, as well as any temps you may want to render during a project. The rest of the Disk Output tabs are for setting format-specific options, and these options vary slightly according to the system on which Chalice is operating:

- **IRIX:** On SGIs running IRIX, the Disk Output node features Output, DPX, Fido, FRP, Jpeg/Jfif, Movie, Movie/MPEG, and Tiff.
- **NT:** On systems running Windows NT, the Disk Output node features Output, DPX, Fido, FRP, Jpeg/Jfif, and Tiff.

The movie-related tabs, which provide options for SGI, QuickTime, and MPEG formats, do not appear on NT because Chalice only exports AVI Video for Windows format on NT.

Finals Output Parameter Tab



Finals Path

The File parameter enables you to specify the name of the file sequence and the directory location to which the files should be written.

You can key the complete pathname into the File data entry field or you can click the File (+) button to open a Choose File dialog box. In the Choose File dialog you can navigate the directory structure of your hard disk or network to select a directory. For more information on using the navigational features of Chalice file dialogs, see Opening an Existing File in chapter 5, "Building Chalice Networks."

Format

The Format parameter enables you to specify the format in which the image files will be written to disk. The default file format is Tiff. In most cases, you will save the file in the same format as the source file you imported into the Disk Input node. However, you may save the file in any format supported by Chalice. For a complete list see "Image File Formats Supported by Chalice" at the beginning of this chapter.

Compression
Click the Compression checkbox to specify that Chalice use compression when writing the files to disk, if you have chosen a format that supports compression.

To compress files in Tiff or a movie format, click the Compression checkbox here and go to the Tiff or Movie parameter tab to select additional compression options. Some files, such as Jpegs, are always compressed. To control the quality of Jpeg compression, use the Jpeg/Jfif parameter tab.

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 Disk Input node.

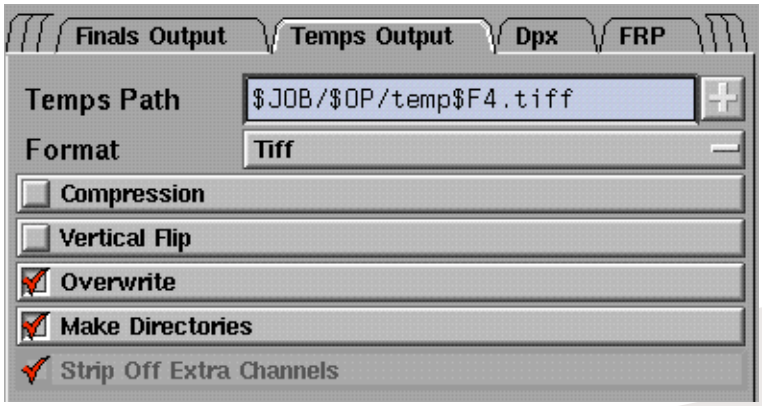
Overwrite
If the Overwrite box is checked (which is the default), Chalice will overwrite any files with the same name as the name you assign to the output files.

Make Directories
The Make Directories box, when checked (which is the default), lets Chalice create new directories, as needed, based on the directory path specified in the File parameter.

Strip Off Extra Channels
Some of the image file formats supported by Chalice only support images with three or fewer channels. The Strip Off Extra Channels box is checked by default whenever you choose to output imagery in one of these formats. If you attempt to output image data that contains more channels than are supported by the selected file format, you will receive an error message. The message will indicate which channels are supported by the format chosen.

Temps Output Parameter Tab
The Temps Output tab is used only when you need to specify a different pathname, format, or compression option for any temporary composites that may be rendered during a project. Temps are rendered by using the Temps Execute command in the Exec node instead of the Finals Execute command.

The Exec node not only controls whether finals or temps are output to disk, but also enables you to set the output file resolution (for temps only) at a fraction of full size. However, you may also want to output temps in a different file format than that specified for the finals, or to output temps using compression and finals uncompressed. It is for this purpose that the Temps Output tab is provided in the Disk Output node.



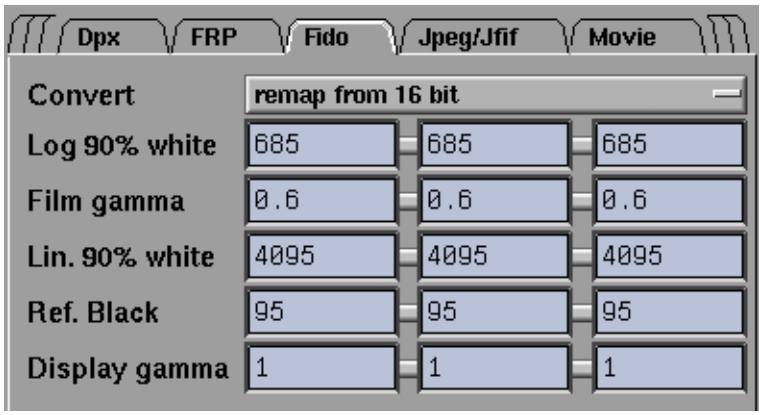
With the exception of the Temps Path default setting, the parameters are identical to those in the Finals Output tab. Refer to the Finals Output parameter descriptions (above) if you need more information.

Temps Path

The Temps Path parameter works exactly like the Finals Path parameter in the Finals Output tab. The Temps Path parameter, however, defaults to a different filename (temp$F4), although temps are written into the same directory as the finals unless you change the path. For more information, refer to the section on Filename and Path Conventions at the beginning of this node description.

DPX and Fido Parameter Tabs

The DPX Parameter Tab and the Fido Parameter Tab both feature the same set of parameters to facilitate the conversion of linear data (16-bit or floating point) to these Cineon 10-bit log formats for output. Therefore, both tabs are described here, illustrated by the Fido Tab.



Convert

The Convert parameter popup menu offers five conversion options: None, Extract from 16-bit, Extract from Float, Remap from 16-bit, and Remap from Float. The following chart illustrates the type of conversion to choose for your data:

Parameters Applicable to Each Convert Option	Convert to Log Options				
	None	Extract from 16-bit	Extract from Float	Remap from 16-bit	Remap from Float
Log 90% White		685	685	685	685
Film Gamma		0.6	0.6	0.6	0.6
Linear 90% White		4095			
Reference Black				95	95
Display Gamma				1.0	1.0

Select **None** if the image data is already in 10-bit log format. Otherwise, select the "remap from" or "extract from" option that corresponds to the method used to linearize the Cineon files originally (in the Disk Input node, e.g.).

The **Extract From** options perform the inverse of the "insert into" method used in the Disk Input node to convert log files to linear. Use Extract from 16-bit or Extract from Float if the data was originally converted to linear by the Insert into 16-bit or Insert into Float options.

The **Remap From** options perform the inverse of the "remap to" method used in the Disk Input node to convert log files to linear. Use Remap from 16-bit or Remap from Float if the data was originally converted to linear by the Remap to 16-bit or Remap to Float options.

Converting non-Cineon imagery: If the imagery to be output as Cineon was never converted from Cineon (if it was created in SoftImage or Photoshop, for example), select the appropriate "remap from" option for the data type (16-bit or float).

Conversion Parameters

The following chart illustrates which conversion parameter applies to each conversion method. The values shown represent the defaults, which are based on specifications for Cineon conversion published by Cinesite and Kodak.

Type of Data to Be Converted to DPX/Fido	Disk Output Conversion Option to Use
Cineon Linear, converted using "insert into" option in Disk Input	Extract From (16-bit or Float)
Remapped Linear, converted using "remap to" option in Disk Input	Remap From (16-bit or Float)
Original Linear (a 16-bit tiff, e.g.)	Remap From (16-bit or Float)
Cineon 10-bit Log File	None

Log 90% White

The Log 90% White parameter enables you to modify the default RGB log values of 685 for the 90% white card.

Film Gamma

The Film Gamma parameter enables you to modify the default RGB conversion values for film gamma.

Lin. 90% White

The Linear 90% White parameter enables you to modify the default RGB linear values of 4095 for peak white in the conversion from 16-bit linear color space.

Reference Black

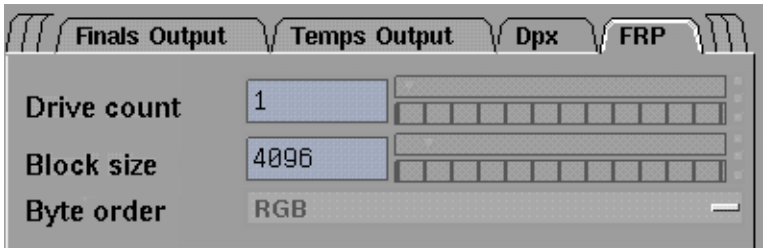
The Reference Black parameter enables you to modify the default RGB log values of 95 for reference black.

Display Gamma

The Display Gamma value is used in the Softclip conversion algorithm. The default value of 1.0 should not need modification in most cases.

FRP Parameter Tab

"FRP" stands for "Fast Realtime Playback." As its name implies, FRP enables you to save a sequence to disk and play it back in realtime, assuming you have the necessary hardware support. FRP files can be played in a flipbook by accessing the Play command in the Chalice File menu.



When you save a sequence in FRP format, the frames are saved (at their native resolution) as 8-bit data in a single file. FRP is platform specific; that is, an FRP file created on IRIX cannot be shared with an NT system running Chalice.

Tip:

You can also save a sequence to disk in FRP format from any Chalice flipbook. Flipbooks are described in chapter 6, "Monitors and Flipbooks."

For best performance, it is essential that the optimal segment size for your filesystem is used when writing the data to your disk, or disk array. To calculate this value, Chalice uses the information in the FRP tab parameters. Ensuring that accurate data is entered for these parameters will improve playback speed dramatically.

Note:

To determine the correct segment size value, Chalice multiplies the image width (X) by the image height (Y) by the number of channels (C) in the image. The result is then divided by the block size (B) of your file system and the result is rounded up to the next full integer. This number is multiplied by the block size:

round(((X * Y * C) / B)+.5) * B

Drive Count

This parameter value should reflect the number of drives in your filesystem. The default value is 1, unless you have changed the "Number of drives in filesystem" parameter in the Monitor Preferences panel (which is accessed from the Preferences menu).

Block Size

This parameter value should represent the blocksize of your filesystem, in bytes (e.g., 4096). Like the Drive Count parameter, the default Block Size value is based on the corresponding parameter in the Monitor Preferences panel.

Byte Order

This popup menu enables you to select the most appropriate setting for your workstation. Choosing the correct byte order will improve playback performance of the FRP; however, it is frame buffer dependent, so we can only offer the general guidelines that NT users will most likely want to select RGB, while SGI users of O2 and Reality Engine hardware will want RGBA, and users of Indigos and Octanes, ABGR.

Jpeg/Jfif Parameter Tab

Quality

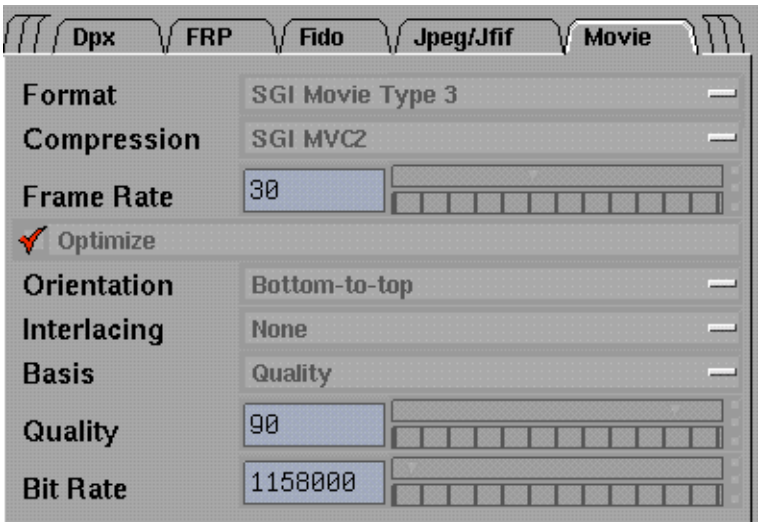
The Quality parameter lets you determine the relative image quality for a file you wish to subject to Jpeg compression. You set the Quality parameter by keying in a numerical value in the range of 1 to 100 or by using the mouse to set the tuner to the desired value.

The lower the numerical value assigned, the lower the image quality will be, but the higher the level of compression will be. The default is 90; the actual useful range is approximately 35–90 (with an image compressed to 35 being a very low quality image).

Movie Parameter Tab (IRIX Only)

If you have elected to export an image sequence as a movie file by selecting Movie from the Format parameter menu in the Output tab, you can use the Movie parameter tab to specify the type of movie format, compression, and other options.

The optimal values of some parameter settings may vary based on the type of movie file you wish to create. For more information about these movie file formats, consult chapter 3, Media Convert, of the Digital Media Tools Guide in your IRIS InSight online library.



Format

The Format parameter popup menu enables you to specify which movie format Chalice should use: SGI Movie Type 1, Type 2, or Type 3; Apple QuickTime; or MPEG-1 Systems. The default format is SGI Movie Type 3.

Compression

The Compression parameter popup menu enables you to specify the type of compression Chalice should use:

- Select None if you do not wish to use compression.
- To compress an SGI format movie, select one of the following: SGI JPEG, SGI RLE, SGI RLE24, SGI MVC1, or SGI MVC2 compression.
- To compress an Apple QuickTime format movie, select one of the following: QuickTime Photo (JPEG), QuickTime Animation (RLE), QuickTime Video (RPZA), or QuickTime Compact Video (Cinepak) compression.
- To compress an MPEG-1 Systems format movie, select MPEG-1 Systems compression. (The Movie/MPEG parameter tab allows you to change the default compression settings for an MPEG-1 movie.)

Note:

The Compression checkbox in the Output parameter tab must be checked to enable any of the movie compression options.

Frame Rate

You specify the frame rate for the movie by keying a value in the range of 8 to 60 frames per second into the data entry field, or by using the mouse to set the tuner to the desired value. The default value is 30 frames per second.

Optimize

Check this parameter if you wish to optimize the movie file for faster playback. The process involves creating a temporary copy of the file, however, which means that optimization not only takes longer but requires twice as much temporary disk space.

Orientation

The Orientation parameter popup menu allows you to specify the order in which the pixel data for each frame is written: Bottom-to-top or Top-to-bottom. The default is Bottom-to-top, which is optimal for software playback on your SGI system. However, if the movie is destined for playback on a VCR or other analog device, specify Top-to-bottom.

Interlacing

The Interlacing parameter enables you to specify whether the file should be interlaced. The default is **None**. However, you can specify **Odd** to interlace the frames starting with the Odd field lines, or **Even** to interlace the frames starting with the Even field lines.

Basis

If you have selected a lossy compression method such as Jpeg, the Basis parameter popup menu enables you to select the Quality or Bit Rate options for modifying compression settings. The default basis is Quality.

The lower the numerical value assigned in the Quality or Bit Rate parameter, the lower the image quality will be, but the higher the level of compression will be.

Quality

The Quality parameter enables you to specify the relative image quality for a file by keying a numerical value in the range of 1 to 100 (the default setting is 90) into the data entry field or by using the mouse to set the tuner to the desired value.

Bit Rate

The Bit Rate parameter enables you to specify the level of compression in terms of bits per second of transmission, rather than image quality.

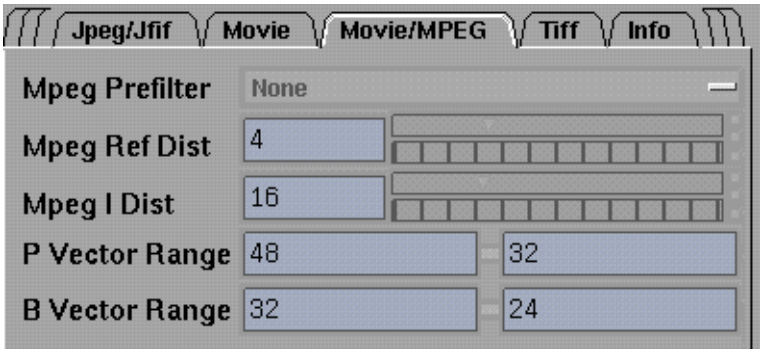
You set the Bit Rate parameter by keying a numerical value the range of 1 million to 10 million into the data entry field or by using the tuner dial to set a value. The default is 1,158,000 bits per second.

Movie/MPEG Parameter Tab (IRIX Only)

If you have elected to export an image sequence as an MPEG-1 Systems movie using MPEG compression, you can use the Movie/MPEG parameter tab to modify the default MPEG compression options.

Note:

It is recommended that you consult the section on "Customizing Compression of MPEG Files" in chapter 3 of the Digital Media Tools Guide in your IRIS InSight online library before changing the default parameter settings.



MPEG Prefilter

This parameter enables you to specify whether Chalice should perform low-pass filtering on the image data before compression to reduce noise and thus facilitate the compression process.

The default is None (no prefiltering); however, you can use the Prefilter popup menu to specify one of two levels of prefiltering: Medium or Maximum.

MPEG Distance Parameters

MPEG compresses frames in groups, rather than singly. These parameters allow you to modify the frame pattern by changing the default values for the distance between reference frames and the distance between "I" (Intra, or key) frames.

MPEG I Dist

You can modify the default value of 12 for the distance between I frames by keying an integer value in the range of 1 to 60 into the data entry field or by using the mouse to set the tuner to the desired value.

MPEG Ref Dist

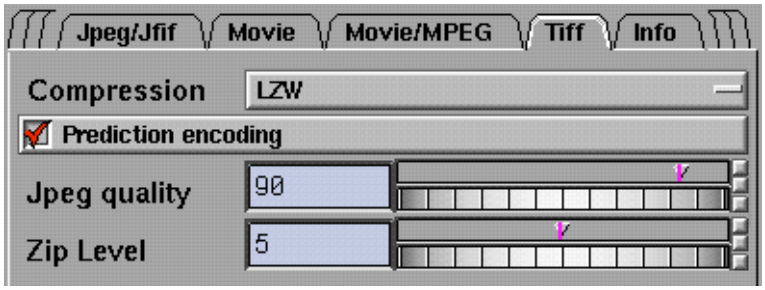
You can modify the default value of 4 frames for the reference distance by keying an integer value in the range of 1 to 12 into the data entry field or by using the mouse to set the tuner to the desired value.

P and B Vector Range Parameters

These parameters allow you to modify the pixel values representing the search vectors used in the compression. The P Vector Range represents the predicted motion vector; the B Vector Range, the bidirectional vector. You can modify the default settings by keying new values into the data entry fields. The field on the left represents the X (horizontal) axis; the field on the right, the Y (vertical) axis. Higher values will result in a higher quality image but will be correspondingly slower.

Tiff Parameter Tab

If you have elected to export image files in Tiff format, you can use the Tiff parameter tab to specify the type and level of compression to be used.



Compression

The Compression parameter menu enables you to select the type of compression you wish to apply to a Tiff file: JPEG, LZW (the default), Packbits, or Zip (the highest compression).

Note:

The Compression checkbox in the Output parameter tab must be checked to enable any of the Tiff compression options.

Prediction Encoding

This parameter allows you to specify whether Chalice should use prediction encoding to optimize the compression process (the default is checked, or on). This method calculates the difference between adjacent pixels and encodes the difference rather than the original data. Certain types of imagery, such as computer generated models, encode very well using this method. However, for noisy images it may be preferable to turn prediction encoding off to obtain the best result.

Jpeg Quality

The Jpeg quality parameter enables you to determine the relative image quality of files for which you have specified Jpeg compression in the Compression parameter menu. The lower the numerical value assigned, the lower the image quality will be, but the higher the level of compression will be.

Zip Level

The Zip Level parameter enables you to set the level of compression of files for which you have specified Zip compression in the Compression parameter menu. Because, unlike Jpeg, Zip is a lossless compression method, the only penalty for assigning higher numerical values, which represent higher levels of compression, is that the compression process will take correspondingly longer.

13 TOOL NODES

The nodes in the Tools menu enable you to control specific functions and processes and to prepare other node data for further processing. These nodes are general utility tools that you can use in the construction of a Chalice network.

The **Exec** node is essential to any grail network; it controls the processing of image data for final output. After the Exec, two of the most commonly used tool nodes are the Channel Swap and Sequence nodes.

Channel Swap, which takes one or two inputs, enables you to add, delete, and swap channel data, even between two different inputs. For example, you can use Channel Swap to add an alpha channel to an RGB image. **Sequence** is a multi-input node: for every input you connect to the Sequence node, another input connector is created. Sequence allows you to specify frames for use in a network. It can be used to reconcile disparate frames ranges in source imagery before compositing, to hold or skip frames in a sequence, etc.

The Track and Pixel Meter nodes each take a single input, from which they generate data that can then be accessed by other nodes in the network. **Track** creates tracking (*x,y* position) data which can be used to match move, stabilize, or corner pin. The **Pixel Meter** node meters and averages the RGBA values of selected pixel areas.

The **Text** node has been placed in the Tools menu because it is often used as a diagnostic tool. By using simple expressions, the Text node can display any current parameter value over an image. Of course, this node could just as easily have been placed in the Composite or the Effects menu, as it can be used to generate text effects over other imagery for output.

The other utility nodes include Switch, Time Stretch, UNIX, and Comment, all of which are described in this chapter. The Tools node menu includes, in alphabetical order:

Channel Swap Node ..... p. 13.2
Comment Node ..... p. 13.4
Exec Node ..... p. 13.5
Pixel Meter Node ..... p. 13.9
Sequence Node ..... p. 13.12
Switch Node ..... p. 13.17
Text Node ..... p. 13.18
Time Stretch Node ..... p. 13.24
Track Node ..... p. 13.25
UNIX Node ..... p. 13.31

CHANNEL SWAP NODE

The Channel Swap node enables you to quickly and easily modify channel data in your images in numerous ways. With Channel Swap, you can

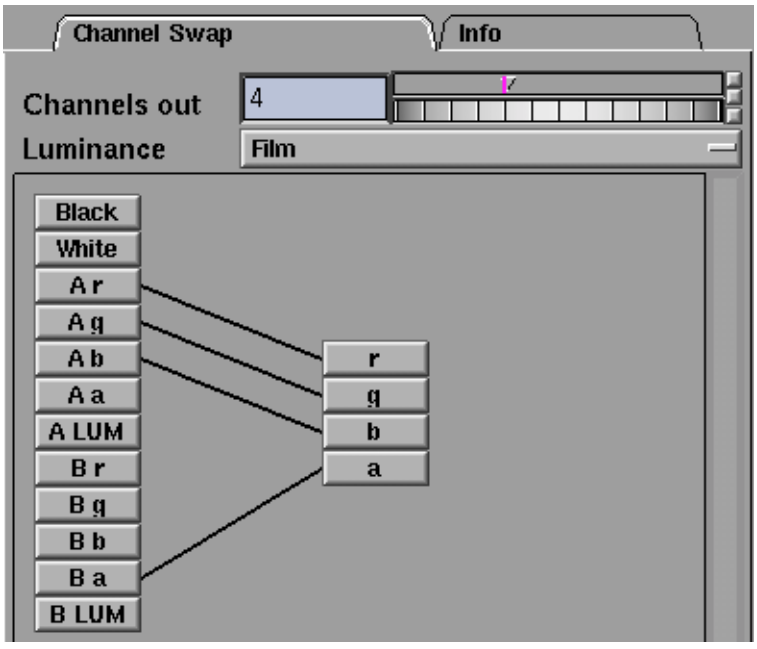
- add channels
- delete channels
- swap channels from two separate input images
- insert black, white, or luminance values into a channel

The Channel Swap node accepts one or two inputs. You determine the output image by specifying the number of channels it will contain, from one to 10. The default is four channels (RGBA). Then you can:

- assign each output channel the values of any input channel, or
- fill an output channel with black, white, or the luminance value of any input.

Channel Swap Parameter Tab

The output channels appear in the right column; the input channels appear in the left column. The input channels column always displays Black (transparent) and White (opaque). When you connect an input node to the Channel Swap node, the channels that compose the input image appear below Black and White, followed by an entry for luminance ("A LUM" for the top input image; "B LUM" for the bottom input).



To reassign the output channels, click and drag a connector line from one of the output channels (on the right) to an input channel (on the left). You can connect an input channel to multiple output channels. However, each output channel can accept only one connector.

Channels Out

To change the number of output channels, simply key a value in the range of 1 to 10 into the Channels Out data entry field, or use the tuner to set the parameter to the desired value.

Luminance

This parameter enables you to specify the type of luminance to be calculated. You can select Film, NTSC Video, or PAL video from the Luminance popup menu.

Film

Chalice computes the film luminance value as follows:

red channel * 0.333 +
green channel * 0.333 +
blue channel * 0.333

NTSC Video

Chalice computes the NTSC luminance value as follows:

red channel * 0.299 +
green channel * 0.587 +
blue channel * 0.114

PAL Video

Chalice computes the PAL luminance value as follows:

red channel * 0.2125 +
green channel * 0.7154 +
blue channel * 0.0721

COMMENT NODE

The Comment node provides you with a means for documenting a dataflow. Although all nodes include a comment field in their Info parameter tabs, the Comment node is a more conspicuous marker that makes it immediately apparent that a specific comment exists.

You can place as many Comment nodes in the workspace as you need to document the network. When you create a Comment node, Chalice assigns it a unique name (for example, "comment1"). It can be helpful to replace the default name with a more descriptive name. Type a unique name in the Name field of the Comment node panel.

Reading Comments:

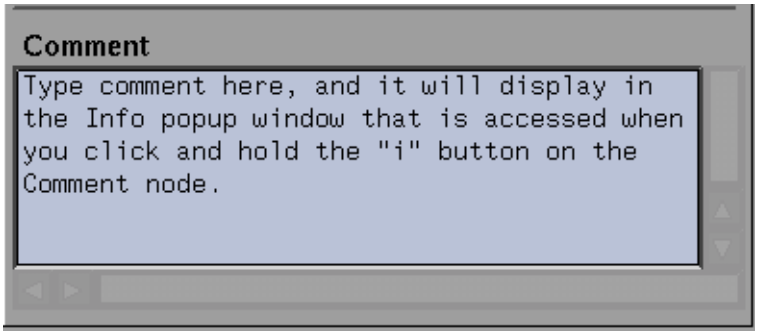
Comments can easily be referred to whenever you are working in a Chalice network: Click and hold the Info (i) button on the right side of the Comment node, and your comment will be displayed in the popup panel. You do not have to access the Comment node panel to read a comment, only to edit one.

The Comment node features an Info parameter tab only. Unlike other nodes, the Comment node passes data without change to and from any nodes to which it is connected. Therefore, the Comment node can accept one input or no input, and its output connector may or may not be connected to another node in the dataflow.

If the Icon Image box in the local node menu of the Comment node is checked, the Frame Icon of the Comment node will display the image from the input node (if a node is connected to the Comment node). To save room in the workspace, turn off (uncheck) the Icon View box in the Comment node menu.

Info Parameter Tab

The feature of the Info tab that is relevant to the Comment node, the comment field, is described here. (For a complete description of the Info Tab, refer to chapter 6, "Node Panels and Parameters.")



Comment Field

To key in a comment, click the left mouse button in the Comment field and begin typing. If you wish to enter a carriage return, hold down the Alt key on the keyboard and press the Enter key. (Otherwise, the text will stream past the visible edge of the field to the left.)

To access the text for editing, click and drag the scroll bars at the bottom and side of the Comment field to view portions of the text that extend beyond the field frame.

Tip:

You can open a text editor window in which to write and edit your comments. Click the cursor in the Comment field and press **Alt-e** to bring up the text editor window.

EXEC NODE

The Exec node is the main process control function for the Chalice system. The Exec node instructs the system to begin processing a Chalice network and specifies which frames to process and the order in which to process them. In other words, the Exec node "executes" a Chalice network.

The Exec node is the terminus node for the network. The Exec node has no output connector, since no other node may be placed downstream from it. The Exec node accepts multiple inputs.

Using the Exec Node

Each time you connect an input to an Exec node, Chalice creates an entry in the Exec parameter tab that enables you to specify the sequence of execution for the frames referenced by the network.

You specify the starting and ending frames, and an increment value. For example, if you key in 1 for the Start value, 50 for the End value, and 2 for the Increment value, when you execute or validate the network, the Exec node sends the frame values 1, 3, 5, ..., 49 upstream to the source nodes to begin processing. (For more information about how image data flows through a network, see Dataflow Mechanics of the Chalice System in chapter 4, "Key Concepts.")

As an alternative to keying in Start, End, and Increment values you can specify the name of an ASCII text file that contains a list of numbers representing the frames to be processed.

The Exec node also includes a slicing option that enables you to slice frames into smaller sections for greater efficiency when processing large image formats.

During processing, the Exec node displays an Exec notifier panel that indicates the current frame. You can cancel processing by clicking the Cancel button on the notifier panel. If you cancel processing, all frames that have already been written to disk are retained. You can resume processing with the first unprocessed frame, or start again from the beginning and overwrite the first batch of frames.

Processing Multiple Exec Entries

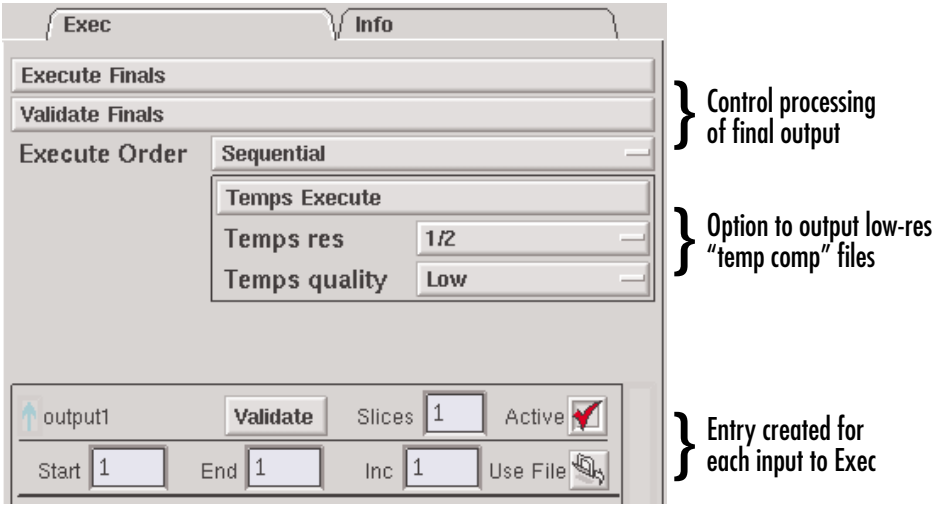
Each entry to the Exec node includes a checkbox labeled "Active" that enables you to turn the layer off (or back on) before processing. This gives you complete control over whether or not any particular sequence connected to the Exec is processed in the next execution cycle.

Executing Finals and Temps

During the creation of an effects shot, it is often necessary to create temporary composites ("temp comps"). Composited frames may be needed for evaluation purposes or for reference by another artist before the final shot is ready to be rendered.

Temp comps are usually lower resolution and quality than the final output imagery will be. The name, directory path, and format of these files are specified in the Temps tab of the Disk Output node; however it is the Exec node that controls which type of file is output: press the Finals Execute button to render finals and the Temps Execute button to render temps.

Exec Parameter Tab



Execute Finals

Press the Execute Finals button to instruct Chalice to begin processing. All inputs to the Exec node will be processed, unless the corresponding "Active" button for that input entry is disabled (unchecked) in the Exec panel entry list.

For example, if a Disk Output node is connected to the Exec, the Execute Finals command will process the image data and write the file sequence to disk as specified in the Finals Output tab of the Disk Output node.

Validate Finals

The Validate Finals button allows you to verify that all files referenced by the nodes in the network are present and accessible to the system and that there are no procedural errors in the network.

It is generally a good idea to validate a network before you attempt execution. When you press the Validate button, the Exec node will perform the validation operation; however, no data will be calculated or processed.

Execute Order

Use the Execution Order popup menu to choose between Sequential or Interleaved execution.

The **Sequential** option indicates that all the frames in the first Exec sequence entry will be processed before Chalice continues with the next Exec sequence entry.

The **Interleaved** option indicates that Chalice will process the first specified frame from each Exec sequence entry, then the second frame from each Exec sequence entry, and so on.

Temps Execute

The Temps Execute button enables you to render "temp comps" to disk, as specified in the Temps Output parameter of the Disk Output Node (see chapter 12, "I/O Nodes"). The size and quality of the output are determined by the Temps Res and Temps Quality parameters of the Exec node (see below). Higher quality and resolution may require considerably more processing time.

Temps Res

The Temps Res menu allows you select a lower resolution for the Temps Execute output. You can choose from 1/8, 1/4, 1/2, and Full. The default is half of full resolution.

Temps Quality

The Temps Quality menu allows you to set the quality of the Temps Execute output to Low, Medium, or High.

Exec Panel Entries

Each time a node is connected to the Exec node, a corresponding entry is created in the Exec panel. It includes the name of the input node, parameters to specify start and end frames and the increment, a "local" validate option, a slicing option, and a checkbox to turn the entry on or off.

Key in a value for the **Start** parameter to specify the first frame to process.

Key in a value for the **End** parameter to specify the last frame to process.

Key in a value for the **Increment** parameter to specify the frame increment of the sequence to be processed. For example, if you key in 1 (for the Start value), 50 (for the End value), and 2 (for the Increment value), this specifies the following values: 1, 3, 5, ..., 49.

Use File

There are two ways to specify frames that you wish to execute. You can key values into the Start, End, and Increment parameters, which is the default, or you can specify an ASCII text file that contains a list of numbers.

When you click the Use File button (the file folder icon), the parameter changes to display a path field and plus button, which accesses a standard Chalice Choose File dialog box. The file you choose should be an ASCII text file that contains a list of numbers delimited by carriage returns. These numbers represent a list of frames and are sent upstream to the nodes connected to the Exec sequence as an alternative way of specifying the frames you wish to execute.

Select the Use File icon again to toggle back to the Start, End, and Increment fields.

Validate

The Validate button enables you to validate only the associated Exec sequence entry. (If you wish to validate all entries connected to the Exec, use the Validate Finals button at the top of the node panel.)

Slices

The execution of a complex Chalice network requires a substantial amount of memory. Depending on your system configuration and the imagery and complexity of a particular Chalice network, you may need to divide the processing into smaller chunks to work within with the memory constraints of your system. The Slices parameter enables you to "slice" image processing into smaller chunks.

Unfortunately, you cannot tell how much slicing your network will require until you attempt execution. If possible, avoid slicing altogether. If slicing is required, use the minimum that enables Chalice to successfully process the network. For example, begin with four slices. If this fails, try eight slices, and so on.

For more information about slicing, see Memory Usage in Chalice in chapter 4.

Active

This checkbox is used to specify whether or not to execute the associated sequence during the next execution cycle. By default, the box is checked, indicating that the entry layer is active and will be processed. To disable processing for the layer, uncheck the box.

PIXEL METER NODE

The Pixel Meter node enables you to sample color values in selected areas of an image sequence. This data can then be displayed in the Pixel Meters window, or it can be accessed by other nodes.

The Pixel Meter node accepts one input. The Pixel Meter node must be connected to an Exec node to actuate the pixel metering process. Chalice will store the resulting data in the Pixel Meter node.

Tip:

If the Exec node contains more than one sequence entry, turn off all other entries before executing the metering sequence to avoid unnecessary processing. For more information, see the Exec Node description.

Meters Parameter Tab

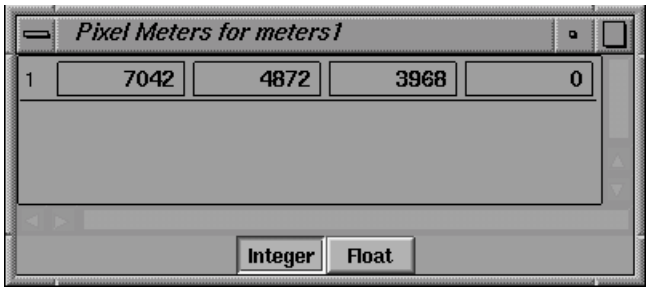
The Meters parameter tab enables you to specify the size and location of an area (or "pixel meter") in an image. The pixel metering function, which is actuated by the Exec node, will then calculate an average of the RGBA values present in the area of the image you specified.

Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node, which enables you to interactively position pixel meters in the image. For each pixel meter you create by clicking the Add Pixel Meter bar, a corresponding pixel meter appears in this monitor.

See Pixel Values

The plus button displays the Pixel Meters panel, which lists the average of the values contained in each of the RGBA channels for each pixel meter for the current frame. Values can be displayed as integer or floating point data, by pressing the Integer or Float buttons at the bottom of the Pixel Meters panel.



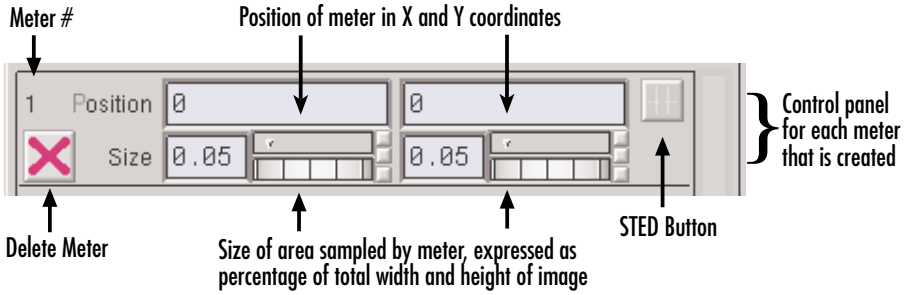
You can use the Frame Controller to change the current frame, and the values displayed in the Pixel Meters window will update accordingly.

Add Pixel Meter

Each time you click the Add Pixel Meter button, Chalice adds an additional pixel meter in the lower-left corner of the image in the Meter Monitor and makes an additional entry in the Pixel Meter List.

Pixel Meter Entries

A corresponding entry panel is created in the Meters tab for each pixel meter that you create. For each entry you can specify the position of the pixel meter and the size of the area to be sampled.



X Position and Y Position

You specify the location of each pixel meter by keying a numerical value into the X Position and Y Position data entry fields. Alternatively, you can use the work monitor to interactively position pixel meters. The data entry fields will update accordingly.

X Size and Y Size

You specify the size of each pixel meter (the dimensions of the area being sampled) by keying a numerical value into the X Size and Y Size data entry fields or by using the mouse to set the tuners to the desired values.

Delete Button

Click the red "X" button to delete an entry from the Pixel Meters list.

STED Button

Each Pixel Meters list entry includes a STED button, which you can click to display the Chalice Space-Time Editor. You can use the STED to view the pixel meter values computed thus far as a graph. The STED will also enable you to animate the pixel meter location and size over a range of frames. For more information, see chapter 9, "Space-Time Editor (STED)."

Using Pixel Meter Data in Conjunction with Other Nodes

You can use pixel meter data to make color adjustments in other nodes, such as Brightness, by entering into the parameter field of the other node an expression that uses the following syntax:

meters("meters node name", <pixel meter#>, <channel#>, <frame#>)

The Red channel is 0, Green is 1, Blue is 2, and the Alpha channel is 3.

For example, keying the expression meters("meters1", 1, 0, $F) into the Brightness parameter (Expressions must be checked to key in the expression) would reference the red channel data for the current frame from pixel meter number 1 of a Pixel Meter node named "meters1." (For more information, see appendix B, "Chalice Expression Language.")

Using Other Node Data in Conjunction with the Pixel Meter Node

You can also use data from other nodes in conjunction with the Pixel Meter node. For example, you can access tracking data that you have already generated in a Track node to drive the position of a pixel meter to obtain average RGBA values for a moving object.

Follow these steps:

1. **Connect the node containing the image for which you wish to obtain pixel meter data to a Meters node, and connect the Meters node to the Exec node.**
2. **In the Meters node panel, click the Add Pixel Meter bar.**
   A corresponding pixel meter entry will appear in the Pixel Meters list.
3. **Check the Expressions box and enter the following expressions into the X and Y Position fields of the pixel meter entry:**

   ```
   track("track1",XV,1,$F)
   ```

   ```
   track("track1",YV,1,$F)
   ```

   Note:

   Replace the name "track1" in these expressions with the name of the particular Track node that contains the tracking data you wish to reference. (Be sure to retain the quotation marks around the node name.)
4. **In the Exec node panel, press the Execute button.**
   Chalice will calculate the values and store them in the Meters node.
5. **In the Pixel Meter node panel, press the See Pixel Values button.**
   Chalice will display the Pixel Meters panel, which lists the RGBA values for each pixel meter at the current frame.
6. **Open the Pixel Meter work monitor to display the current position of each pixel meter in the image.**

SEQUENCE NODE

The Sequence node enables the Chalice system to address any sequence of frames for processing. You can use the Sequence node to perform the following tasks:

- remap frame numbers
- hold (repeat) frames
- skip frames
- specify a list of frames in any order
- combine frames from multiple input sequences

The Sequence node works by remapping frame numbers according to your specifications. In Chalice, the Exec node is the main processor and sends frame numbers to the nodes connected to the left of it, upstream in the network, when you execute (process) a shot. In most situations, these numbers represent the current output frame.

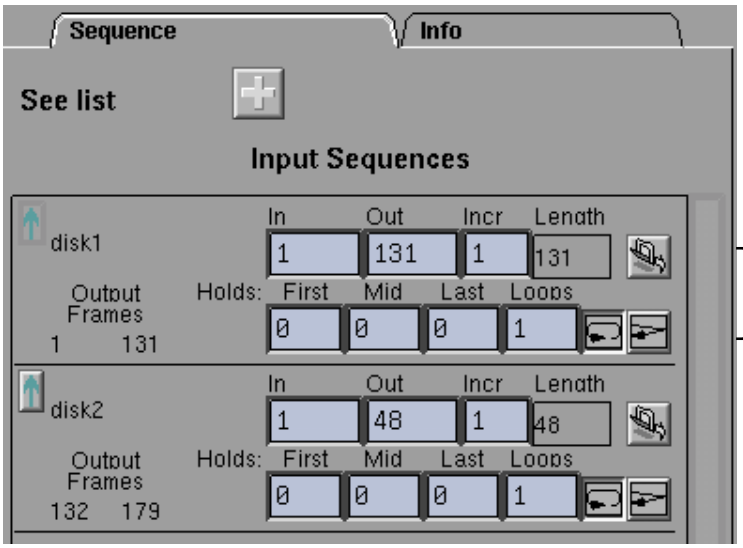
The Sequence node essentially translates the numbers that are delivered to it on the right to a new set of numbers that it will pass to the left, upstream. Once the numbers are determined, the actual image data that the Sequence node receives from upstream nodes is passed downstream (to the right) without modification by the sequencer.

Note:

You can resequence frames directly in the Disk Input node. The Sequence node is used when you want to re-sequence frames later in the network, or when you need to resequence frames from multiple source nodes.

Sequence Parameter Tab

Chalice creates an entry panel in the Sequence tab for each input node you connect to the Sequence node. This is where you specify which input frames the Sequence node will use and how the frames will be ordered.

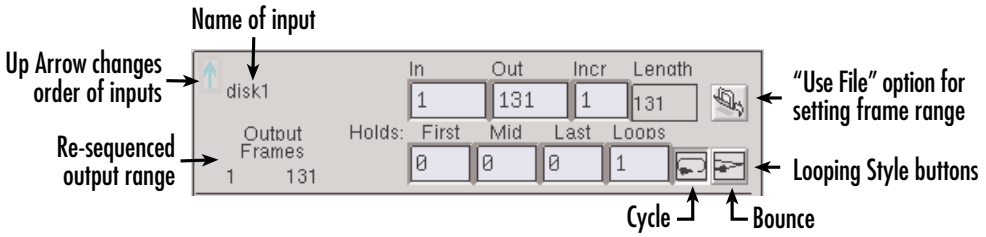


See List

The See List button (+) displays a window that shows the numbers delivered to the right of the Sequence node, the input frames selected, and the frames delivered to the left of the Sequence node.

Sequence List Entries

The entry panel for each input to the sequence node features several parameter controls, which are described in the following paragraphs. The next section of this node description, Sequencing Examples, also helps explain how to use this node.



Up Arrow Icon

You can easily change the order of the entries in the Sequence List area. Clicking the Up Arrow icon in a panel moves that entry one step closer to the top of the list.

In, Out, and Incr. Fields

Key in a value for the In parameter to specify the **first frame** of the input to use.

Key in a value for the Out parameter to specify the **last frame** of the input to use.

Key in a value for the Incr. parameter to specify the **frame increment**. For example, an increment value of 1 specifies every frame, and a value of 2 specifies every other frame.

Use File Option

There are two ways to specify how an input can be re-sequenced. You can key values representing frame numbers into the In, Out, and Increment parameters (the default), or you can create an ASCII text file that contains a list of numbers.

To access the text file list, click the Use File button. The parameter will change to display a path field and an associated Choose File button (plus icon), which brings up a dialog box that enables you to select a file. This file should be an ASCII text file that contains a list of numbers delimited by carriage returns. These numbers represent a list of frames and will override the In, Out, and Increment parameter values.

This option is particularly useful for specifying a highly random order of frames. (See also Example 3: Single Sequence, Irregular Ordering of Specific Frames.)

Click the Use File button (file folder icon) again to toggle back to default mode.

Hold Fields

Key in a value for the **First Hold** parameter to specify the number of times you wish to repeat (hold) the first frame of the entry sequence.

Key in a value for the **Last Hold** parameter to specify the number of times you wish to repeat (hold) the last frame of the entry sequence.

Key in a value for the **Mid Hold** parameter to specify the number of times you wish to repeat (hold) the middle frames of the entry sequence. The term "middle" in this case refers to all the frames in the sequence that fall between the first and last frames.

> **Tip:**
>
> One common application of the Hold parameters is to double frames. Simply key the same number of Hold frames into all three Hold parameter fields. To double frames, key in 1 for the value; to triple frames, key in 2; to quadruple frames, key in 3; and so on.

Loops

Key in a value for the **Loops** parameter to specify the number of iterations of the sequence. If you specify more than one iteration, use the Cycle and Bounce buttons to specify the type of looping (see also Example 2: Repeats and Looping):

- Selecting the **Cycle** button instructs the Sequence node to repeat a sequence in order. For example, a sequence of three frames will repeat as 1, 2, 3, 1, 2, 3, 1, 2, 3, etc.
- Selecting the **Bounce** button instructs the Sequence node to repeatedly play forward, then back, through a sequence. For example, a sequence of three frames will repeat as 1, 2, 3, 2, 1, 2, 3, 2,1, etc.

Sequencing Examples

The following examples should help you understand how to use the Sequence node to accomplish specific types of resequencing, as well as exactly what results to expect.

Example 1: Using Two Sequences and Remapping

In this example we will set up two streams of imagery to be sequenced one after the other by connecting two Disk Input nodes to a Sequence node. The Sequence node will then display two sequence entry panels.

We wish to produce a 14-frame sequence, and have set the parameters for the Exec node accordingly. We wish to use frames 700 through 704 from one Disk Input node for the first five frames of the output, and to use frames 21 through 29 from the other Disk Input node for the following 9 frames.

We will assign the following parameters to the first sequence entry:

In = 700	Out = 704	Increment = 1	
First Hold = 0	Mid Hold = 0	Last Hold = 0	Loops = 1

We will assign the following parameters to the second sequence entry:

In = 21	Out = 29	Increment = 1	
First Hold = 0	Mid Hold = 0	Last Hold = 0	Loops = 1

In this case, as the Exec node sends the numbers 1 through 14 upstream, the Sequence node acts like a switch and a remapper of numbers.

For the first five frames the Sequence node will request frames 700–704 from the Disk Input node connected to the first sequence entry and pass the image data for those frames downstream, renumbered 1 through 5.

As the Exec node requests numbers 6 through 14, the Sequence node will switch to the second sequence entry and request frames 21–29 from the second input node. Then it will pass the image data for those frames downstream, renumbered 6 through 14.

The following chart maps the behavior of the Sequence node. (This information is also generated by the Sequence node when you click the See List button.)

Input Accessed	Frame Delivered on Left	Number Delivered on Right
Disk Input node 1	700	1
Disk Input node 1	701	2
Disk Input node 1	702	3
Disk Input node 1	703	4

Disk Input node 1	704	5
Disk Input node 2	21	6
Disk Input node 2	22	7
Disk Input node 2	23	8
Disk Input node 2	24	9
Disk Input node 2	25	10
Disk Input node 2	26	11
Disk Input node 2	27	12
Disk Input node 2	28	13
Disk Input node 2	29	14

Example 2: Repeats and Looping

This example demonstrates two types of looping (repeat) control. In this case we will repeat each sequence entry once by changing the Iteration value to 2. For the first entry we will select the Cycle button to repeat the sequence in a simple loop. For the second entry we will select the Bounce button to repeat the sequence in a ping-pong fashion.

We will assign the following parameters to the first sequence entry:

In = 700 Out = 704 Increment = 1
First Hold = 0 Mid Hold = 0 Last Hold = 0 Loops = 2
Loop control: Cycle

We will assign the following parameters to the second sequence entry:

In = 21 Out = 29 Increment = 1
First Hold = 0 Mid Hold = 0 Last Hold = 0 Loops = 2
Loop control: Bounce

Given these parameter settings, the Sequence node will generate the following result:

Input Accessed	Frame Delivered on Left	Number Delivered on Right
Disk Input node 1	700	1
Disk Input node 1	701	2
Disk Input node 1	702	3
Disk Input node 1	703	4
Disk Input node 1	704	5
Disk Input node 1	700	6
Disk Input node 1	701	7
Disk Input node 1	702	8
Disk Input node 1	703	9
Disk Input node 1	704	10
Disk Input node 2	21	11
Disk Input node 2	22	12
Disk Input node 2	23	13
Disk Input node 2	24	14
Disk Input node 2	25	15
Disk Input node 2	26	16
Disk Input node 2	27	17
Disk Input node 2	28	18
Disk Input node 2	29	19

Disk Input node 2	28	20
Disk Input node 2	27	21
Disk Input node 2	26	22
Disk Input node 2	25	23
Disk Input node 2	24	24
Disk Input node 2	23	25
Disk Input node 2	22	26
Disk Input node 2	21	27

Example 3: Single Sequence, Irregular Ordering of Specific Frames

In this example, assume that the Sequence node has one input consisting of a 100-frame sequence and that we wish to select 10 very specific frames for output. We wish to select frame 5 of the input sequence as the first frame, frame 8 as the second frame, and so on as listed below:

5
8
4
9
10
1
6
2
3
7

To accomplish this, we will create an ASCII file of the 10-frame sequence in the exact order we require and access it from the Sequence node.

In the Sequence parameter tab, click on the Range button to indicate that you wish to use a file instead of the In and Out parameters. A Choose File button will appear. Click this button to display the Choose File dialog box and select the ASCII file. The Sequence node will extract the numbers from the file and generate the following result:

Frame Delivered on Left	Number Delivered on Right
5	1
8	2
4	3
9	4
10	5
1	6
6	7
2	8
3	9
7	10

If we were to increase the default value of 1 in the Loops parameter, the Sequence node would repeat the numbers from the ASCII file in either a Cycle or Bounce fashion, depending on which button we selected. As with Loops, whatever values we set in the Hold parameters would also be applied to the frames of the file list.

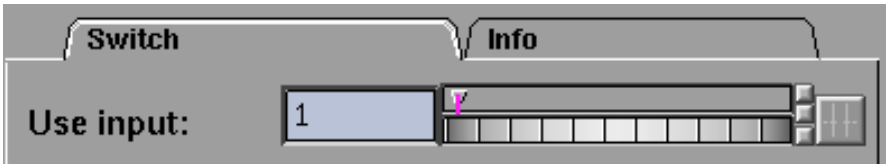
SWITCH NODE

The Switch node enables you to switch among multiple inputs in the order you specify, by using expressions, such that each frame of the output may come from any of the inputs. The Switch node accepts from one to 10 inputs.

Switch Parameter Tab

Use Input

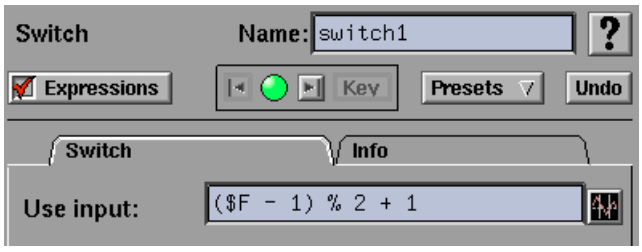
You can specify the number of the input to use for the current frame (as specified in the Frame Controller panel) by keying a number in the range of 1–10 into the data entry field or by using the mouse to set the tuner to the desired input number.



Most commonly, however, you will use an expression to specify the order of inputs.

Using Expressions in the Switch Parameter

First, you will need to check the Expressions box in the Switch node panel. Then you key in an appropriate expression. In the expression, 1 represents the first (top) input; 2 is the second input; 3 is the third input, etc.



Assume that you wish to switch back and forth between two inputs frame by frame:

frame 1 = input 1
frame 2 = input 2
frame 3 = input 1
frame 4 = input 2
…and so on.

To do so, connect the two inputs to the Switch node and check the Expressions box in the node panel. Then key the following expression into the Expressions field:

```
($F - 1) % 2 + 1
```

For more information about using expressions, see appendix B, “Chalice Expression Language.”

TEXT NODE

The Text node enables you to create and manipulate text for final output or for testing purposes. You can specify font characteristics, color, and opacity, as well as rotate, scale, and move the text layer.

Using the Text Node to Evaluate Expressions

You can enter expressions in the text field, such that the data referenced at any particular frame will be displayed in the monitor for that frame.

The simplest example is $F, which displays the current frame number. For example, when you type "$F" in the text entry field, the number "1" will be displayed in the monitor when viewing the first frame, the number "2" when viewing the second frame, and so on.

The ability to evaluate expressions may be most useful for displaying parameter data that you have animated in other nodes.

For example, you could access data from a Brightness node to display the current brightness value at each frame by using the Chalice ch function, with the expression surrounded by single back quotes (the " ` " symbol). You enclose the expression in back quotes to indicate that it should be evaluated, rather than being treated as text:

Current Brightness: `ch("brightness1/brightness")`

The above entry in the Text field would be displayed in the monitor as follows (where "0.5" is an example to represent the actual numerical value at the current frame):

Current Brightness: 0.5

To display the ` symbol as text, precede it with a backslash: \`

Note:

If you need more information about how to use expressions to access channel data, refer to the description of Data Channel Access Functions in appendix B.

The Text node requires one input, which may be any number of channels.

The output of the Text node will be the same number of channels as the input image, with the text composited into the RGB or RGBA channels.

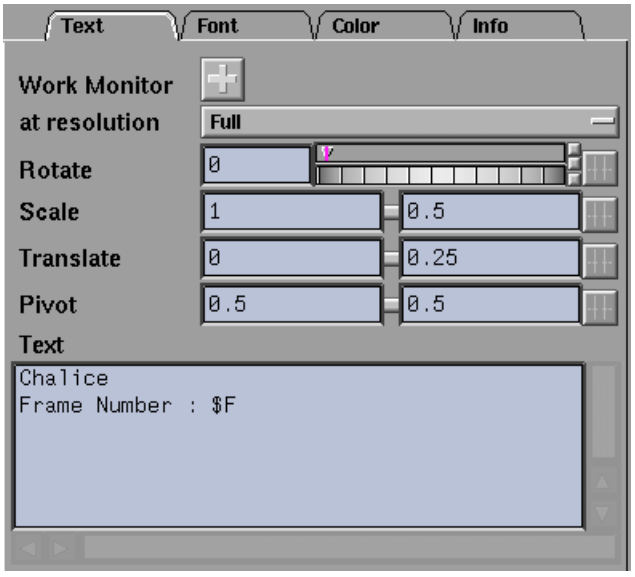
THE TEXT NODE PARAMETER TABS

The Text node features three tabs for creating and manipulating text: Text, Font, and Color.

The **Text** tab is where you enter and position text over the background. The **Font** tab provides parameters for selecting a font and adjusting spacing attributes. The **Color** tab controls the color of the text.

Text Parameter Tab

The Text parameter tab enables you to enter and edit the text to be used, to display the text layer in a work monitor, and to modify the text layer as a whole (by scaling, rotating, etc.). Transformations can be made using the interactive tool in the work monitor, or by keying specific values into the parameter fields.

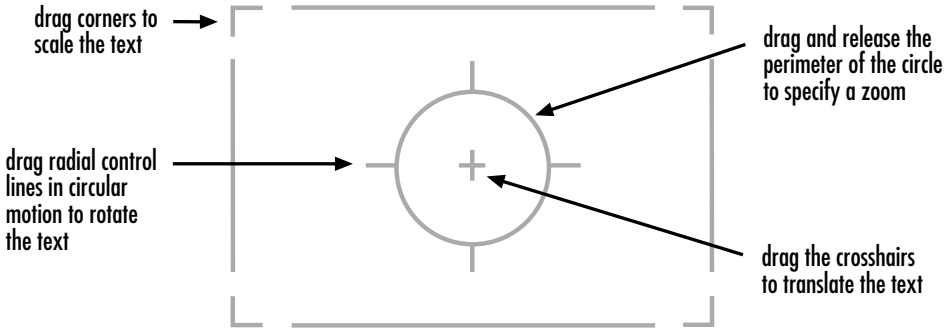


Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node, which enables you to interactively rotate, scale, and translate the text over the input image.

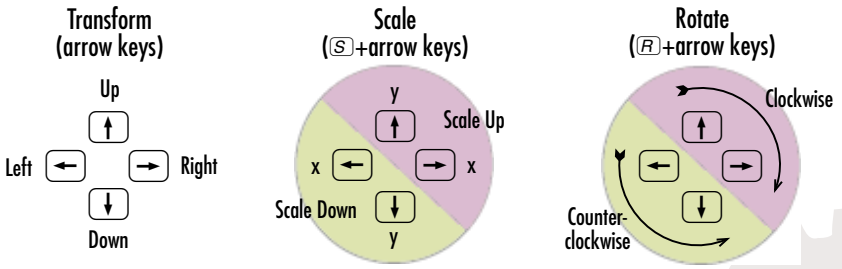
Work Monitor Tools

The work monitor for the node includes a control device that features a boundary box and a circle-with-crosshairs tool:



Work Monitor Hot Keys

Hot keys can be used in conjunction with the work monitor tool to "nudge" the device in small increments, as illustrated below:



You can also use the Rotate, Scale, Translate, and Pivot parameters to enter precise numerical values for these operations.

Rotate

The Rotate parameter enables you to rotate the text layer relative to the input image. Alternatively, you can use the interactive controls in the Text work monitor to rotate the text layer.

A positive value rotates the text in a counterclockwise direction; a negative value rotates the text in a clockwise direction. The rotation is centered around a pivot point that you specify by using the Pivot parameter (see below). You set the Rotate parameter by keying in a numerical value specifying the degree of rotation, in the range of 0–360, or by using the mouse to set the tuner to the desired value.

Scale

The Scale parameter enables you to scale, or proportionally adjust, the text layer relative to the input image. Alternatively, you can use the interactive controls in the Text work monitor to scale the text layer.

You set the Scale parameter by keying in a pair of multiplier values in the range of 0–1 for the X and Y coordinates. For example, entering 0.5 in both data entry boxes will create a text area that is half as high and half as wide as the input image. The 0,0 point is the lower-left corner of the source image. Keying the values 1,1 in the data entry boxes will display the text area at full size (at the same size as the input image).

Translate

The Translate parameter enables you to shift the position of the text layer along the X and Y axes of the input image. Alternatively, you can use the interactive controls in the Text work monitor to translate the text layer.

You set the Translate parameter by keying in a pair of numerical values. The first value shifts the text area along the X axis (horizontally). The second value shifts the text area along the Y axis (vertically). The distance is measured as a percentage of the X or Y axis within a range of 0 to 1. For example, values of 0.5, 0.0 will shift the text to the right a distance equal to one-half the width of the input image.

Pivot

The Pivot parameter enables you to specify a pivot point to be used in conjunction with the Rotate parameter (see above). The pivot point is the center of the rotation.

You set the Pivot parameter by keying in a pair of values in the range of 0–1. These values represent an XY location specified as a percentage of the input image width and height. The lower-left corner of the image is the 0,0 point; the upper-right corner of the image is located at 1,1; the center of the image is 0.5, 0.5.

Text

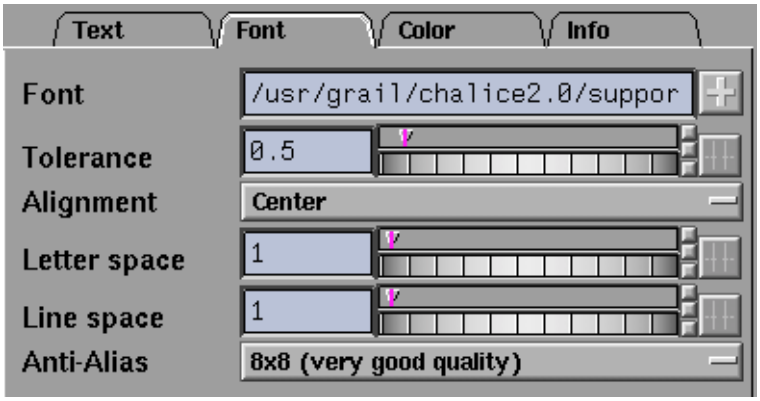
The Text parameter is a data entry field in which you can type any text that you wish. (To format the text, use the parameters in the Font and Color tabs.)

Just as you can with other Chalice text entry fields, you can cut (Alt x), copy (Alt c), and paste (Alt v) text; select all the text in the field (Alt a); and add line breaks (Alt Enter). To bring up a separate text editor window, use the “Alt e” command.

To access and display expression data, refer to the section on Using the Text Node to Evaluate Expressions at the beginning of this node description.

Font Parameter Tab

The Font parameter tab enables you to specify font attributes for the text.



Font

The Font parameter enables you to specify the Adobe Type 1 PostScript font to use. Click the plus icon to bring up a standard Chalice Choose File dialog. Navigate to the directory containing the font you wish to use and double-click on the font name.

Chalice provides several sample typefaces: Baskerville (in roman, bold, and italic fonts), Brush Script, and Liquid Crystal. They are located in the "psfonts" directory in the Chalice support directory. You can place additional fonts in this directory, or navigate to any directory that contains the font you wish to use.

Tip:

Use a monospace font such as Courier to display parameter values in diagnostic applications of the Text node. The equal width of all characters in monospaced fonts keeps the changing parameter values from seeming to jump from frame to frame.

Tolerance

The Tolerance parameter enables you to specify the level of tolerance applied as each letter is drawn on screen.

As you raise the tolerance level, fewer lines are used to approximate each letter shape. This means that for the most accurate representation you should use a low tolerance value, such as the default value of 0.5.

For faster rendering, especially when you are using the Text node for testing purposes rather than final output, specify a higher tolerance value. You may also want to experiment with high tolerance values for creating interesting and unusual letter shapes.

Alignment

The Alignment parameter enables you to specify that the lines of text be aligned flush left, flush right, or centered in the text layer by selecting an option from the popup menu. The default is centered text.

Letter Space

Use the Letter Space parameter to scale each letter to increase or decrease the space between letters in each line of text, within the current boundaries of the text layer. (This operation is somewhat analogous to kerning in the printing process.)

Line Space

Use the Line Space parameter to scale each letter to increase or decrease the space between lines of text, within the current boundaries of the text layer. (This operation is somewhat analogous to leading in the printing process.)

Anti-Alias

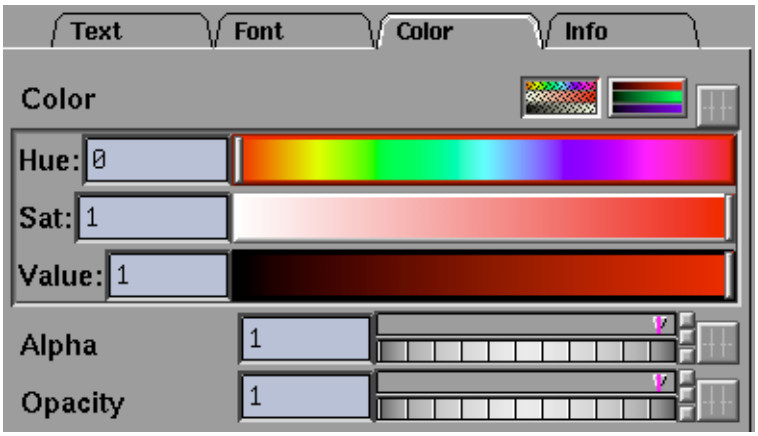
The Anti-Alias parameter enables you to specify the level of anti-aliasing to be applied to the text by selecting one of these options from the popup menu: None, 2x2, 4x4, 8x8 (the default), 16x16 or 32x32.

These values define the size of the grid into which a pixel is divided during the anti-aliasing process. The greater the number of subpixels, the higher the quality will be and the longer the process will take:

- If you select "None," the resulting text will suffer from severe aliasing, which is the jagged, staircase effect visible in diagonal lines and curves.
- For previews, "2x2" should be adequate.
- Both "4x4" and "8x8" will likely produce very effective anti-aliasing, but if necessary you can select even higher quality.
- Settings higher than "8x8" will slow down rendering considerably.

Color Parameter Tab

The Color parameter tab enables you to specify the color and opacity of the text.



Color

The Color parameter enables you to specify the color values of the text using RGB or HSV color space. The default mode is HSV. Click the RGB button to switch to RGB mode. The Color tab display will update to present RGB sliders in place of the Hue, Sat, and Value sliders.

HSV Mode

To specify the Hue, key a value in the range of 0–360 into the data entry field, or use the slider bar to select the desired hue. The Sat(uration) and Value parameters are specified in the same way, within a range of 0–1.

RGB Mode

You specify RGB values by keying in a value in the range of 0–1 or by using the slider bars.

Alpha

The Alpha parameter enables you to control the alpha channel value that is output from the Text node for compositing in another node. This parameter setting is only relevant when the input image has an alpha channel, and it affects the value of pixels with text-layer coverage only.

(To adjust the transparency level of the text as it appears in the Text node display, use the Opacity parameter instead.)

You can key a value in the range of 0 (transparent) to 1 (opaque) into the data entry field, or you can use the tuner to set the desired value.

Opacity

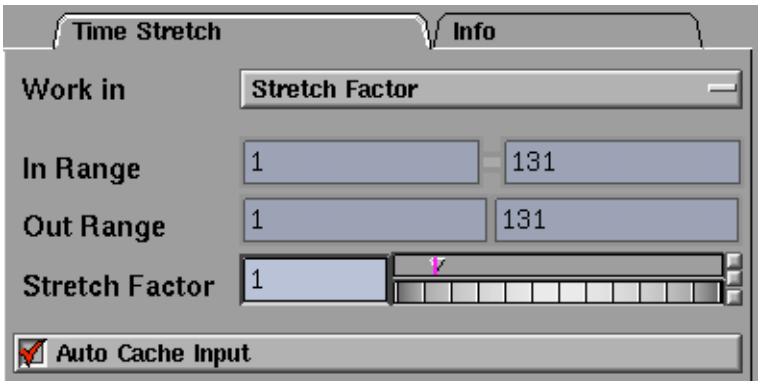
The Opacity parameter enables you to specify the opacity of the RGB channels in the text layer without adjusting the alpha channel, or if the input has no alpha channel. Key a value in the range of 0–1 into the data entry field, or use the tuner to set the desired value. A value of 0 represents total transparency; 1 is totally opaque.

This parameter adjusts RGB channel values of the relevant pixels and not the alpha channel value, if there is one.

TIME STRETCH NODE

The Time Stretch node changes the number of frames in the sequence that is input to the node to lengthen or shorten the sequence, as you specify. Time Stretch duplicates or deletes frames, using linear interpolation to dissolve between frames where necessary to soften the transition.

Time Stretch Parameter Tab



Work In

This parameter enables you to select the type of measurement system to work in: Frames or Stretch Factor.

In Range

This reference display shows the frame range of the sequence input to the Time Stretch node.

Out Range

This parameter is activated when you choose to work in Frames. Simply key a new ending frame number for the sequence into the active field.

Stretch Factor

This parameter is activated when you choose Stretch Factor from the Work In menu, enabling you to specify the factor by which the input sequence will be stretched. Depending on the value specified, Time Stretch will output a longer sequence by creating additional frames (for values greater than 1) or output a shorter sequence by deleting frames (for values less than 1).

For example, a factor of 2 will double the length of the sequence, while a factor of 0.5 will cut its length in half.

For reference, the Out Range parameter will update to show you the equivalent frame range for whatever factor you specify.

Auto Cache Input

This option, which is on by default, affects the cache setting of whatever node is serving as the input node to the Time Stretch node. The Auto Cache Input option resets the frame cache of the input node based on the current Factor value.

Auto Caching will greatly improve the speed of rendering; however it requires substantial space (in RAM or on disk) to cache the needed input frames.

TRACK NODE

The Track node enables you to set a point or points in an image, which the Track node then accurately follows throughout the sequence, creating an *x,y* value for every frame. The track position data can then be accessed from other nodes to perform various functions.

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). The Match Move and Stabilize nodes are described in chapter 17, "Movement Nodes."

The Corner Pin node (see chapter 17, "Movement Nodes") can access tracking data to control the position of corner pin points. Track data can also be used for other purposes. For example, the Pixel Meter node (described earlier in this chapter) can access track data to drive the position of a pixel meter.

Generating Track Data

First you need to create a track point for each area to be tracked in the image and position each point manually at the first frame of the range to be tracked (refer to step 1, below). Next you make any desired adjustments to the tracking parameters, such as the maximum correlation error to allow (step 2). Finally, you specify the frames to track and actuate the track function (step 3). The Track node will correlate the image data from frame to frame to generate the *x,y* values.

Once the track data has been generated, you can review and adjust it (see Re-tracking Points and Frames). You can also track frames manually (see Manual Tracking).

Step 1: Create Track Points

Start by opening the Track work monitor at the first frame to be tracked. (The work monitor button is located on the Tracker tab in the node panel.) The work monitor displays track points over the image as you create them. To actually create a track point, go to the Points tab and click the "Add Point" button. An entry will appear on the tab for every point you create.

Position the point interactively in the work monitor, or by using the corresponding entry in the Points tab to enter numerical values. The work monitor will update to reflect any changes in a track point entry.

Step 2: Adjust Tracking Parameters

If needed, adjust the parameters on the Tracker tab to get the best track for the current imagery. In particular, you will want to review the "Track On" parameter setting in the Tracker tab and the "Size" and "Speed" parameter settings in the Points tab:

The **Track On** parameter enables you to specify how Chalice will distinguish the areas defined by the track points from surrounding portions of the image in subsequent frames, based on luminance or color channels. The more detail, contrast, or color variance present in the area you select, the better Chalice will be able to track the object.

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 effectively using the Red, Green, or Blue channels.

Tip:

Another method you can use to improve a track is to run the image through a filter node 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.

For example, if your imagery is noisier than the Track node's grain filter can handle, you might use the Gaussian filter in a Blur node to filter out high-frequencies and remove noise, grain, and aliasing. Or you could use a Contrast node to increase the contrast and get a more accurate track. (Be aware that raising the contrast too much would also increase any noise or aliasing problems.)

In the Points tab, the **Size** parameter enables you to adjust the size of the area to track and the **Speed** parameter controls the size of the surrounding search region.

In general, the smaller you set the Size and Speed parameters for a track point, the faster the Track node will process the data. On the other hand, larger Size and Speed parameters take longer to process but may produce significantly better track data for some imagery.

Step 3: Set Frame Range and Actuate Track

Set the range of frames to track in the Tracking Frame Range controller. The Track node requires that all frames in the specified range be sequential; that is, there may be no gaps between frames.

Tip:

If you need to do more complex resequencing to specify the appropriate frames to track, insert a Sequence node between the source node and the Track node. For example, if you wish to track every other frame, set the increment in the Sequence node to 2.

Then actuate the track function by clicking the actuator button that corresponds to the order in which the frames should be tracked, such as forward to the end of the specified range. Refer to Tracking Frame Range Parameters later in this node description for more information.

Re-tracking Points and Frames

When the node has finished tracking, you can review the track path and point markers in the work monitor and make any necessary adjustments. Track points can be repositioned manually at any frame, and you can re-track selected points and frames.

For example, you might want to adjust the Maximum Error and Track On parameters before re-tracking a subset of frames that have less image detail than the successfully tracked frames.

The work monitor can display the path of each track point, along with a marker for the point position at each frame. Refer to the Draw Path and Marker in Monitor description to find out how each marker is color-coded.

You can reposition any track point for the currently displayed frame. Clicking on a point marker for a different frame will move you to that frame automatically, without altering the position of the point. Then you will be able to change the position of the tracking point for the new current frame.

Each track point can be enabled or disabled before executing the track function by using the Enable/Disable button associated with the track point entry in the Points tab. This means that you can re-track a point or points without affecting existing track data for other points.

Similarly, you can re-track a specific frame range for all enabled points without affecting existing track data for frames outside the new range setting.

Manual Tracking

The Track node provides the option of setting points manually at any frame in the sequence. Hand-set points appear blue in the work monitor display.

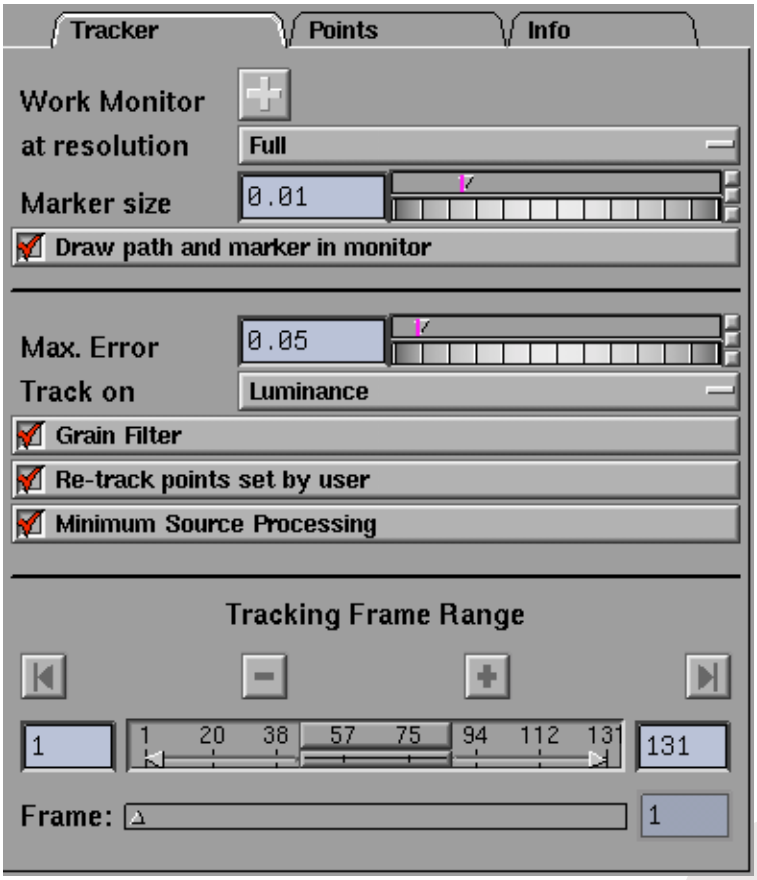
If there are frames in between two hand-set positions for a point, Chalice will linearly interpolate the positions for those frames (untracked points are red). This gives you the option of adjusting some or all of these positions manually. However, these interpolated point positions can also be auto-tracked using the normal track function (they will turn green after auto-tracking).

You can specify whether or not the node should re-track any hand-set positions by checking or unchecking the "Reset Points Set by User" parameter.

Tracker Parameter Tab

The Tracker tab provides parameters for:

- controlling work monitor display options,
- specifying how the image will be processed by the Track node during the tracking function,
- setting the frame range to use for the track, and
- actuating the track function.



Work Monitor Display Parameters

Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node, which enables you to position the points that specify the areas to be tracked.

Marker Size

This parameter enables you to increase or decrease the size of the markers that designate tracking points in the work monitor.

Draw Path and Marker in Monitor

This box, when checkmarked, instructs Chalice to display tracking data for all frames in a sequence. This option effectively draws the path of the tracking points in the Track work monitor. The color of the marker provides the following information about the tracking point:

- **Green** indicates that the point was tracked by the node.
- **Blue** indicates that the point was set manually (as at the first frame).
- **Red** indicates that the point is an interpolated position between two manually set points. If the frames are subsequently tracked by the node, any interpolated points will be tracked, after which they will turn from red to green.
- **Yellow** indicates that the node lost track for that point at that frame.

Tracking Parameters

Maximum Error

This parameter enables you to set the maximum allowable error value for the correlation of data between frames. If the error value rises above this value, the Track node will cease processing and generate an error message. The range is 0–1, with 0 being a perfect match. Any error value of 0.15 or less will be a good match. Typically this value is set at around 0.05.

The error value for each point at each frame tracked is displayed in the Track Points entry panel on the Points tab.

Track On

This menu allows you to select the type of image data Chalice will use to distinguish a tracking point from surrounding pixels. You can track on luminance (Film, NTSC, or PAL) or a color channel (Red, Green, or Blue). For more information on how Chalice computes luminance types, see the Channel Swap Node description of Luminance.

Grain Filter

The Grain Filter parameter, when checked, instructs the Track node to use 3 x 3 averaging to remove noise from the image during the tracking operation. This does not remove noise from the source image itself, of course—it filters noise as part of the tracking process only.

Re-track Points Set by User

This checkbox enables you to specify whether the node should re-track any points you have set or adjusted manually. Unless you check the box before re-tracking, the node will retain the position data created by manually setting a point at a frame.

Minimum Source Processing

A checkmark in this box instructs Chalice to display only a small area around a point while processing the tracking data. The rest of the image will be set to black. This can speed the process considerably, as it frees Chalice from reprocessing the entire image at each frame.

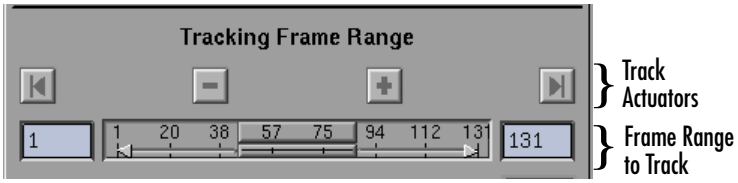
Tracking Frame Range Parameters

The Tracking Frame Range parameters enable you to specify the range of frames to track and the tracking direction from the current frame (forward to the end of the sequence, or backward to the beginning). You can auto-track

- the entire sequence,
- a subset of frames, or
- one frame at a time.

Track Actuator Buttons

Click any of these four buttons to start the tracking process. The node will then generate the track data based on which button you pressed and how you have set the other tracking parameters.



The plus and minus buttons track forward (+) or back (-) one frame. The arrow buttons track from the current frame either forward to the end of the range or backward to the start of the range.

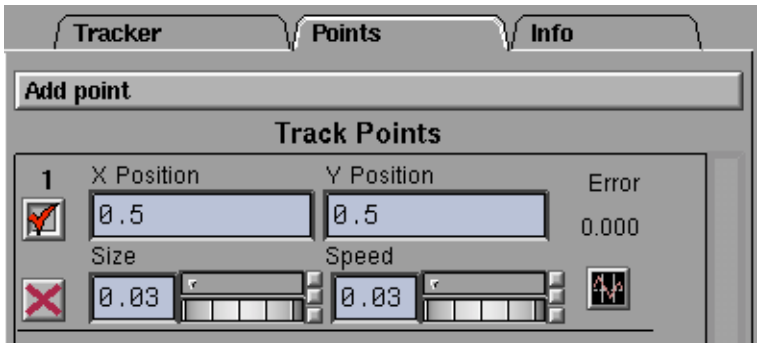
Frame Range Controls

The Frame Range controls enable you to set the frame range to be tracked. This is especially useful in situations when most of the frames tracked well, but you need to re-track a subset of frames using revised parameter settings and/or hand-set key frame points.

> **Note:**
>
> Each time you actuate the track, every enabled track point in the specified frame range will be re-tracked. However, you can disable (temporarily "turn off") any points that do not need re-tracking by using the Enable/Disable buttons in the Points tab.

Points Parameter Tab

The Points tab enables you to create, review, adjust, and delete track points.



Add Point

Click the Add Point button to add a track point to the monitor. You can add multiple points, if you desire. Chalice will create an entry in the list area for every point added that will enable you to set its position, size and speed parameters.

Track Point List Parameters

Each time you click the Add Point button, Chalice creates an entry in the list area on the Points parameter tab. Each entry in the list includes the following parameters:

Enable/Disable Point Button

The Enable/Disable Point Button, when checked, instructs Chalice to display the tracking point in the monitor and include it in all operations. If the box is deselected, the point will disappear from the monitor and will be masked from operations.

Delete Point Button

Click this red "X" to delete the point from the Track Monitor and from the Track Points List Area.

X Position, Y Position

These parameters enable you to specify the *x,y* coordinate location for a tracking point. Alternatively, you may use the work monitor display to drag the point in to a new location. Chalice will update the changes in the X position and Y position parameters accordingly.

Size, Speed

Track points are represented by a rectangle with crosshairs in the center (the Size parameter), surrounded by a circle (the Speed parameter):

- The **Size** parameter specifies the size of the rectangle, which defines the pixel area to be tracked and correlated. The Size rectangle should fit as closely as possible the feature to be tracked.
- The **Speed** parameter specifies the radius of the circle in pixels. This circle represents the size of the area to be searched by the Track node to find a good correlation of the pixels in the rectangle ("Size") area.

You set both the Size and Speed parameters in the list area by keying in numerical values or by using the mouse to set the tuner to the desired value.

Error

This parameter displays the correlation error value for 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.

STED Button

Click the STED button, available for each point tracked, to view the tracking data for that point in the Chalice Space-Time Editor.

UNIX NODE

The UNIX node enables Chalice users running IRIX on SGI workstations to pass a sequence of images to an external UNIX filter for processing.

Note:

The UNIX node is not available in the NT version of Chalice.

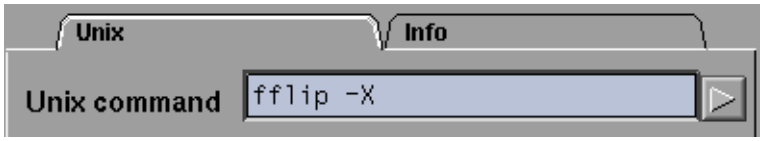
The UNIX node pipes image data to a UNIX filter by sending the data via stdin (standard input), executes the UNIX filter script, and pipes the data via stdout (standard output) back to the UNIX node.

This enables you to pass image data through any UNIX command or filtering program to perform a variety of image processing tasks. The UNIX filter must, however, be written in such a way that it accepts standard input and writes data back out to standard output.

You key the name of the desired UNIX command or filter into the UNIX Command parameter. When the UNIX node is executed, it will call the appropriate UNIX command or filter.

The UNIX node accepts one input.

UNIX Parameter Tab



UNIX Command

The UNIX command parameter enables you to specify a UNIX command or call a UNIX filtering program. Key the UNIX command or the name of the filtering program into the UNIX Command parameter data entry box.

Two examples are predefined and built into the UNIX node: Flip X (fflip-X) and Flip Y (fflip-Y). These commands call shell scripts that enable you to flip images horizontally or vertically. The source code for these two examples is available in the following directory:

```
usr/grail/chalice2.0/support/src/fflip
```

You can also call other UNIX filtering programs that accept input via stdin and write data to stdout to perform a variety of image processing tasks.

14 CONVERSION NODES

The Convert menu provides nodes for converting image files from one bit depth to another, as well as two nodes for premultiplying and unpremultiplying RGBA image data. The most commonly used conversion nodes are Linear Remap, Bit Reduce, and Bit Expand. The chart below recommends which node to use to convert to and from various image types:

	CONVERT TO:			
CONVERT FROM:	Cineon 10-bit Log	Float or 16-bit Linear	8-bit Linear	Cineon Linear
Cineon 10-bit Log	—	Disk Input node Bit Reduce node	Bit Reduce node	Disk Input node Cineon Insert node
Float or 16-bit Linear	Disk Output node	—	Linear Remap node Bit Reduce node	—
8-bit Linear	Bit Expand node	Linear Remap node Bit Expand node	—	—
Cineon Linear	Disk Output node Cineon Extract node	—	Bit Reduce node Linear Remap node	—

The **Linear Remap** node is strictly for converting linear data to a larger or smaller linear colorspace, such as 8-bit per channel linear data to 16-bit per channel linear data and vice versa. Linear Remap is easy to use and allows you to change the default remapping values based on the actual high and low values in your imagery.

Bit Reduce and **Bit Expand**, on the other hand, are optimized for converting nonlinear data to linear colorspace and vice versa. They offer several conversion methods and precise control over a number of parameters, however they are correspondingly more complex to use.

Bit Reduce and Bit Expand can also be used to convert linear data from one bit depth to another when you want to manipulate curves or to take advantage of the "intelligence" method, which creates conversion values based on pixels you select in the work monitor.

Cineon Insert and **Cineon Extract** are specialized conversion nodes: The Cineon Insert node converts 10-bit log data to linear colorspace using the "insert" method, which is an option for Cineon conversion provided in the Disk Input node. Cineon Insert is only used when you want to perform this specific conversion at some other point in the network. Similarly, Cineon Extract is only used when you want to convert this inserted linear data back to 10-bit log at some point in the network before the Disk Output node.

The **Premultiply** and **Unpremultiply** nodes do exactly as their names suggest. Premultiply multiplies the RGB channels of the input by the alpha channel, while Unpremultiply restores the original RGB channel values of a premultiplied RGBA image.

The Convert menu includes

Bit Expand Node ..... p. 14.2
Bit Reduce Node ..... p. 14.7
Cineon Extract Node ..... p. 14.17
Cineon Insert Node ..... p. 14.19
Linear Remap Node ..... p. 14.21
Premultiply and Unpremultiply Nodes ..... p. 14.23

BIT EXPAND NODE

The Bit Expand node enables you to convert linear imagery from a lower resolution format to a higher resolution linear format (16-bit or float) or to log (raw 10-bit Cineon) format.

Note:

Cineon 10-bit log is considered a higher resolution format because of the fact that it can hold the same amount of image information per channel as 19 bits in linear space.

The Bit Reduce node is the complement to this node, performing the inverse of the Bit Expand operations. For the most detailed explanation of these conversion methods, refer to the Bit Reduce Node description.

REVERSING A "BIT REDUCE" OPERATION

Most frequently the Bit Expand node is used to convert imagery that was previously "reduced" (converted) in a Bit Reduce node. To use Bit Expand for this purpose you merely have to select the specific Bit Reduce node that was used, and Bit Expand will perform the inverse of that operation. No other parameters need be set when using this option.

USING BIT EXPAND

For converting imagery that has not gone through a Bit Reduce node (see above), you can choose from four different conversion methods: Black & Rolloff, Cineon Softclip, Linear, and Curves.

Converting to Cineon

The **Black & Rolloff** or **Cineon Softclip** method should be used when converting data to Cineon 10-bit log or Cineon Linear formats.

Converting to Linear

When converting to straight 16-bit linear or float linear, use the **Linear** method as it expands the input data to fill the colorspace specified for output. If necessary, you can use the **Curves** method instead of Linear. The Curves interface is more complex, but it enables you to manipulate splines on a graph to control the conversion curve for each channel.

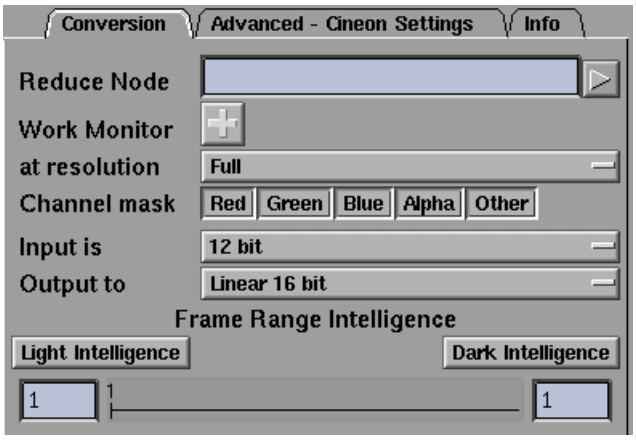
Follow these steps:

1. **Specify the Input and Output formats in the Conversion tab.**
2. **Bring up the Bit Expand work monitor.**
3. **Select the type of conversion process you want to use from the Conversion Menu.**
   Use the guidelines above to pick the method most suited to the converted data format.
4. **Adjust the default mapping parameters, if necessary, for the conversion method you have selected:**
   - For Black & Rolloff, Cineon Softclip, or Linear conversions, use Light or Dark Intelligence to provide starting points for in/out mapping. You can then adjust these values manually as needed.
   - For Curves, manually adjust the conversion curves at each channel by dragging value points on the curve. You can add new points on the curves as necessary. Use the Slopes menu to adjust the shape of the curve into and out of the value points.

Note:

Specific parameters for each method are described in the following sections.

Conversion Parameter Tab



"Reduce Node" Parameter

This parameter enables you to reverse the operations previously performed in a Bit Reduce node. You can either key the name of the appropriate Bit Reduce node directly into the entry field or select it from the associated popup menu, which lists every Bit Reduce node in the workspace.

Once you have specified the name, you will not need to enter any additional values or instructions. Chalice will automatically perform the inverse of each operation specified in the Bit Reduce node.

Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node, in which you adjust various conversion parameters. These parameters are described in the next section, "Bit Expand Work Monitor."

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel buttons.

Input Is

The Bit Expand node accepts 8-bit through 16-bit linear or float data as input. You will not need to set this parameter if your input is 8-bit linear. However, it is necessary to specify the format type for 10-bit through 16-bit input.

Output To

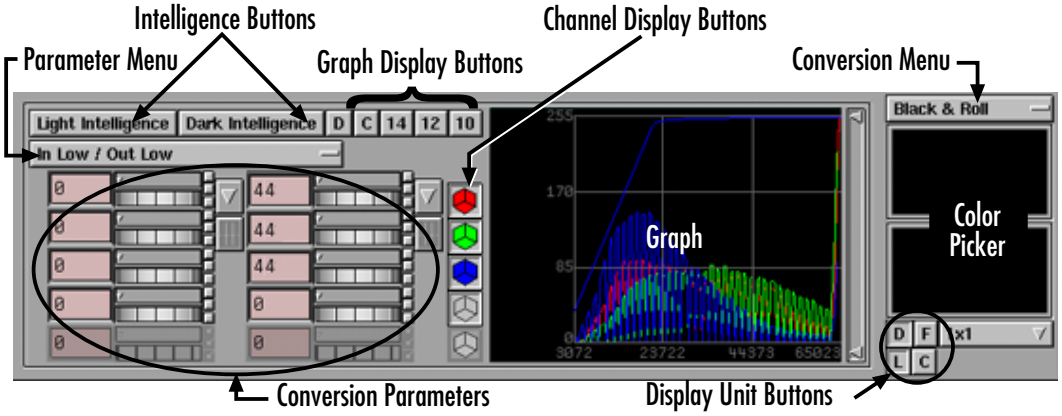
The Output To menu enables you to specify the data format of values sent to output. You can convert the input image to Cineon 10-bit log, Linear 16-bit, or Linear float.

Frame Range Intelligence

Frame Range Intelligence allows you to draw the histogram information used by Light and Dark Intelligence from a range of frames rather than from a single frame. You can enter values directly from the keyboard or use the slider bar to set the range. The Light Intelligence and Dark Intelligence buttons work exactly like the ones located in the work monitor, which are described in the next section.

Bit Expand Work Monitor

The work monitor provides parameters to control the format conversion of the input and output types specified in the Node Panel.



Light and Dark Intelligence

The Light Intelligence and Dark Intelligence buttons are used in conjunction with the Black & Rolloff, Cineon Softclip, and Linear conversion methods. Click the button (Light or Dark) most descriptive of the input image, and Chalice will generate mapping values based on the input image data. You can then adjust these values manually as needed.

For more information, see the description of Light and Dark Intelligence in the Bit Reduce node description.

Parameter Menu

The Parameter menu allows you to select the conversion parameters to display for the method you selected from the Conversion menu.

Conversion Parameters

These two columns of tuners enable you to change the default mapping values used by the selected conversion method: Black & Rolloff, Cineon Softclip, or Linear.

Because these conversion methods use different algorithms, the values you enter will vary depending on the type of conversion you have selected. Specific instructions for each conversion method are outlined in the Conversion Menu section below.

Note:

The Curves method requires a different, graphical interface.

Cascade Buttons

If you would like the value you have entered in the red channel to affect all channels equally, select the cascade button. This will spare you from having to key numerical values into each parameter.

STED Buttons

A STED button is provided for both columns. You can click either button to display the Chalice Space-Time Editor (STED), which will enable you to animate these parameters over time. For more information, see chapter 9, “Space-Time Editor (STED).”

Graph

The graph visually illustrates the conversion scheme specified by the controls. The horizontal axis represents the input image. The range will depend on the type of input selected in the Input Is menu on the Conversion tab in the node panel. The vertical axis represents the output values. These values will also vary depending on the type of conversion specified. Changes in the conversion parameters will be updated in the graph.

The Graph also displays a histogram for the color distribution of the input image at each channel. The horizontal axis represents the color values at a pixel. The vertical axis represents the number of pixels in the image with a particular color value. You can zoom in on sections of this histogram by sliding the triangular wedges located on the right of the graph.

Note:

Although the values on the horizontal axis are accurate for both curves, the vertical scale of the conversion curve and the histogram are very different. The values on the vertical axis represent output mappings, not the number of pixels for each color value. Therefore, the histogram display provides only general information about the shape of the color curve.

Display Depth Buttons

These buttons provide a graphical display of the conversion process in various formats. You can choose to display the data in Default view (i.e., the colorspace of the input image), Cineon view (0-4095), 14-bit (0-16383), 12-bit (0-4095), or 10-bit (0-1023). These values will appear on the horizontal axis. The vertical axis always displays the range of the format type sent to output.

Channel Display Buttons

These buttons turn on and off the graphical display for each color channel. All channels are turned on by default.

Color Picker

The Color Picker can display the RGBA value of any pixel or pixel area in an image. See the Bit Reduce Color Picker description for more information on how to use this tool.

Conversion Menu

This menu allows you to select the method Chalice will use to convert the input format to the output format. The default is Black & Rolloff. You can also use Cineon Softclip, Linear, or Curves. Each conversion method is described below.

Black & Rolloff

The Black & Rolloff conversion process enables you to optimize the conversion of an image file from a lower format, such as 8-bit linear, to a Cineon format, either Cineon Linear (16-bit or float) or raw Cineon (10-bit log). The default values are based on Cineon specifications.

The Black & Rolloff conversion method maps Low, Middle, and High color values from the input (left column of Conversion parameter tuners) to the output (right column).

The **Low** parameter specifies the lowest value included in the conversion. The **Mid** value represents the 90% density point. The **High** value sets the highest value in the conversion.

For a more information about this conversion process, see the Black & Rolloff Method section of the Bit Reduce node description.

Cineon Softclip

The Cineon Softclip setting enables you to convert imagery to a Cineon format using a conversion scheme based on Reference Black and White, Film Gamma and Display Gamma. The default values are based on specifications published by Cinesite and Kodak for this type of conversion.

The **Reference Black** specifies the lowest value included in the conversion. The **Reference White** value represents the 90% white value. The default is 4095, which maps to a 10-bit log value of 685.

Film Gamma refers to the gamma, or color distribution curve, of the input image. **Display Gamma** refers to the gamma of the monitor used to display the output image. The defaults are 0.6 and 1.0, respectively.

Note:

Refer to the equations in appendix C, "Softclip Conversion," for the precise conversion process used in the Softclip operation, and to the Cineon Softclip Method section of the Bit Reduce node description.

Linear

The Linear method is appropriate for converting imagery to a higher linear format: 16-bit or float. It is not recommended for Cineon output. Linear allows you to specify the high and low values of the conversion only. You cannot specify a midrange value for this method.

In the **Low** parameters, you specify the mapping from low in the input to low in the output. In the **High** parameters, you specify the mapping from the input high to the output high.

Curves

Like the Linear method, the Curves method is used to convert imagery up to 16-bit or floating point linear, not for Cineon conversion. Curves allows you to manually adjust the format conversion curves in a graph. Unlike the other conversion options, you do not specify values for predefined parameters. Instead, you alter the curves for each individual channel by manipulating splines on a graph using points and slope functions.

See the section of the Bit Reduce node description entitled "Bit Reduce Curves Editor" for a detailed description of how to use the tools in the Curves Editor panel.

Advanced - Cineon Settings Parameter Tab

The parameters on this tab represent default values that are used by functions in Bit Expand that convert to or from Cineon data.

In most cases, these parameters should be left at their default values, which are based on specifications for Cineon conversion published by Cinesite and Kodak. Refer to appendix C, "Softclip Conversion," for more information.

BIT REDUCE NODE

The Bit Reduce node enables you to optimize the conversion of linear image data to a lower bit depth, to retain as much of the relevant data from the original image as possible. It also optimizes the conversion of Cineon 10-bit log image data to a linear format.

The Bit Expand node is the complement to this node, performing the inverse of the Bit Reduce operations.

USING BIT REDUCE

Most commonly Bit Reduce is used to convert Cineon 10-bit log or Cineon Linear files to 8-bit linear. ("Cineon Linear" refers to a Cineon file that has been converted to 16-bit or float linear using the default conversion in the Disk Input node.) The Bit Reduce node features two conversion methods specifically designed for this purpose: Cineon Softclip and Black & Rolloff.

However, Bit Reduce also provides straight Linear conversion and the Curves method, in which you manually adjust the shape of distribution curves in a graph to control the conversion. These methods are suitable for converting 16-bit CGI to a lower bit depth.

The Cineon Softclip, Black & Rolloff, and Linear methods all provide default values which can be adjusted to achieve the best result for a particular image. For predominantly light or dark images, you can use Light and Dark Intelligence to generate appropriate starting-point conversion values.

The following steps provide a general guideline for using the Bit Reduce node:

1. **Specify the Input and Output formats in the Conversion tab.**
2. **Bring up the Bit Reduce work monitor.**
3. **Select the type of conversion process you want to use from the Conversion menu in the work monitor.**
4. **Adjust the default mapping parameters, if necessary, for the conversion method you have selected.**
   - The specific parameters available for **Black & Rolloff, Cineon Softclip,** and **Linear** conversion are describes in the "Conversion Parameters" section. For these methods, you can also take advantage of Light or Dark Intelligence to provide starting values for the remapping.
   - The **Curves** interface, in which you adjust the conversion curves for each channel by manipulating points and applying various slope functions, is described in the "Bit Reduce Curves Editor" section.

Choosing a Conversion Method

The Bit Reduce conversion method to choose will depend on your imagery and needs:

- Black & Rolloff and Cineon Softclip are designed for converting Cineon Log and Cineon Linear images.
- The Linear or Curves method should be used for non-Cineon 16-bit images, in which the color values are distributed across the entire colorspace.

Black & Rolloff v. Cineon Softclip

These methods are optimized for converting digitized film footage by providing the ability to apply a nonlinear curve to the highs to avoid harsh clamping. (Hence the terms "rolloff" and "softclip.")

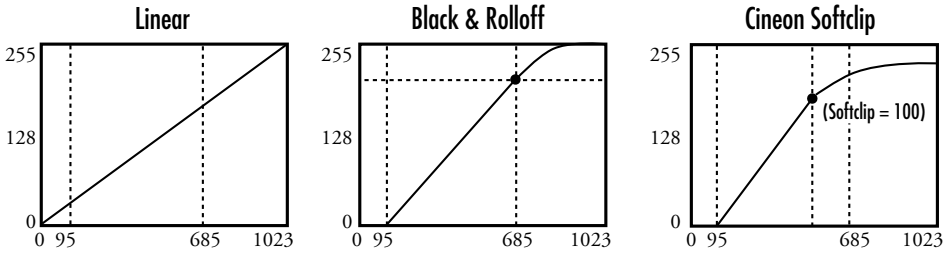
In **Black & Rolloff**, you specify the color values to use for the Low, Mid, and High values (for both the input data and the output). The conversion algorithm maps the values from Low to Mid in a linear curve and applies a nonlinear curve from the Mid to the High value to gradually roll off the highs.

In **Cineon Softclip**, however, you do not map the low, mid, and high values from one color-space to the other by assigning input and output values for each. Instead, you specify the low (Reference Black) and high (Reference White) values in the input data, and the conversion algorithm uses the Softclip value to compute a breakpoint above which the curve may become nonlinear.

If the Softclip parameter is left at its default value (which is 0), the distribution curve remains linear along its entire length from low to high. In other words, this is a "hard" clip of input values above Reference White.

As the softclip value increases, however, the slope of the curve from the breakpoint changes, gradually leveling off the curve for values above the breakpoint. The maximum recommended Softclip value is 100, which creates a curve from the breakpoint to the maximum color value of the input image.

Bit Reduce Conversion Methods
(10-bit Log to 8-bit Linear Examples)

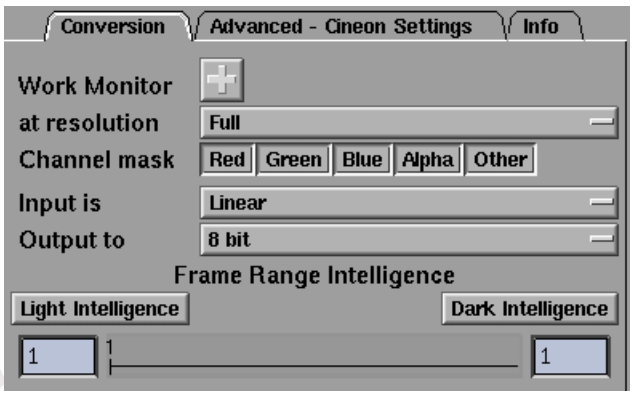


Linear v. Curves

The **Linear** conversion method is a straight linear remapping of values. You specify the Low and High values to map from the input to the output. **Curves**, on the other hand, enables you to specify the distribution curve for each channel by manipulating splines in a graph editor. (For this reason, the Curves method uses a different configuration in the monitor panel.)

Linear is the simplest method to use; Curves enables you to change the slope of the curve. Start by using Linear, and resort to Curves as your imagery requires.

Conversion Parameter Tab



Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node, in which you adjust various conversion parameters. These parameters are described in the next section, “Bit Reduce Work Monitor.”

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel buttons.

Input Is

The Bit Reduce node accepts log or linear data. Use the “Input is” menu to inform Chalice which format type will be read for conversion.

Output To

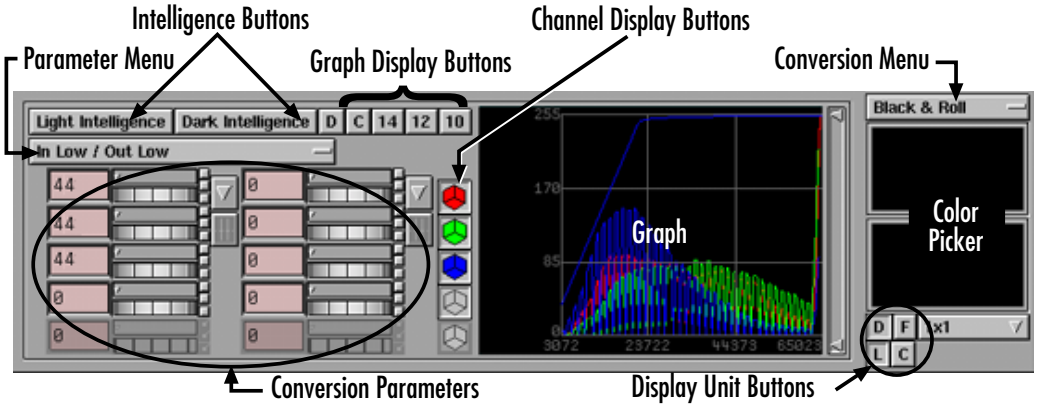
The Output To menu is used to specify the data format to which the input imagery will be converted: 8-bit, 10-bit, 12-bit, 14-bit, or 16-bit linear.

Frame Range Intelligence

Frame Range Intelligence allows you to draw the histogram information used by Light and Dark Intelligence from a range of frames rather than from a single frame. You can enter values directly from the keyboard or use the slider bar to set the range. The Light Intelligence and Dark Intelligence buttons work exactly like the ones located in the work monitor, which are described in the next section.

Bit Reduce Work Monitor

The work monitor provides parameters to control the format conversion of the input and output types specified in the node panel.



Light and Dark Intelligence

The Light Intelligence and Dark Intelligence buttons can be used in conjunction with three conversion methods: Black & Rolloff, Cineon Softclip, and Linear. (The exception is Curves.)

The default Low, Mid, and High conversion values used by these methods are optimized for “average” imagery. Because light images tend to contain more distinctions at higher color values than dark images, the optimal conversion values for predominantly light and dark imag-

ery differ somewhat. Light Intelligence and Dark Intelligence instruct Chalice to set the conversion parameters accordingly.

To use Light or Dark Intelligence:

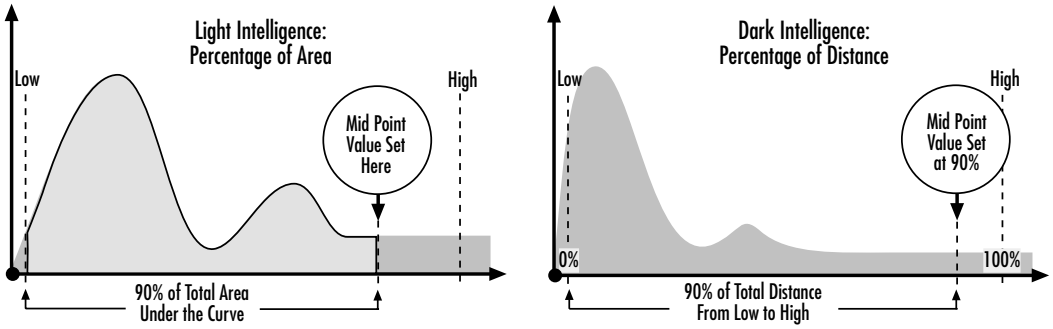
- Select the conversion method you want to use from the Conversion Menu.
- Then click the Intelligence button (Light or Dark) most descriptive of the image to be converted. Chalice will generate mapping values based on the selected conversion method.
- You can then adjust these parameter values manually as needed after viewing the initial result.

The Low and High parameters for all three conversion methods are set to the minimum and maximum values in the input image that occur above a certain frequency. These values remain the same for both Light and Dark Intelligence. (This means that you will achieve the same result by using either option when Linear is the selected conversion method, because the Linear method does not use a mid value.)

If you are using Black & Rolloff or Cineon Softclip, Intelligence will also generate a mid-range value. This value is entered as the Reference White parameter value for Cineon Softclip and the In Mid value for Black & Rolloff.

Dark and Light Intelligence use different algorithms to generate this value:

- Dark Intelligence returns the point at 90% of the distance between the Low and High values.
- Light Intelligence uses the histogram, returning the point where 90% of the area under the curve has been covered.



Parameter Menu

This menu allows you to select the conversion parameters to display for the type of conversion you specified in the Conversion menu. For example, if Black & Rolloff conversion is specified, this menu will allow you to view and change the In Low/Out Low, the In Mid/Out Mid, and the In High/Out High parameters in the tuners below the menu. The fourth conversion method, Curves, displays a different interface (to manually adjust curve shapes) in the monitor panel.

Conversion Parameters

These two columns of tuners enable you to change the default mapping values used by the selected conversion method: Black & Rolloff, Cineon Softclip, or Linear. For Black & Rolloff and Linear, the left column represents input image values to map to the output image values

in the right column. (Cineon Softclip is an exception.) The following "method" descriptions give guidelines for using these parameters.

The Conversion parameters display a pair of values for the currently selected Parameter menu item. Separate tuners are provided for each channel to allow you to change them individually. Values can be animated by double-clicking the STED button associated with each column to access the STED window. For more information, see chapter 9, "Space-Time Editor (STED)."

Tip:

Use the Cascade button (the downward pointing arrow next to each parameter column) to copy the value entered into the top tuner to the other tuners in the column. This shortcut saves you from typing the new value into all the fields.

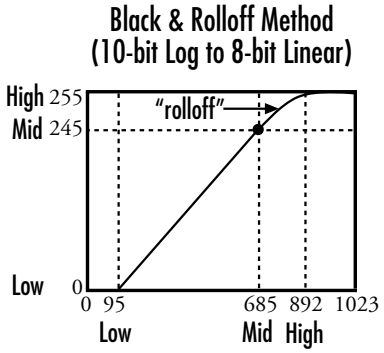
Use the "Display Unit" buttons to change how parameter values are displayed, to work with the numbers that are most applicable to the situation. For example, you can view the default Reference Black/Reference White parameter values used by Cineon Softclip either as Cineon (95 and 685 respectively) or as linear (44 and 4095).

Conversion Menu

This menu allows you to select the method Chalice will use to convert the input format to the output format. The default is Black & Rolloff. You can also use Cineon Softclip, Linear, or Curves. Each conversion method is described below.

Black & Rolloff Method

Black & Rolloff is used to convert Cineon-type files. The Black & Rolloff method enables you to specify Low, Middle, and High color values for the input (In column) and the output (Out column) colorspaces. The values from the Low to Mid point value are converted linearly, while the values from the Mid to High point are gradually "rolled off" in a nonlinear curve.



One advantage of the Black & Rolloff method is that you can inspect the input image to determine the actual image values on which to base conversion values. The default values are based on Cineon specifications, and take into account some of the characteristics of negative film.

For example, negative film is not perfectly clear; consequently, the lowest pixel values in the range are not used. So rather than map 0 in the input to 0 in the output, 44 in the input is mapped to 0 in the output.

The more unused pixels you can exclude from the conversion, the more room there will be to map meaningful pixels into the smaller bit space. Bit Reduce lets you specify the best Low, Mid, and High values for the particular image you are converting. Do you need more space for distinctions among lower values? Set the Mid value higher. Does the image have specular highlights you don't want to lose? Increase the High value and/or lower the Mid value.

The **Low** parameters specify the optimal black point. The default for the In Low is 44, the equivalent of 95 in 10-bit log space, which is mapped to 0 in the Out Low.

The **Mid** values set the top end of the midrange mapping points for the conversion. The default In Mid value is 4095, the equivalent of 685 in log space, which represents the 90% density point in the source image. By default it is mapped to 245 (8-bit) or 62965 (16-bit) for the Out Mid value. Values above this point will be gradually rolled off to the specified High value.

The **High** value represents the point at which the input will be clamped to maximum in the output. The default In High value is 20,000 (or 892 in log space). You may want to use the Inspect panel or other display tool to determine the best In High value to map to the Out High value, which is the maximum value in the output colorspace (i.e., 255 in 8-bit).

Cineon Softclip Method

Like Black & Rolloff, the Cineon Softclip method is used to convert Cineon 10-bit log and Cineon Linear image data to another colorspace, such as 8-bit linear. This method provides a Softclip parameter that can be used to control the slope of the conversion curve at the high end to eliminate the effects of harsh clamping.

The Softclip conversion assigns output values by taking the inverse log of the input Cineon data, then scaling and adjusting according to the Reference White, Reference Black, Film Gamma, Display Gamma, and Softclip parameter values, and the output format. (Refer to the equations in appendix C, "Softclip Conversion," for the precise conversion process used in the Softclip operation.)

The default parameter values for Cineon conversion are based on specifications published by Cinesite (1993) and Kodak (1995). Both of these documents are available for downloading from the Silicon Grail website (www.sgrail.com).

Reference Black specifies the lowest value included in the conversion. All pixels with color values below this number will be set to zero in the output image. The default value is 44 (95 in log).

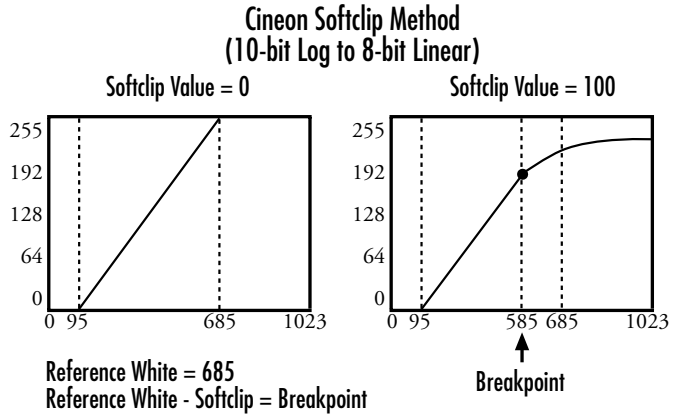
The **Reference White** parameter represents the 90% white value. The default value is 4095 (685 in log).

Film Gamma refers to the gamma, or color distribution curve, of the input image. **Display Gamma** refers to the gamma of the workstation monitor on which the output image will be displayed. The defaults are 0.6 and 1.0, respectively.

Softclip allows you to gradually level off the distribution curve for colors above Reference White. The Softclip value is subtracted from Reference White to create a breakpoint. The slope of the curve changes above the breakpoint, rolling off the highs.

Softclip should be considered a log value; that is, it is subtracted from the log value (i.e., 685) of Reference White, not the linear (4095).

The default Softclip value is 0, which will create a linear curve from Reference Black to Reference White, clamping all values above Reference White. A Softclip value of 100 creates a nonlinear curve, starting from the breakpoint, that gradually levels off to the maximum color value for the input image.



Linear Method

The Linear method is the most direct way to convert non-Cineon 16-bit linear imagery to a lower bit depth. As the name implies, it is a straight linear conversion that maps the specified low and high value in the input image to the specified low and high for the output. Chalice will assign all values in between in a linear fashion.

In the **In Low/Out Low** parameters, the left column represents the lowest input value to map to the low value specified in the right column (representing the output). All values below the In Low value will be clipped to the Out Low value.

In the **In High/Out High** parameters, the left column represents the highest input value to map to the corresponding value in the right column (the output). All values above the In High value will be clipped to the Out High value.

Curves Method

The Curves method allows you to manually adjust the format conversion curves in the graph. Unlike the other conversion options, you do not specify a value for predefined parameters, which are then used by a conversion algorithm to generate a distribution curve for the conversion. Instead, you alter the curves for each individual channel by manipulating splines on a graph using points and slope functions.

Selecting Curves from the Conversion menu changes the configuration of the Bit Reduce work monitor panel. The Curves Editor panel is displayed, which provides the tools needed to manipulate the conversion curves for each channel. The Bit Reduce Curves Editor section describes in detail how to use these tools.

Display Unit Buttons

These four buttons allow you to change the numerical units used to display values in the Bit Reduce work monitor. These settings affect the display of Conversion parameter values, Conversion Graph values, and Color Picker values. (The Color Picker display is controlled by the top row of buttons only.) The two sets of buttons work together as follows:

Top Row (Integer/Float):

- **D** displays values in integer at the bit depth selected for output.
- **F** displays values in floating point.

Bottom Row (Linear/Log):

- **L** displays values in linear.
- **C** displays values in Cineon log.

Conversion Graph

The graph visually illustrates the specified conversion scheme. The horizontal axis represents the input image. For example, if the input is 16-bit linear, the horizontal axis will be in the range of 0–65535. The vertical axis represents the output. For example, if 8-bit is the selected output, the vertical axis will range from 0–255.

Graph Display Buttons

These buttons can be used to change the data format in which the input image is displayed from the default. The default view (the "D" button) displays the horizontal axis in the same range as the input.

Default View
Cineon View (0-4095)
14-bit View (0-16383)
12-bit View (0-4095)
10-bit View (0-1023)

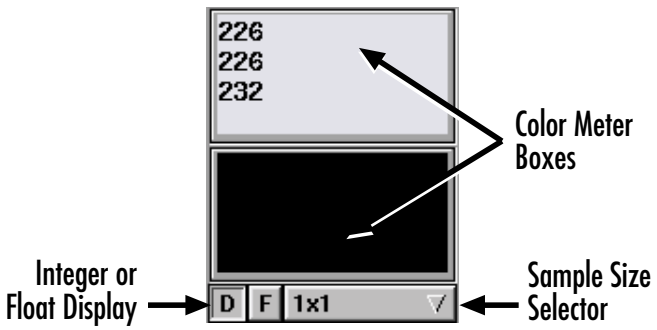
The graph also displays a histogram of the color distribution of each input channel. You can zoom in on sections of this histogram by sliding the triangular wedges located on the right of the graph.

Channel Display Buttons

These buttons turn on and off the histogram display for each input channel. All channels are turned on by default.

Bit Reduce Color Picker

The Color Picker features two color boxes, each of which can display the RGBA value of any pixel or pixel area in an image. The Color Picker is especially useful for comparing color values in two different images using A/B Wipe.



To use the Color Picker, simply click the mouse in one of the two **Color Meter boxes**. The cursor will change into an eye dropper—this is the Color Picker tool.

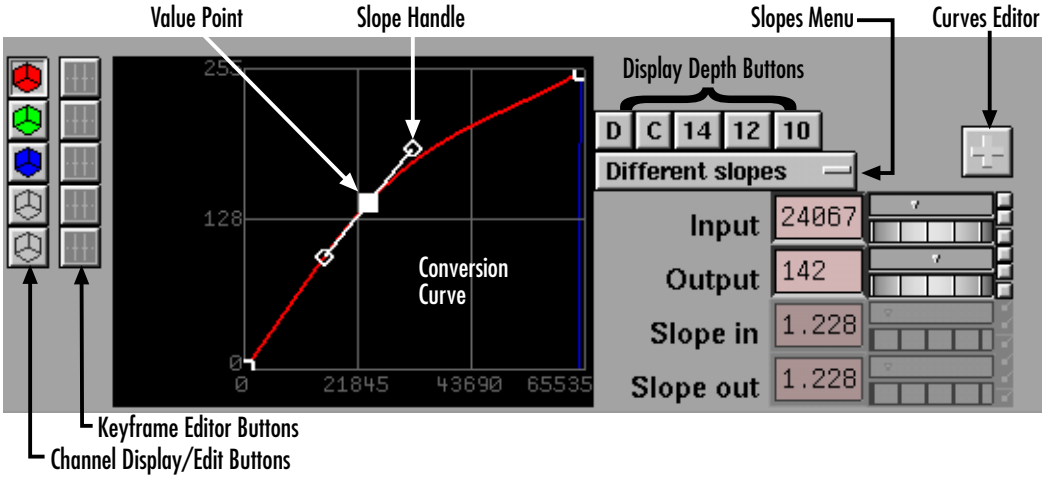
Then click on any pixel in the image that you want to meter. The color values of that pixel will appear in the Color Meter box and the cursor will revert to normal shape. If you are sampling an area of pixels, their average will be displayed.

To sample an area of pixels, click the **Sample Size Selector** to select the appropriate size pixel grid from the popup menu: 1x1, 3x3, 5x5, 7x7, 9x9, or 11x11.

Color Picker values can be displayed as integer or float. Click the "D" button to display integer values (in the selected output depth). Click "F" to view the values in float.

Bit Reduce Curves Editor

The Curves Editor panel appears when Curves is selected from the Conversion menu:



In the Curves Editor panel, you can manually adjust the conversion curves in the graph. The Channel Display buttons determine which curves are displayed and activated for editing. The Slopes menu enables you to apply various function curves.

You can **add** a new value point anywhere on a curve by clicking on that part of the curve. To **delete** a point, click on the point to select it and then press the Delete key on the keyboard. To **move** a value point, click on it and drag it to a new location.

Tip:

The Curves Editor Button (+) enables you to bring up an identical Curve Editor panel in a resizable window. Increasing the size of the graph can make it easier to select and manipulate points and curves.

Channel Display Buttons

These buttons allow you to display or suppress the color curve for a given channel. Only curves that are displayed can be altered.

Keyframe Editor Buttons

These buttons allow you to set key frames to vary the curves over time. Click the button once to activate the channel for animation. Then you can use the Key Framer on the node panel to set key frames, or click the Keyframe Editor button again to bring up the Keyframe Editor window for the associated channel.

In the Keyframe Editor, select a key frame by clicking it once (it will highlight). You can move the key frame by dragging it. You can copy or delete any selected key frame by right-clicking in the editor window to display a popup menu with these two choices. When you select "Copy," the currently selected key frame is copied to frame number under the cursor.

Slopes Menu

The Slopes menu allows you to specify the type of curve that goes into and out of each value point. The following choices are available:

- **Continuous:** This setting keeps the slope into and out of a value point constant. It allows you to drag the value point to a new location without altering any other aspect of the curve.
- **Equal Slopes:** This setting allows you to vary both the position and slope into and out of the selected value point. When you choose this option from the Slopes menu, a Slope Handle will appear. By clicking and dragging these handles, you can alter the slope of the curves into and out of the value point. However, this setting will ensure that these slopes remain equal.
- **Different Slopes:** Value points with a Different Slopes setting are controlled in the same manner as Equal slopes. However, this setting permits variance in the slope into and out of a value point.
- **Linear:** This setting keeps the slope into and out of a value point linear. The value of Slope In and Slope Out will be determined by the position of the point relative to adjacent points.

Value Point Parameters

The Input, Output, Slope In, and Slope Out tuners display values for the currently selected point on a curve in the graph. As an alternative to dragging a point or handle to reposition it, you can key specific values into the data entry fields to remap values.

The **Input** parameter displays the color value of the selected point for the image being converted, while the **Output** parameter displays the corresponding value for the converted output image.

The **Slope In** and **Slope Out** parameters display the slope into and out of a specified value point. The minimum and maximum values are 0 and 10, respectively. If the Equal Slopes or Different Slopes function has been selected from the Slopes menu, the fields become active and you can key values into the data fields to define the slopes.

Curves Editor Button

This button brings up a larger graph than the one provided in the workspace. It provides a resizable window that can make selecting and moving curves easier. Otherwise, it has the same functionality as the graph in the Curves Editor panel.

Advanced - Cineon Settings Parameter Tab

The parameters in this tab represent default values that are used internally by the Bit Reduce node as part of the Cineon Softclip algorithm. They do not represent values sent directly to output.

In most cases, these parameters should be left at their default values, which are based on specifications published by Cinesite and Kodak. They are provided for those specialists in film digitization who may wish to change the defaults for certain sequences. Refer to appendix C, "Softclip Conversion," for more information.

CINEON EXTRACT NODE

The Cineon Extract node enables you to convert Cineon Linear imagery back to native Cineon 10-bit log format. “Cineon Linear” refers to Cineon log data that has previously been converted to 16-bit linear using the “insert” method, which maps the log 90% white point to 90% white in linear space (e.g., 4095 in 16-bit). Cineon Extract performs the inverse operation of the Cineon Insert node.

In most cases, you would convert Cineon Linear back to Cineon log in the Disk Output node. However, the Cineon Extract node gives you the option of converting Cineon Linear back to 10-bit log at an earlier point in a network, in case you want to perform an operation on log data before output.

Note:

To convert any other type of data (any type of data besides Cineon Linear) to 10-bit log format, use the Bit Expand Node.

Cineon Extract Parameter Tab

The Cineon Extract tab provides controls for the Log 90% White card, the Film Gamma, and the Linear 90% White code value. These parameters control the overall brightness and color distribution of the output image. The defaults are based on Cineon specifications.

Log 90% White

The Log 90% White parameter enables you to set the values of the 90% white card, which is used as the reference for a relative log exposure of 0 in 10-bit log space. The default value is 685.

Film Gamma

The Film Gamma parameter enables you to set the conversion values for film gamma, which controls the straight-line portion of the characteristic curve for typical negative film. The default value is 0.6.

Linear 90% White

The Linear 90% White value allows you to specify the 90% point in the linear output. The Log 90% White point maps to this value. The default is 4095.

The above parameters contain separate entries for the Red, Green, Blue, Alpha, and Other parameters, enabling you to vary the conversion from channel to channel. If you wish to en-

ter a value into all parameters, key the value into the Red parameter and click the Cascade button. You can also suppress an operation for a channel using the Channel Mask buttons.

About the Conversion Algorithm

The values in the above parameters specify certain variables in the conversion equation. The operation sends out a scaled, adjusted logarithmic conversion of the linear input. Note that the equation is the inverse of the Cineon Insert conversion:

Output = log10 (Input / Linear 90% White) / (0.002 / Film Gamma) + Log 90% White

The above equation is based on the algorithm provided in "Conversion of 10-bit Log Film Data To 8-bit Linear or Video Data for the Cineon Digital Film System," published by Kodak Motion Picture & Television Imaging.

For more information on the Log 90% White card, Film Gamma, and the Linear 90% White code value, see either the document referred to above or "Grayscale Transformations of Cineon Digital Film Data for Display, Conversion, and Film Recording," published by the Cinesite Digital Film Center.

Both of these documents are available for downloading from the Silicon Grail website (www.sgrail.com).

CINEON INSERT NODE

The Cineon Insert node converts raw Cineon film data to Cineon Linear format. “Cineon Linear” refers to 10-bit log data that has been converted to 16-bit or floating point linear using the “insert” method, which remaps the Cineon 90% white point to 90% white in linear space (e.g., 4095 in 16-bit) to preserve any highlight data above the white point.

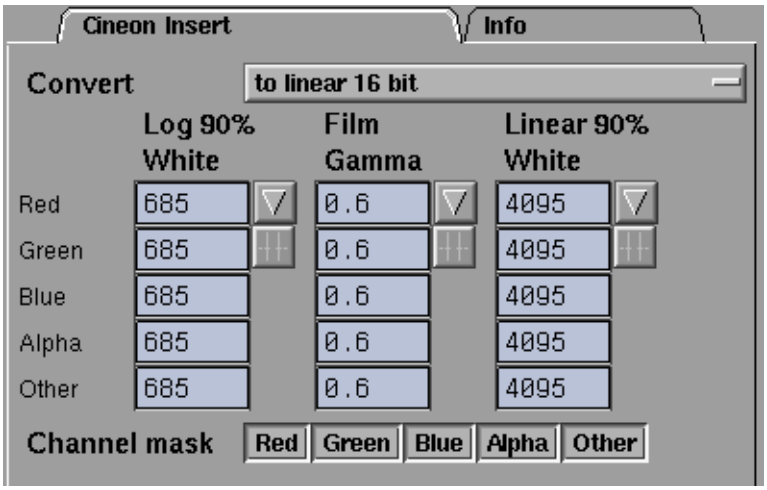
Cineon log files are usually converted to a linear format when they are loaded into a Disk Input node. However, this conversion can be disabled when you wish to work with 10-bit log data. If at some point later in the network you wish to work with Cineon Linear, you can use the Cineon Insert node to perform the insert method of conversion on the log data.

Note:

To convert Cineon 10-bit log to any other format, such as 8-bit linear or straight 16-bit linear, use the Bit Reduce Node.

Cineon Insert Parameter Tab

This tab provides controls for the Log 90% White card, the Film Gamma, and the Linear 90% White code value. These parameters provide control over the shape of the conversion curve. The defaults are based on Cineon specifications.



Convert

This menu enables you to select whether to convert to integer (16-bit) or floating point data. This will not affect the conversion process itself; instead, it determines how the output values will be interpreted.

Log 90% White

The Log 90% White parameter enables you to set the values of the 90% white card, which is used as the reference for a relative log exposure of 0 in 10-bit log space. The default value is 685.

Film Gamma

The Film Gamma parameter enables you to set the conversion values for film gamma, which controls the straight-line portion of the characteristic curve for typical negative film. The default value is 0.6.

Linear 90% White

The Linear 90% White parameter enables you to set the values for the 90% white code in the conversion to linear color space. The default value is 4095.

The above parameters contain different entries for the Red, Green, Blue, Alpha, and Other parameters, enabling you to vary the conversion from channel to channel. If you wish to enter a value into all parameters, key the value into the Red parameter and click on the Cascade button. You can also suppress an operation at a given channel using the Channel Mask buttons.

About the Conversion Algorithm

The Log 90% White, Film Gamma, and Linear 90% White parameters specify certain variables in the conversion process. The operation involves a scaled, adjusted inverse log of the logarithmic (base 10) input data:

OUTPUT = 10^((INPUT – Log 90% White) * Film Gamma / 0.002) * Linear 90% White

The above equation is based on the algorithm provided in "Conversion of 10-bit Log Film Data To 8-bit Linear or Video Data for the Cineon Digital Film System," published by Kodak Motion Picture & Television Imaging.

For more information on the Log 90% White card, Film Gamma, and the Linear 90% White code value, see either the document referred to above or "Grayscale Transformations of Cineon Digital Film Data for Display, Conversion, and Film Recording," published by the Cinesite Digital Film Center.

Both of these documents are available for downloading from the Silicon Grail website (www.sgrail.com).

LINEAR REMAP NODE

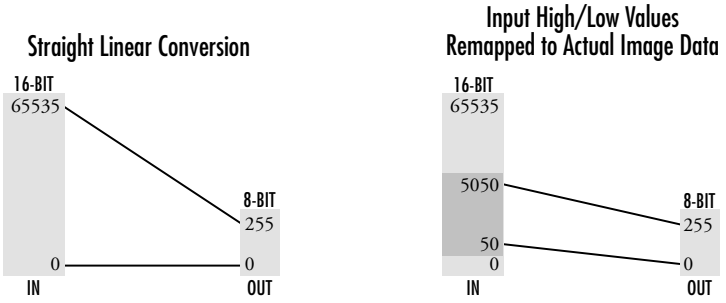
The Linear Remap node enables you to convert image data to and from 8-bit, 16-bit, and floating point representations.

The Linear Remap node automatically determines the data format of an input node and displays it in the In column. You simply select the output format from the Out column popup menu.

OPTIMIZING LINEAR CONVERSION

The Linear Remap node also enables you to rescale linear data, which can be helpful for data optimization. For example, assume that you wish to convert 16-bit data to 8-bit data. The 16-bit colorspace ranges from 0 to 65535 bits; 8-bit colorspace ranges from 0 to 255 bits.

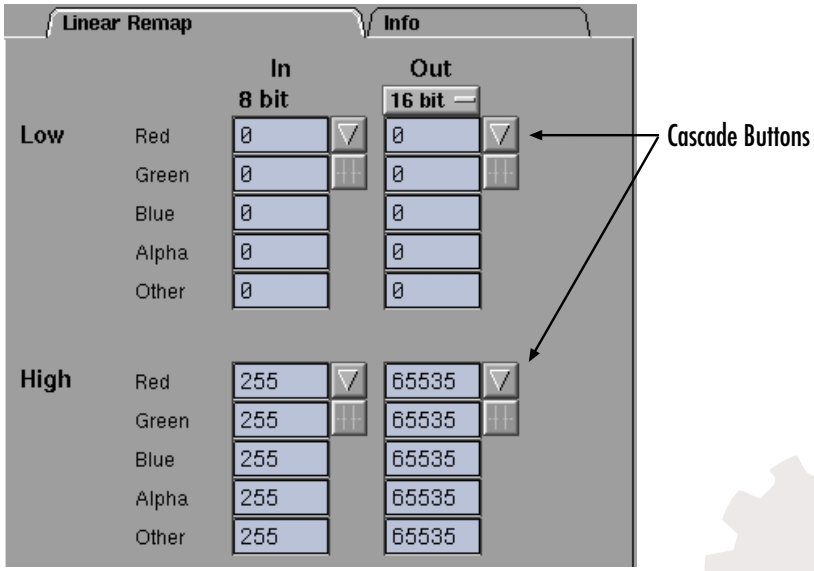
However, in this particular case, imagine that your 16-bit data occupies a range of only 50 to 5050. The linear conversion tables of the Linear Remap node enable you to scale the actual range of your input data (50 to 5050) down to the full range of the 8-bit color space (0 to 255). This saves you from trying squeeze data in the range of 5051 to 65535 into the relatively narrow 8-bit color space and enables you to spread the actual range of your input data over the maximum possible range of the output data format you specified.



Tip:

To find out the useful range of any specific imagery, use the Inspect panel or the Histogram utility monitor available in all Chalice monitors.

Linear Remap Parameter Tab



In Column

The Linear Remap node automatically senses the data file format of the input node and displays it in the In column.

Out Column

The Linear Remap parameter tab allows you to specify the output data format. Click on the popup menu to select 8-bit, 16-bit, or Float(ing point) data representations.

Using the Cascade Buttons

The Cascade button spares you from repeatedly keying the same value into the Red, Green, Blue, Alpha, and Other channel parameters.

When you click the Cascade button, Chalice replicates the value you key in for the Red channel parameter throughout the Green, Blue, Alpha, and Other channel data entry boxes.

PREMULTIPLY AND UNPREMULTIPLY NODES

The Premultiply and Unpremultiply nodes each take a single RGBA input. For every pixel in the input image, the Premultiply node multiplies the value of each color component (the RGB channels) by the corresponding alpha channel value. The Unpremultiply node, on the other hand, attempts to restore the RGB channels of a premultiplied RGBA image to their original, non-premultiplied values.

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, 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 Chalice. Composite nodes such as Mcomp, Atop, Inside, Outside, and Over will premultiply the input images for you, unless you specify in the composite node that 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 Chalice is used strictly for color correction.

How Premultiply Works

The Premultiply operation is automatic and does not require user modification; therefore the node panel features a standard Chalice Info tab only.

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 of the image:

Argb * Aa

The multiplication in this formula is fractional, 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 fraction first by dividing it by an appropriate value: for 8-bit, the alpha is divided by 255; for 16-bit, by 65535.

Restoring Non-premultiplied RGB Channel Values

The Unpremultiply node attempts to restore the original, non-premultiplied RGB values of a premultiplied RGBA image. The accuracy of this operation varies, however, depending 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.

Computer generated imagery, such as the output from a 3D rendering program, is almost invariably premultiplied. However, you may want to manipulate the alpha channel data to adjust the matte characteristics before compositing the image over a background. If the image is already premultiplied, the pixel values can become skewed.

For example, in a non-premultiplied image, a full red pixel with half alpha coverage would be represented as [1, 0, 0, .5]. After premultiplication, the same pixel would be represented as [.5, 0, 0, .5]. Assume that you adjust the matte such that the alpha channel value for this pixel changes from .5 to .75.

If this matte adjustment occurred after the pixel had been premultiplied, the value would become [.5, 0, 0, .75], whereas if the adjustment occurred before premultiplying, the pixel value would become [.75, 0, 0, .75]. These two values would generate a different result for the same pixel in a comp. Therefore, you may choose to run the image through an Unpremultiply node before adjusting the matte.

How Unpremultiply Works

The Unpremultiply operation is automatic and does not require user modification; therefore the node panel features a standard Chalice Info tab only.

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 of the image:

Argb * Q

Q represents the alpha channel value, modified as follows (assuming floating point):

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 fraction first by dividing it by an appropriate value: for 8-bit, the alpha is divided by 255; for 16-bit, by 65535.)

15 COMPOSITE NODES

The Composite nodes enable you to combine two or more image sequences in a variety of ways. Several concepts essential to digital compositing, including alpha coverage and premultiplication, are discussed in the first section of this chapter, "Formulas and Terminology for Computing Composite Nodes." The functionality of some composite nodes is closely related. Therefore, this chapter describes the nodes in several groups, starting with the multi-input nodes.

The Mcomp node can composite an unlimited number of RGBA image inputs using the "over" operator. The other multi-input node, Zcomp, is used instead of Mcomp when the image layers to be composited include z-depth channel data (RGBAZ):

Mcomp Node 15.4
Zcomp Node 15.6

The next group of nodes composite two image inputs based on the way in which the alpha channel data affects the other channels of the inputs, as specified in each node description:

Over, Atop, Inside, and Outside Nodes 15.9

The third group of nodes composite images based on the pixel values of the two input nodes:

Diff, Minus, Multiply, and Plus Nodes 15.11

The rest of the composite nodes, which are not particularly related, are described last:

Dissolve Node 15.13
MinMax Node 15.14
Ultimatte AE (AdvantEdge) Node 15.15

With the exception of the multi-input nodes, Mcomp and Zcomp, all of the Composite nodes require two inputs, which must match in resolution and bit depth.

Compositing Disparate Imagery

To use any of these nodes to composite inputs of different dimensions, bit depths, or frame ranges, you must use an adjustment node between one of the input nodes and the composite node to make sure the data is appropriately prepared for compositing.

For example, you may need to use a Transform or Scale node to adjust the image size, a Bit Reduce or Bit Expand node to change bit depth, or a Sequence node to reconcile disparate frame ranges. For more information about these adjustment nodes, refer to the relevant node descriptions elsewhere in this manual.

Formulas and Terminology for Computing Composite Nodes

In the descriptions of the various composite nodes that follow, we resort to the following terminology. "A" denotes the A image, or top input; "B" denotes the B image, or bottom input; and "R" denotes the resulting composite 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, while "Argba" would refer to all four channels of the A image. Consequently, the formula "Rrgba = Argba + Brgba" would mean "Rr = Ar + Br," "Rg = Ag + Bg," and so on.

Fractional Multiplication

All multiplication in these formulas is assumed to be fractional. For 8-bit and 16-bit images, this means that the alpha channel value ("a") is converted into a fraction by dividing it by an appropriate value (for floating point images, it is assumed that the alpha channel value is already in the range of 0–1). The alpha is then multiplied by the color value (U):

8 bit values	R = U * a/255
16 bit values	R = U * a/65535
floating point values	R = U * a

Alpha Channel

The alpha channel contains the transparency/opacity information for each pixel of the RGB channels it accompanies. In Chalice, 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 Chalice node monitors as a monochrome image where white represents a value of 1 and black represents 0.

In this manual, the terms *alpha channel* and *matte* can be considered synonymous, except where the finer distinction is needed between an alpha channel defined as a node output channel and a matte defined as a temporary channel of matte data used for a specific type of image manipulation (as in certain Ultimatte matte operations).

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 any over-type composite, where the alpha channel value determines how much the RGB channels contribute to the result. In Chalice, these operators include the Over, Mcomp, Zcomp, Inside, Outside, and Atop nodes.

About Premultiplication

First of all, 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.

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.

It is important to know the premultiplication status of an image because premultiplication is an initial step in the computation process of a compositing operation. If you specify (using a checkbox or popup menu in a composite node panel) that an image is already premultiplied, Chalice will skip this step. But if you specify that an input image has not been premultiplied, Chalice will premultiply it, as follows:

Argb = Argb * Aa (to premultiply the A image)
Brgb = Brgb * Ba (to premultiply the B image)

Tip:

If you don't get the results you expect when using Mcomp, Atop, Inside, Outside, or Over, double-check the premultiplication setting of the inputs in the node panel.

Premultiplication with Ultimatte

When you specify that an image is premultiplied, Chalice checks the value of the alpha channel for each pixel in the image. If the alpha is zero, Chalice assumes that the RGB contribution to that pixel is also zero.

However, if an image to be composited has been processed through an Ultimatte node, this may not be the case—the RGB channels may contain nonzero values even when the alpha channel equals zero. Therefore, when compositing such imagery using the Mcomp or Over nodes, you should select the "Is Ultimatte" setting from the popup menu accessed by clicking the Premultiplied button on the node panel.

Note:

The Ultimatte AdvantEdge composite node does not have a premultiplication selector; it assumes that the top input is an Ultimatte processed foreground image, such as the output of an Ultimatte PFG matte node.

MCOMP NODE

The Mcomp node enables you to perform multilayer compositing on a virtually unlimited number of image inputs. You simply connect imagery from upstream nodes to the input connectors of the Mcomp node. Each time you add a new input to the Mcomp node, another input connector is automatically created.

Input Layers

You can connect imagery from upstream nodes in any order, and you can easily change the order of the inputs in the Layers tab of the node panel without reconnecting them. You can also temporarily disable layers to see the composite without the contribution of some inputs.

Note:

Because the Mcomp node is not limited to a specific number of inputs, and you can add and delete inputs to a particular composite, the top input to the Mcomp node is always the background image, while each successive input is layered over the previous one in a series of "over" operations. (Refer to the Over Node description if you need more information about this operation.)

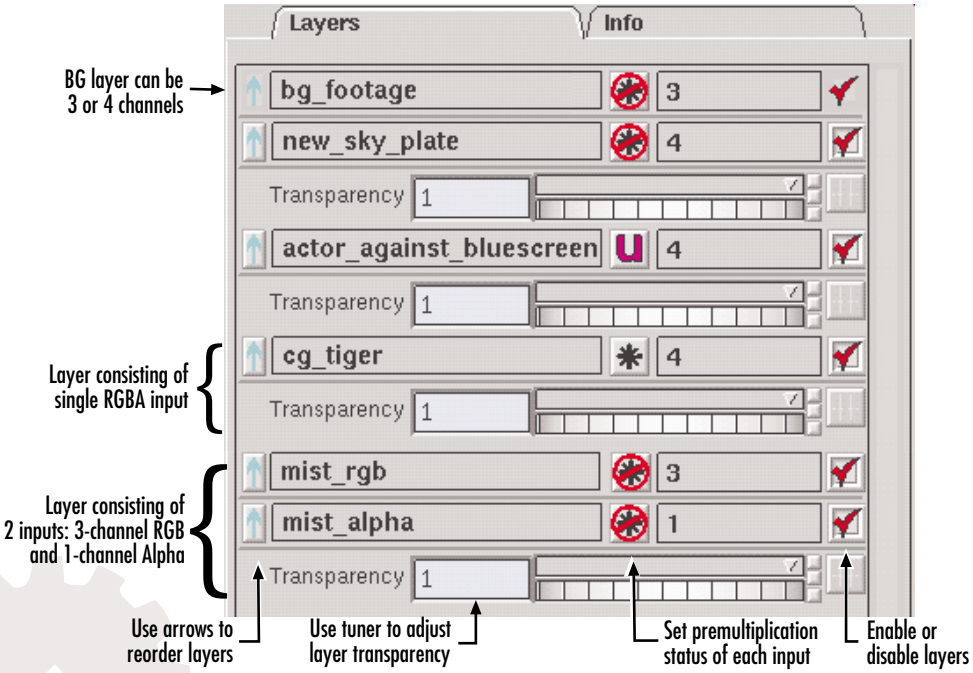
Input Methods

The Mcomp node does not require an alpha channel for the background image (the top input in the Mcomp parameter list). However, an alpha channel is required for all other layers in the composite, either as part of a single input (RGBA, for example) or as the second of a pair of inputs (an 3-channel RGB input followed by a 1-channel Alpha input).

You can use either or both input styles within a single Mcomp node; that is, one layer can be a single input, such as RGBA, and another layer can consist of two separate inputs, such as an RGB image followed by an alpha input.

If the background layer is an RGB image (no alpha), when computing the composite Chalice treats the input as if there were an alpha value of 1 for every pixel.

Mcomp Parameter Tab



When you create a new Mcomp node, the Mcomp parameter tab is initially blank. A new layer entry is created for each input you connect to the Mcomp node. The controls provided for each layer are described below.

Transparency Tuner

The Transparency parameter enables you apply a multiplier value 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 this parameter. Therefore, the default value of 1 specifies that the input values of the alpha channel will be used for the matte.

Values greater than one will increase the opacity of the matte; values less than 1 will decrease the opacity. For example, a value of 0.5 would reduce the opacity of the matte by 50 percent, and a value of 0 would create a black (transparent) alpha channel.

Premultiplication Status

Click and hold the Premultiplied button to display a popup menu that enables you to specify whether the input image for that layer is premultiplied (CGI), not premultiplied (live action), or Ultimatte (bluescreen). For more information, see About Premultiplication.

…is premultiplied (CGI)

…is not premultiplied (live action footage)

…is an Ultimatte image

Tip:

Use the Data Format Preferences panel to set default state for this parameter to match the type of data you use most. This panel is described in chapter 11, “Setting Preferences.”

Layer Checkbox

This feature lets you turn an individual input layer off and on without disconnecting and reconnecting the associated input image from the Mcomp node. Click the Layer icon to toggle it to the opposite state. When this button is checked, the layer is enabled. When unchecked, the layer is disabled, and the associated Transparency parameter will be grayed out.

The background layer (top input) must be enabled; therefore, the Layer button for the background is grayed out.

Up Arrow

You can quickly and easily reorder each input layer in the composite. Pressing the Up Arrow icon for a particular layer moves the layer one step closer to the top of the list. The top entry in the Mcomp list of inputs is actually the bottom, or background, layer of the composite.

Note:

If you use a three-channel image for the background, it must remain the bottom layer of the composite, since the over operation used by the Mcomp node is based on the alpha coverage of the foreground image.

ZCOMP NODE

The Zcomp node enables you to use both z-depth and alpha channel information to composite multiple layers of imagery. (If your images do not contain z-depth data, use the Mcomp node to do multilayer composites.)

The Zcomp node uses 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.

Input Methods

The Zcomp node enables you to use one of two input methods, Combined or Split:

- If you select the **Combined** input option, each layer of the composite must be a five-channel (RGBAZ) input image.
- The **Split** input option enables you to use a pair of inputs for each layer, with the first input being a four-channel RGBA image and the second input being the Z channel for that image.

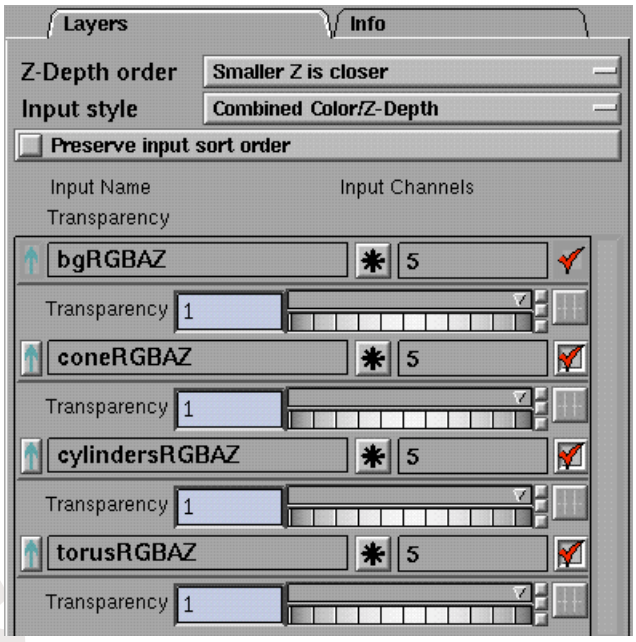
  Note: If you are using the Split input style, each RGBA input must be followed in order by its associated z-depth channel input. If you disconnect one of the pair of inputs, the other input is automatically disconnected.

When using pairs of inputs in Split mode, the RGBA and Z images do not have to be the same bit depth. For example, you can use 8-bit RGBA inputs with floating point z-depth inputs. However, all of the RGBA inputs must be the same bit depth, and all of the Z inputs must be the same bit depth.

The Zcomp node accepts a virtually unlimited number of inputs, with a minimum of two inputs using Combined style or four inputs using Split style. The output of the Zcomp node is an RGBAZ image when using Combined style; when using Split style the output is RGBA.

Layers Parameter Tab

The Layers parameter tab enables you to specify the Z-Depth Order and Input Style to use. Be sure to set these two parameters to match your input data.



Each time you add an image to the Zcomp node, a corresponding layer is created in the Layers tab. Whether the image is a single input (RGBAZ image) or a pair of inputs (RGBA image and separate Z channel), only one new entry is added to the list.

Note:

When the Split input style is used, (RGBA image and separate Z channel), each layer will be labeled with the name of the RGBA input.

Z-Depth Order

You must set this parameter to match your z-depth channel data. If larger values represent pixels that are closer, use the default setting, select **Larger Z is closer** from the Z-Depth Order popup menu. On the other hand, if smaller values represent pixels that are closer, select **Smaller Z is closer.**

Input Style

You must set this parameter to indicate whether you are using a single input or a pair of inputs for each layer. The default setting, **Split Color/Z-Depth**, specifies that you are using pairs of inputs, with the information split between an RGBA input and a Z channel input. If you are using single RGBAZ inputs, select **Combined Color/Z-Depth** from the Input Style popup menu.

Preserve Input Sort Order

The Zcomp node uses z-depth data to determine the order in which each pixel will be layered in the composite. If the z-depth values for two pixels are equal, however, you may wish to specify that the input order of the layers be used to sort the pixels. The input order is the order in which the layers are listed in the Layers tab of the node panel, and this order can be changed by using the Up Arrow icons.

This option must be checked to ensure that the input order is retained. It is not active by default, however, because it slows performance of the Zcomp node.

Transparency

The Transparency parameter enables you apply a multiplier value 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 this parameter. Therefore, the default value of 1 specifies that the input values of the alpha channel will be used for the matte.

Premultiplied Button

Click and hold the Premultiplied button to display a popup menu that enables you to specify whether the input image for that layer is premultiplied, not premultiplied, or Ultimatte. For more information, see the Premultiplication Status section of the Mcomp node description.

Up Arrow

You can quickly and easily reorder each layer in the list. Pressing the Up Arrow icon for a particular layer moves that layer one step closer to the top of the list. The top entry in the list of layers is actually the bottom, or background, layer of the composite.

Note:

The order of the layers in the list is superseded by the z-depth channel values. Therefore, the order of the layers is relevant only when z-depth values are equal and you have checked the Preserve Input Sort Order option.

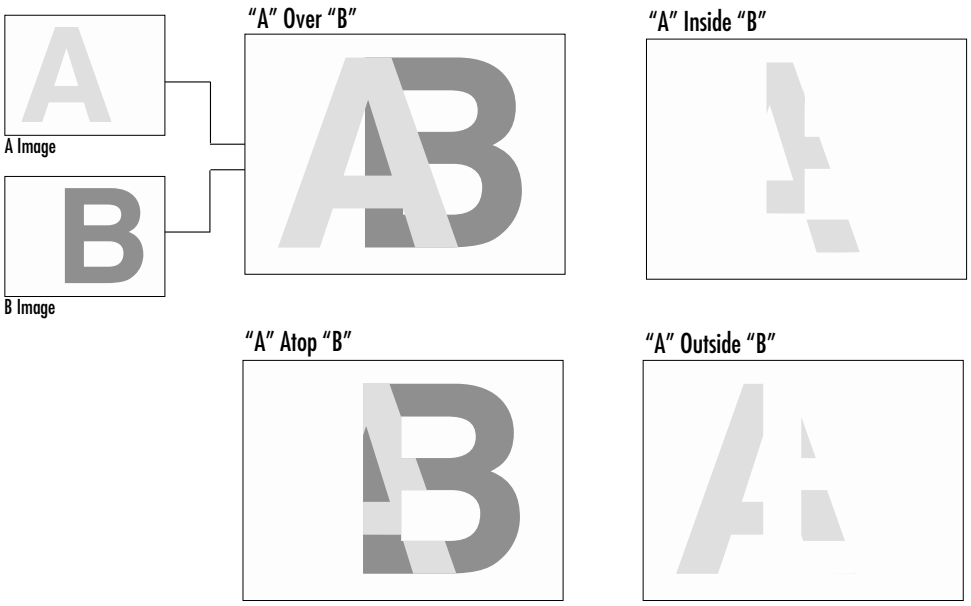
Layer Checkbox

This feature lets you turn an individual input layer off and on without disconnecting and reconnecting the associated input image from the Zcomp node. Click the icon to toggle it to the opposite state. When the box is checked, the layer is enabled. When unchecked, the layer is disabled, and the associated Transparency parameter will be grayed out.

The background layer (top input) must be enabled; therefore, the checkbox for the background is grayed out.

OVER, ATOP, INSIDE, AND OUTSIDE NODES

Unlike the Mcomp and Zcomp nodes, these four composite nodes take only two inputs each, which must be four channels (RGBA). The exception is the Over node, for which the background input does not require an alpha channel.



The Over and Atop nodes require you to specify the premultiplied status of both inputs; for the Inside and Outside nodes, only the premultiplied status of the top (A) image input is relevant as the RGB data of the bottom (B) input does not contribute to the result. For more information, refer to About Premultiplication at the beginning of this chapter.

Note:

For the Over node, you can also input an Ultimatte image. Refer to Premultiplication with Ultimatte at the beginning of this chapter if you need more information.

Over Node

The Over node places a foreground image over a background image in an "A over B" composite. This is one of the most commonly used composites. The A image is placed over the B image wherever the A image has alpha coverage. The Over node accepts two inputs (only the A image input requires an alpha channel) and performs the following computation:

Rrgba = Argba + Brgba * (1 – Aa)

Click and hold the Input Image A or Input Image B button in the Over parameter tab to display a popup menu that enables you to specify whether the image is premultiplied, not premultiplied, or Ultimatte.

Tip:

To do an "under" operation, simply reverse the inputs to the Over node.

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:

Rrgba = Argba * Ba + Brgba * (1 – Aa)

The Atop parameter tab provides checkboxes that enable you to specify whether or not Image A and Image B are premultiplied.

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). 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.

The Inside node accepts two inputs (note that only the B image requires an alpha channel) and performs the following computation:

Rrgba = Argba * Ba

The Inside parameter tab provides a checkbox that enables you to specify whether or not Image A is premultiplied.

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. Effectively, the B image acts as a matte, and the Outside and Inside nodes can be used to create complementary mattes from the same B image.

The Outside node accepts two inputs (note that only the B image requires an alpha channel) and performs the following computation:

Rrgba = Argba * (1 – Ba)

The Outside parameter tab provides a checkbox that enables you to specify whether or not Image A is premultiplied.

DIFF, MINUS, MULTIPLY, AND PLUS NODES

These four nodes composite two images based on the pixel values of two input nodes as specified in the following descriptions. These nodes do not require user modification of parameters; therefore their node panels feature an Info Tab only.

The two required inputs to the Diff, Minus, Multiply, or Plus node must have the same number of channels, however, each input image can consist of from one to 10 channels.

Tip:

To adjust the number of channels in an image, use the Channel Swap node.

The order in which the inputs are connected to the nodes (i.e., top or bottom) is irrelevant, with the exception of the Minus node, as described below.

Diff Node

The Diff 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. The Diff node accepts two inputs and performs the following computation:

$R = |A - B|$

For example:
200 - 50 = 150
50 - 250 = 200
0.5 - 0.25 = 0.25
0.25 - 0.5 = 0.25

Minus Node

The Minus node subtracts the values of the B image from the values of the A image. The Minus node accepts two inputs and performs the following computation:

$R = A - B$

The Minus node is similar to the Diff node; however, the Minus node will clamp integer values at zero (floats are not clamped).

For example:
200 - 50 = 150
50 - 250 = 0
0.5 - 0.25 = 0.25
0.25 - 0.5 = -0.25

Multiply Node

The Multiply node multiplies the values of the A image by the values of the B image. Multiply accepts two inputs and performs the following computation:

$R = A * B$

The Multiply node will clamp integer values at maximum (255 for 8-bit; 65535 for 16-bit).

For example:
20 * 10 = 200
30 * 10 = 255 (if 8-bit)
0.5 * 1.0 = 0.5
10.0 * 100.0 = 1000.0 (no clamping on floats)

Plus Node

The Plus node adds the values of the B image to the values of the A image. The Plus node accepts two inputs and performs the following computation:

R = A + B

The Plus node will clamp integer values at Vmax.

For example:

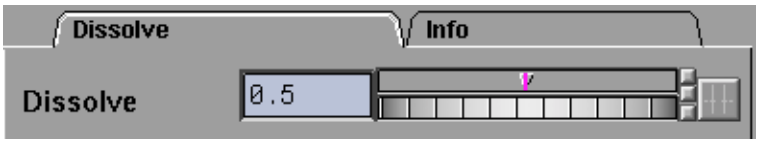
200 + 10 = 210
200 + 60 = 255 (if 8-bit)
65000 + 10 = 65010
65000 + 7000 = 65535 (if 16-bit)
0.5 + 0.2 = 0.7
0.5 + 5 = 5.5 (no clamping on floats)

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 adding the result to the B image multiplied by (one minus the dissolve value):

$$R = [A * V] + [B * (1 - V)]$$

Dissolve Parameter Tab



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% dissolve, and a value of 1 represents no dissolve from the A image.

Animating the Dissolve Value

You can use the STED functionality associated with the Dissolve parameter to control the rate of dissolve over time, or to change the flow of the cross-dissolve such that the B image dissolves into the A image. In the STED, a single function segment in the graph represents the entire dissolve operation by showing the A image transparency throughout the frame range.

To cross-dissolve evenly from image A to image B, for example, at the first frame the A image is at maximum value (opaque). You would typically use a linear function to bring the A image evenly from maximum value at frame 1 to the minimum value at the final frame. Of course, different functions can be applied to create pulsating or other effects.

On the other hand, if the A image is set at 0.5 transparency uniformly across the graph, the result would be a composite image where both inputs were 50% transparent throughout the range of frames.

Note:

A complete description of the Chalice "Space-Time Editor (STED)" is available in chapter 9.

MINMAX NODE

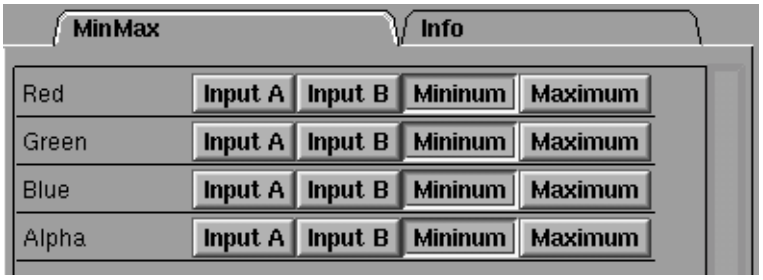
The MinMax node combines two inputs by comparing each channel on a pixel-by-pixel basis and selecting either the lower (MINimum) or higher (MAXimum) value for the output, as you specify in the MinMax node panel. The most common use for the MinMax node is to combine two mattes by selecting the maximum alpha channel values of the two inputs.

The MinMax node enables you to specify the output value for each channel individually. You can specify that Chalice use

- the minimum value as the output for any channel
- the maximum value as the output for any channel
- the A or B input value as the output for any channel

MinMax Parameter Tab

The MinMax tab creates an entry 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.

ULTIMATTE AE (AdvantEdge) NODE

Ultimatte AdvantEdge is a composite node optimized for foreground imagery processed with the Ultimatte PFG node (see chapter 16, “Matte Nodes” for complete descriptions of the other Ultimatte nodes).

Ultimatte AE enables you to 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 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.

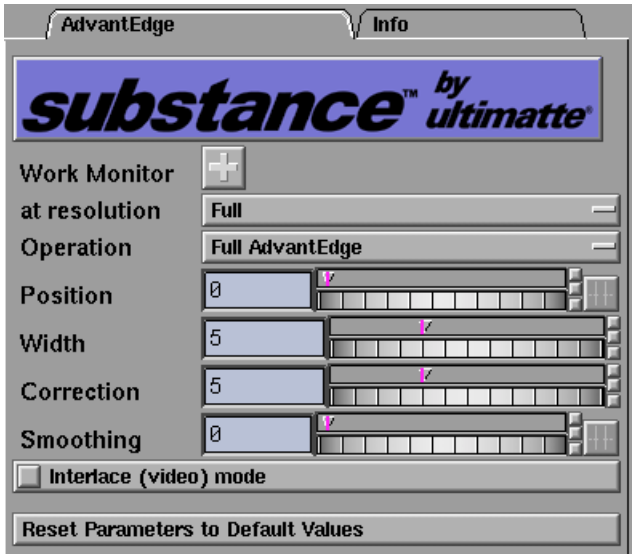
The AdvantEdge node replaces the compositing step performed by the Mcomp node in the Basic Ultimatte Network illustrated in Overview of the Ultimatte Process in chapter 16. The node requires two inputs, a foreground image and a background image. The foreground image must contain an alpha channel.

Note:

Unlike the Mcomp node, Ultimatte AdvantEdge assumes that the premultiplication style of the foreground input is Ultimatte. For more information, refer to About Premultiplication at the beginning of this chapter.

AdvantEdge Parameter Tab

The AdvantEdge parameters allow you to specify how the edges of the foreground image will be adjusted in the composite. You can also access a work monitor to view the areas AdvantEdge has identified for correction.



Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node, which shows the composite image with a custom overlay superimposed on it. This overlay visually highlights transition areas of the matte (matte edges) identified by the node.

Work Monitor Display Parameters

The work monitor provides parameters to control the appearance of the overlay display. These do not affect the output; however, they enable you to control the color and transparency of the overlay, and whether the overlay covers edge or non-edge areas:

- The **Invert** button determines whether the overlay will cover edges or non-edges. By default, the overlay display will cover all areas not identified as edges, displaying edge areas as they currently appear in the composite.
- The **Transparency** tuner controls the overlay, in a range of 0 (transparent) to 100 (opaque). To see how the composite looks without the overlay, set this value to 0.
- The **Overlay Red, Green, and Blue** parameters set the color of the overlay. By default, the overlay display is translucent red.

Operation

The Operation parameter specifies the type of correction to perform:

- **Horizontal AdvantEdge** specifies that the correction and smoothing operations search horizontally only; that is, that only 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 AdvantEdge** specifies that the correction and smoothing operations search vertically only; which means that the upper or lower inside edges are searched. Vertical AdvantEdge will correct edges that run horizontally.
- **Full AdvantEdge** applies both Horizontal and Vertical operations.
- **No AdvantEdge** is useful if you would like to see how your composite would look without applying any edge correction.

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 detail loss within the edge area.

Smoothing

The Smoothing parameter smooths hard or jagged object edges in the direction specified in the Operation menu. Smoothing is not often recommended, as it can degrade fine detail.

Interlace (Video) Mode

This parameter, when checked, configures AdvantEdge for video-type frames 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.

16 MATTE NODES

The Matte nodes enable you to create and modify matte channel data for use in compositing and other image processing operations. They generally take a single input, with the following exceptions:

- The Chromakey node takes a second, optional input for use as a control image to specify which pixels of the primary input image will be processed in the operation.
- The Ultimatte PFG node takes a second, optional input for use as a garbage matte.
- The Ultimatte CSC node requires a second input (a clean plate), which is used as a reference for the screen correction operation.

The Matte nodes include:

Chromakey Node 16.2
Rotospline Node 16.5
The Ultimatte Nodes 16.13
Ultimatte CSC (Classic Screen Correction) Node 16.18
Ultimatte GK (Grain Killer) Node 16.22
Ultimatte RSC (Roto Screen Correction) Node 16.26
Ultimatte PFG (Processed Foreground) Node 16.36
Ultimatte CC (Color Control) Node 16.44

CHROMAKEY NODE

The Chromakey node enables you to create an alpha channel matte based on the colors of the input image. This node uses the values of the original image within a narrow range of hue and saturation to create the matte.

The Chromakey node features Setup and Chroma Select parameter tabs. The Setup tab enables you to specify the type of output and access the Chromakey work monitor. The hue and saturation are controlled by using a color wheel on the Chroma Select tab. The default value is the common "bluescreen," with colors centered on a hue of 230, plus or minus 10 (that is, hues 220 through 240) and saturations in the range of 0.5 to 1.0.

The Chromakey node accepts one or two inputs:

- the primary (top) input image, which must have an alpha channel, and
- the optional (bottom) input, which is used as a control image for the chromakey process.

Setup Parameter Tab

Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node, which enables you to select hue and saturation parameter values interactively, as described in Using the Chromakey Work Monitor in the Chroma Select parameter tab description.

Output

The Output popup menu enables you to select whether the Chromakey node should output "All Channels" available in the source image, the "Alpha Channel Only," "Inverted Alpha Only," or "All Channels, Invert Alpha."

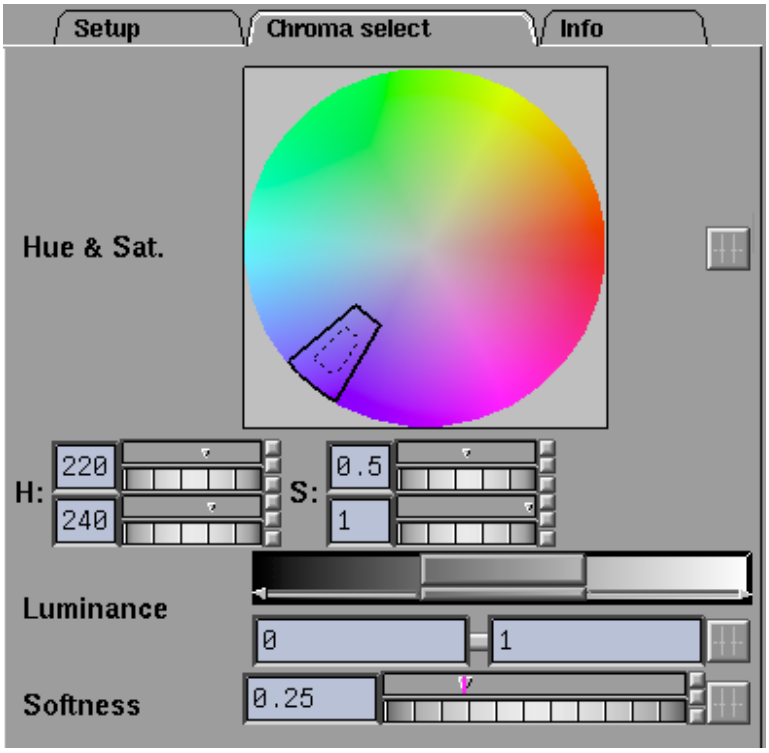
Control Channel

The Control Channel parameter enables you to specify which channel of the control image to use. The control image is the optional second input to the Chromakey node. The value of each pixel of the selected channel in the control image determines whether the corresponding pixel in the primary input image will be processed.

For example, if a pixel in the control image is black (i.e., has a pixel value of 0), the corresponding pixel in the primary image will not be processed. However, if a pixel in the control image has a value other than 0, the corresponding pixel in the primary image will be processed.

Chroma Select Parameter Tab

The Chroma Select parameter tab enables you to specify the Hue and Saturation, Luminance, and Softness for an alpha channel matte.

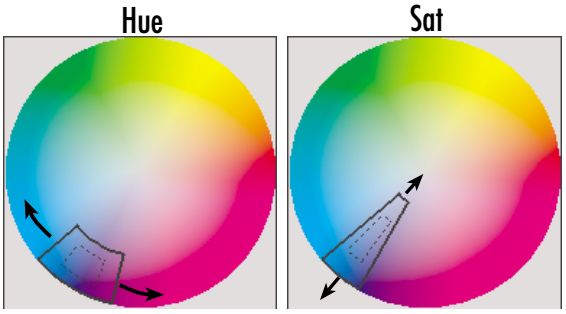


Hue and Saturation

The Hue and Saturation parameters enable you to define the range for both hue and saturation by manipulating a color wheel (see below), keying in values, or setting tuners.

Using the Color Wheel

You can specify a color range by using the color wheel, which contains a selection box with borders that you can click and drag. Drag the dashed selection box to adjust the hue. Click the outer borders of the box and drag radially (around the wheel) to adjust the total width of the selection box and thus the hue range. Click the outer borders of the box and drag axially (towards or away from the center of the wheel) to adjust the total length of the selection box and thus the saturation range.



Using the Chromakey Work Monitor

You can also identify a color range by clicking on an area in your source image, as displayed in the Chromakey work monitor. (This monitor is accessed from the Setup parameter tab.)

Click on an area in the image that contains the desired range of color. The color range that you select from the image will automatically be entered as values in the Hue and Saturation fields, and the selection box will shift to the corresponding location in the color wheel.

Clicking and dragging through a range of colors in the monitor image will enter corresponding values for the Hue and Saturation parameters.

Tip:

To include a color range from a different part of the source image, hold down the Shift key while clicking and dragging the mouse.

Using the Tuners and Data Entry Fields

Alternatively, you can key in values or set the appropriate tuner. For the Hue (H) parameter, you can key in a value or set the tuners to a value between 0 and 360, inclusive. For the Saturation (S) parameter, you can key in a value or set the tuners to a value between 0 and 1, inclusive.

Two pairs of tuners are provided for setting the Hue and Saturation ranges: the Hue tuners control the radial boundaries of the selection area (in the Color Wheel); the Saturation tuners control the axial boundaries.

Luminance

This parameter enables you to select a range of luminance to use for the matte. Click and drag the ends of the arrows to set the range. The associated fields will update the high and low range values accordingly. Alternatively, you can key values from 0 to 1 directly into the fields.

You can also use the Chromakey work monitor to select luminance values by clicking in the alpha channel of the source image. Select Alpha from the Channel Display button popup menu in the Monitor Toolbar to display the matte data. The Chromakey node will create a luminance value based on the pixel area you have selected in the monitor.

Softness

The Softness parameter enables you to control the softness of the matte by keying in a numerical value between 0 and 1.0 or using the mouse to set the tuner to the desired value.

A value of 0 represents no softness, i.e., a hard, opaque matte. The closer the value gets to 1, the softer (more transparent) the matte becomes.

ROTOSPLINE NODE

The Rotospline node enables you to create and manipulate matte shapes with precision and control. Most commonly this functionality is used to create 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.

A Rotospline matte can also be used as a "stencil," or mask; that is, to isolate a portion of an image for further manipulation such as color correction. For example, a matte could be drawn to isolate an automobile that was filmed in black-and-white, thus enabling you to colorize only the car while leaving the background monochrome.

The Rotospline node can be used to create a control image, or mask, for another node, such as Vector Warp or Color Correct.

The Rotospline node features a work monitor that provides you with tools for drawing matte shapes around portions of an image. You can edit, scale, rotate, invert, duplicate, and animate these shapes.

The drawing tools enable you to draw both straight (linear) and curved (spline) line segments. You control the splines by dragging handles that extend from the spline's anchor points.

You can display the source image in the work monitor, to use as a reference when drawing a shape that will be used to isolate a specific object or area in the image, or you can display the matte channel only.

The Rotospline work monitor also enables you to access a key frame editor to set and delete key frames to control the interpolation of shape characteristics over time.

You can create as many matte shapes as you need for an image, and for each shape you can set different parameter values (such as frame range, opacity, and edge characteristics). You can also invert the pixel values of any shape, and use it as a cutout to punch a hole in another shape. And you can change the order in which the shapes are layered.

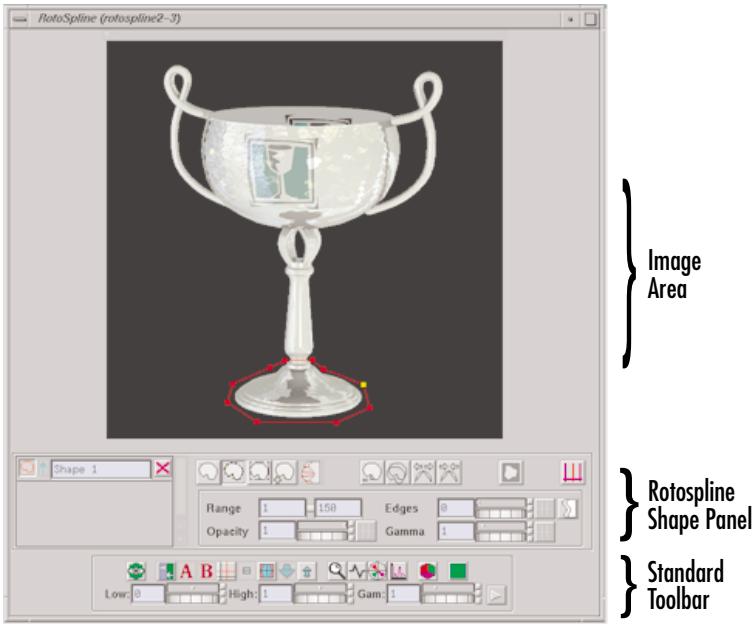
The Rotospline node accepts one input, which may or may not have an alpha channel:

- If the alpha channel of the source image is unsuitable for any reason, you can clear the incoming alpha to black or white, or you can invert it.
- If the source image does not have an alpha channel, the Rotospline node will generate one in which every pixel is black or white, as you choose.

The node output is either an RGBA or a one-channel (alpha) image, as you specify.

Using the Rotospline Work Monitor

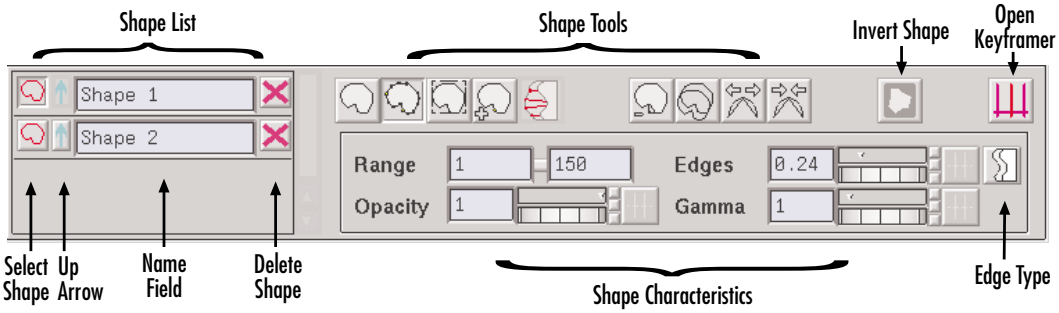
In addition to the standard Monitor Toolbar, the Rotospline work monitor panel provides a shape panel with controls for creating and editing shapes. The shape outlines are displayed over the input imagery in the image area of the work monitor.



Rotospline Shape Panel

The Rotospline shape panel consists of three sections:

- a **Shape List** section, for selecting, naming, reordering, and deleting shapes;
- a **Shape Tools** section, for drawing and editing shapes; and
- a **Shape Characteristics** section, for setting frame range, opacity, edge type, and other shape parameters.



Shape List

The Shape list displays a list of every shape you create. As soon as you create or replicate a shape, a new entry appears in the list.

Select Shape

Click the Select Shape button to make the corresponding shape the active shape. All of the other parameter settings in the panel apply to the currently selected shape.

Up Arrow

The top entry in the Shape list is the bottom layer of the matte image. Each time you create a new shape, that shape is layered over any previous shapes. The Up arrows are used to reorder the list, and consequently the order of the shapes themselves. The order in which shapes are layered affects how overlapping shapes interact.

Click the Up icon for a shape to move that shape one step closer to the top of the list, which is one layer back in the image.

Name
You can use the Name field to assign descriptive names to shapes. Simply select the current name, key in a new name, and press the Enter key.

Delete Shape
Click the Delete Shape button to delete the corresponding shape. Both the shape and the list entry are deleted.

Shape Characteristics
These parameters apply to the currently selected shape. You can select another shape and apply a different set of values to the same parameters.

Opacity, Edge, and Gamma values can all be animated over time by clicking the associated STED button to access the Chalice Space-Time Editor. For more information, see chapter 9, "Space-Time Editor (STED)."

Range
This parameter enables you to specify a range of frames in which the currently selected shape will appear by keying a starting and ending frame number into the data entry fields.

Opacity
This parameter enables you to set the opacity (or relative transparency) of the currently selected shape. A setting of 1 shows pure white (completely opaque) inside the matte; a setting of 0 shows pure black (completely transparent) inside the matte; intermediate values show shades of gray.

You can set the Opacity parameter by keying in a numerical value between 0 and 1 or by using the mouse to set the tuner to the desired value. The default value is 1.

Edge Type
This parameter controls the softness, or blur, of the matte edge of the currently selected shape. Select one of three options from the Edge Type popup menu:

No Edges
Obviously, this option (the default) specifies that no blurring be applied to the edges.

Uniform Edges
This option blurs every edge evenly by the value you enter in the associated tuner, which only becomes active when this option is selected. A value of 0 generates no blurring and a value of 1 produces an extremely soft, blurred edge that is 10 percent of the total width of the image.

Nonuniform Edges
This option enables you to apply a different edge to each spline. This parameter is a special case, in that (unlike the Uniform edge option):

- nonuniform edges must be used in Edge Mode (click the Edge Mode button in the Shape Tools menu to select Edge Mode), and
- nonuniform edge values are adjusted by manipulating edge points in the work monitor, not by using the tuner.

With Edge Mode selected, the points of the selected shape change appearance (they look like hollow squares) to indicate that they can be manipulated to set the width and direction of the edge blur. Simply drag one of these "edge points" to a new position and examine the resulting blur.

When you select another mode, such as Point Mode, you will see that the actual anchor point of the spline has not moved from its original position.

Invert

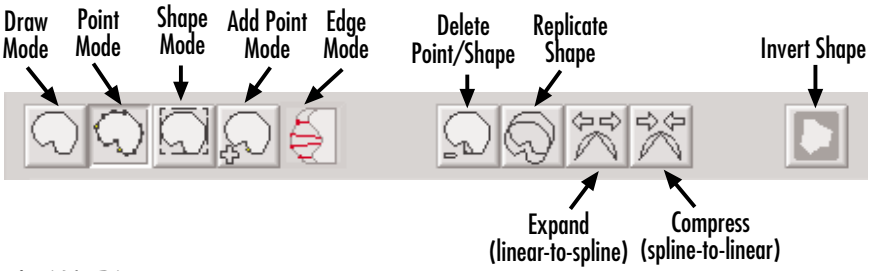
Click the checkbox of this shape-specific parameter to invert the pixel values of the matte (e.g., white changes to black, and vice-versa).

Tip:

Invert is an easy way to create cutouts. For example, to quickly create a donut-shaped matte, you could first create a large circular shape, then create a small circular shape and invert it, and finally center the small inverted circle over the large one.

Shape Tools

The Shape Tools section features nine buttons that control the drawing and editing tools and open the Rotospline key frame editor.



Draw Mode (Alt D)

Draw Mode enables you to draw straight (linear) and curved-line (spline) segments. When you open the Rotospline work monitor for the first time, Draw mode is selected by default (once you have created a shape, the monitor defaults to Point Mode when opened).

Click the Draw Mode button to enter Draw Mode.

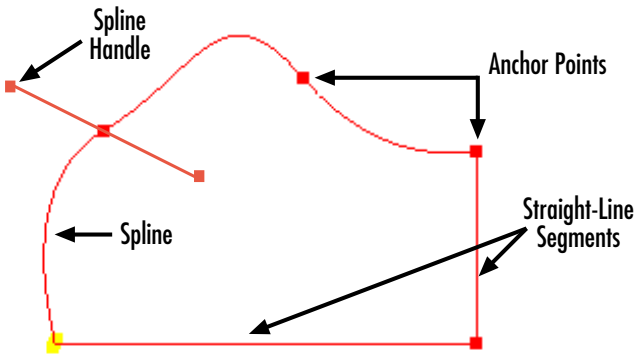
- To draw a **linear segment** (straight line), click anywhere in the image area to create an anchor point, move the mouse, then click again to create a second anchor point. Chalice draws a linear segment connecting the two points. Continue moving and clicking the mouse to add additional segments.
- To draw a **spline** (curved line), click and drag the left mouse button. The spline will terminate when you release the mouse button. You will be able to adjust the slope of the curve by dragging the rotating handles that extend from the anchor points of the spline.

Point Mode (Alt P)

Click the Point button to enter Point Mode. Point Mode enables you to reposition anchor points to adjust the paths of shape segments you create using the other drawing tools.

Click and drag any anchor point or slope handle to reposition it, or use the arrow keys on your keyboard to translate the selected point or handle in 1-pixel increments: the Up arrow key moves the point or point handle up a pixel, the Right arrow moves it a pixel to the right, and so forth.

You can zoom in to position the anchor points more precisely, if necessary, by clicking the Zoom button on the monitor toolbar.



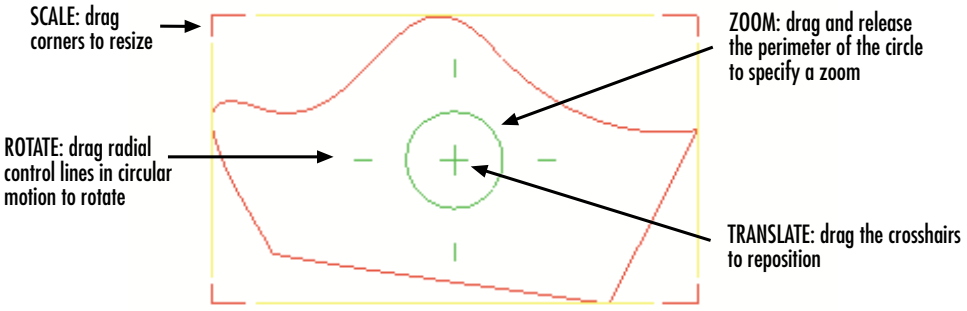
Tip:

To select multiple points, hold down the Shift key while clicking on each point you wish to select, or click and drag in the monitor to draw a bounding box around the points you wish to select.

To translate all selected points simultaneously, either hold down the Shift key while dragging one of the points, or use the arrow keys to nudge the selected points in 1-pixel increments in the direction of the arrow.

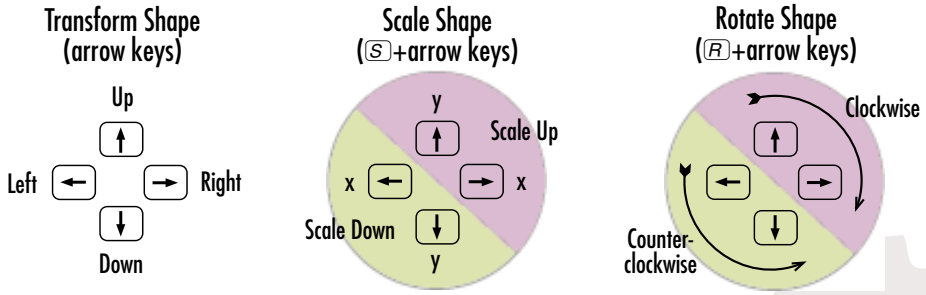
Shape Mode (Alt S)

Click the Shape button to enter Shape Mode. Shape Mode enables you to work with shapes as a whole. You can translate, scale, and rotate the selected shape in this mode. When you click on an established path or shape, Chalice displays a control device that features a boundary box and a circle-with-crosshairs tool:



Hot Keys:

You can use the arrow keys on your keyboard to translate, scale, and rotate a shape in small increments. The default mode for the arrow keys is transform. To scale or rotate the shape, hold down a modifier key while pressing the arrow keys: to scale, hold down the "s" key; to rotate, hold down the "r" key.



Add Point Mode (Alt V)

Add Point Mode enables you to add new anchor points to the shape. Click the Add Point button to enter Add Point Mode. Then click on an established path to add a new point to the shape.

Note:

Whenever you add—or delete—a point from a shape, that point is added or deleted from the shape in every frame of the sequence in which the shape appears, not just the current frame.

Edge Mode (Alt E)

The Edge Mode button only becomes accessible when the Nonuniform Edges option is selected from the Edge Type menu, which is described in the previous section on "Shape Characteristics."

In Edge Mode the points of the selected shape become hollow squares. Any of these "edge points" can be moved by dragging the mouse to adjust the edge blur for the associated spline.

Note:

The position of an edge point affects only the level and direction of blur of the associated edge; moving an edge point does not change the position of the corresponding point when you switch back to another mode.

Delete Point/Shape (Del)

Delete Point/Shape button enables you to delete selected anchor points and shapes:

- If you click the Delete Point/Shape button while an anchor point is selected, that point is deleted and a new line segment is drawn between the anchor points that bordered the deleted point.
- If you click the Delete Point/Shape button while a shape is selected, the entire shape is deleted.

Replicate Shape (Alt R)

Replicate Shape mode enables you to create additional copies of a shape. When you click the Replicate Shape button, Chalice replicates the selected shape, and creates a new, untitled entry in the Shape List. The shape is copied whether the entire shape is selected, or just a single anchor point of the shape.

Expand Points (Alt O)

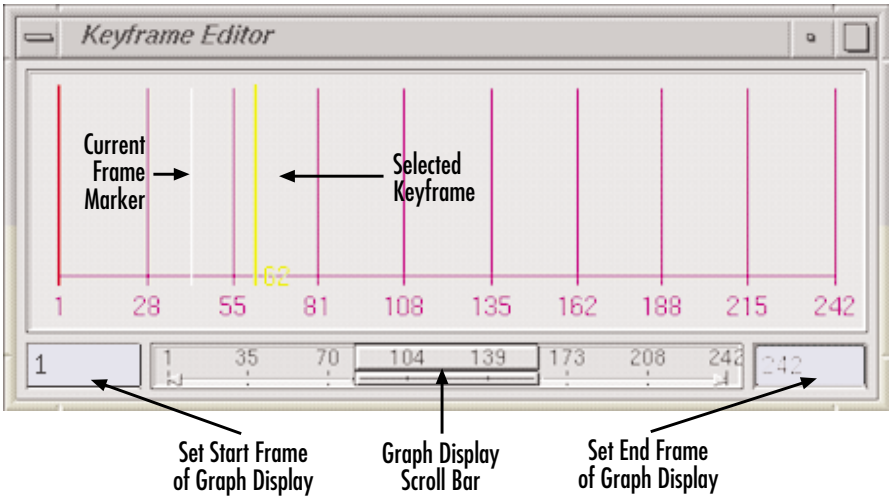
The Expand Points button enables you to **transform linear segments into splines**. While in Point Mode, click on a straight-line anchor point to select it. Then click the Expand Points button. Chalice transforms the point into a spline anchor point and any linear segment connected to the point becomes a spline.

Compress Points (Alt C)

The Compress Points button enables you to **transform splines into linear segments**. While in Point Mode, click on a spline anchor point to select it. Then click the Compress Points button. Chalice transforms the point into a straight-line anchor point and any spline connected to the point becomes a linear segment.

Keyframe Editor

You click the Keyframe Editor button to access the Rotospline Keyframe Editor.



A key frame for a shape is created automatically whenever you move to a new frame in a sequence and modify the shape in any way. The node performs a linear interpolation of the shape characteristics and associated parameter values for the frames between each key frame.

The Rotospline Keyframe Editor displays, in graph form, all key frames for a selected shape. The Keyframe Editor is the most convenient and precise way to create, move, and delete key frames in a sequence:

- To **create** a key frame, click and hold the right mouse button at any point on the graph and select "Copy" from the popup menu that appears under the cursor.
- To **move** a key frame, drag the key frame along the graph using the left mouse button.
- To **delete** a key frame, click on the key frame to select it. Then click and hold the right mouse button and select "Delete" from the popup menu that appears under the cursor.

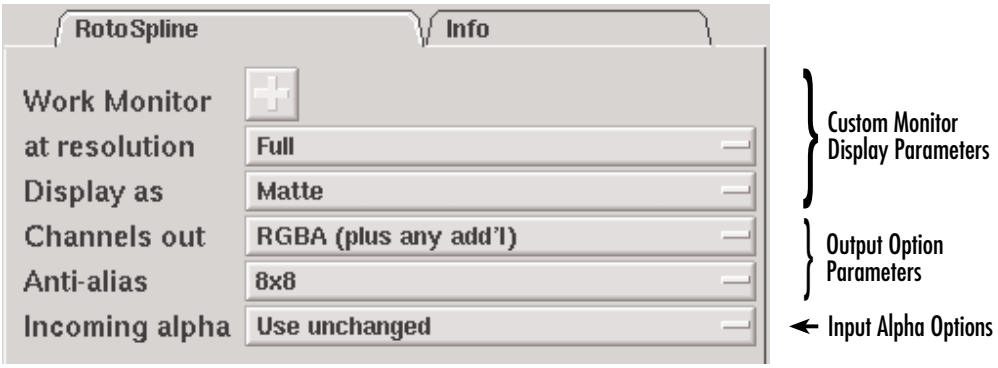
The editor displays key frame data for the currently selected shape only. To view and modify key frames for another shape, select that shape from the Shape list in the node panel. The editor graph will update to display the key frame data for the new shape.

Tip:

To obtain the smoothest effect when animating a matte shape, set as few key frames as possible. This will help to avoid the "rotoscope flicker" effect that can occur at the matte edges when the interpolation from key frame to key frame is too abrupt. The ability to move and delete key frames quickly and easily in the Rotospline Keyframe Editor should help you achieve this goal.

By default, when you open the Keyframe Editor, a graph displays the entire range of frames in the sequence. You can adjust the graph display range by keying new start and end frame values into the text entry fields. The scroll bar enables you to navigate back and forth through the graph display.

ROTOSPLINE PARAMETER TAB



Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node. The functionality of the Rotospline monitor drawing and editing tools is described in the previous section, "Using the Rotospline Work Monitor."

Display As

The Display As popup menu provides three options to control the display of matte shapes in the Alpha channel view of the work monitor. (In RGB view, matte shapes always display in wireframe mode to enable you to see the image behind them.)

The edge settings you specify for each shape in the work monitor are used for the final output. The Display As options affect only how the shapes are displayed in the work monitor:

- **Wireframe:** This setting displays all matte shapes as outlines only and speeds redraws.
- **Matte:** This setting enables you to display matte shapes with soft, or blurred, edges, providing that you have set the associated Edge parameter to a nonzero value.
- **Matte (No Edge):** This setting displays all matte edges as crisp and hard, without any edge detail, even if the Edge parameter is not set at 0. To see edge detail, select the Matte option instead.

Channels Out

The Channel parameter enables you to specify whether the node should output an RGBA image, or the Alpha channel only.

Anti-Alias

The Anti-Alias popup menu enables you to specify the level of anti-aliasing to apply, if any, to the shapes you draw in the work monitor. The higher the setting, the smoother the splines will be displayed and rendered. Of course, higher settings also take correspondingly longer to process.

Incoming Alpha

The Incoming Alpha popup menu enables you to specify the characteristics of the alpha channel data used by the Rotospline node:

- If the input image has no alpha, you can specify that the node create a black (all pixels have a value of 0) or white (all pixels have a value of 1) alpha.
- If the input image does have an alpha channel, you can use it unchanged, invert it, or clear it to black or white.

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 Chalice. (You do not require a separate Ultimatte license to utilize this functionality.)

Ultimatte is designed to optimize the bluescreen compositing process by enabling digital artists to seamlessly composite imagery while retaining shadows, fine detail, and transparent objects in the foreground image.

(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.)

As an example of the bluescreen process, a filmmaker might shoot an actor in front of a background (the bluescreen, or "backing") that has been painted entirely blue, using paints formulated specifically for this purpose (e.g., Rosco Labs Ultimatte Blue #5720, or Ultimatte Super Blue #5722, for wire removal).

Later, using the Ultimatte nodes, a digital artist will be able to prepare a processed foreground image (an image of the actor, with the background matted out) and seamlessly composite it over a background clip of a burning building, lunar landscape, or other dramatic scene.

Chalice includes five Ultimatte matte nodes, each of which is optimized to solve a particular type of bluescreen image processing problem:

Ultimatte CSC (Classic Screen Correction)

This node enables you to correct bluescreens (or red or green screens) to create a perfectly even screen area behind a foreground subject through the use of a reference clip.

Ultimatte RSC (Roto Screen Correction)

This node enables you to correct bluescreens in cases when a reference clip is not available.

Ultimatte GK (Grain Killer)

This node enables you to remove film grain (or video noise) from the backing area of a foreground shot.

Ultimatte PFG (Processed Foreground)

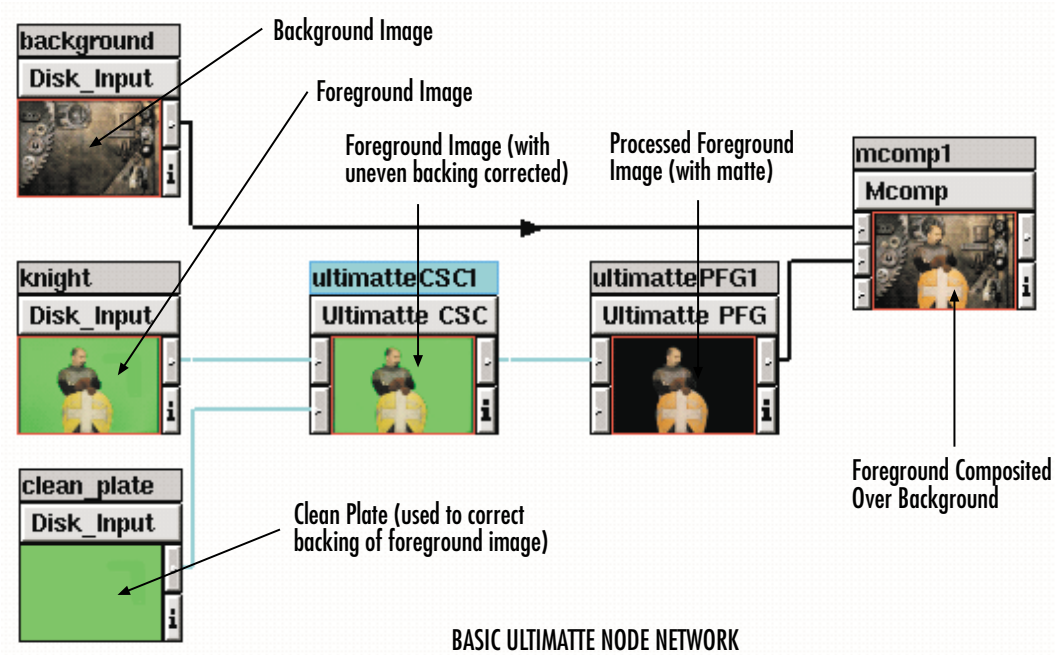
This node enables you to create a processed foreground (PFG)—an image of the foreground subject with any spill removed and the backing area suppressed to black, with the Ultimatte matte attached as the alpha channel. You may later composite this PFG image using a node such as Mcomp, Over, or Ultimatte AdvantEdge.

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.

Overview of the Ultimatte Process

The following illustration provides an overview of how the Ultimatte nodes are used in a typical compositing process. For detailed information on each node and how it is used, refer to the individual node descriptions that follow this overview. You should obtain the best results when you use the Ultimatte nodes in the sequence illustrated.



BASIC ULTIMATTE NODE NETWORK

Step A: Screen Correction

First, the foreground image—videotaped on a greenscreen stage in this case—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 surface imperfections. 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.

Note:

If you do not have a reference clip for the shot, use the Ultimatte RSC node instead of the CSC node to correct the backing.

Step B: Processed Foreground

Next, the output of the Ultimatte CSC node is connected to an Ultimatte PFG node to create a processed foreground that is ready to be composited. The Ultimatte PFG 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.

Grain Killer Option

For filmed imagery (or for a noisy video clip) you will want to use the Ultimatte GK node 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 PFG node.

Note:

If you are using the Ultimatte RSC node instead of the CSC node, do not use Ultimatte GK—the RSC node has this grain killing functionality built into it.

Step C: Composite

Finally, the processed foreground image is composited over the background image using a composite node (Mcomp, in this case).

Note:

Composite nodes such as Mcomp enable you to specify, for each input, whether the image is premultiplied, not premultiplied, or an Ultimatte input.

Be sure to select the "is Ultimatte" option for the Ultimatte input in the Mcomp node panel.

Color Control Option

When you view the composited image, in some cases you may find that the color balance of the foreground image does not match the background image perfectly. To correct this situation, you would insert an Ultimatte CC node into the network between the Ultimatte PFG and Mcomp nodes and use the Ultimatte CC controls to correct the color balance of the foreground image.

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 Chalice composite nodes. This is the true test of whether the various node parameter settings need further adjustment.

In summary, the Ultimatte nodes enable you to create perfect 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 Chalice 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 (the max of R or G) 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.

However, raising matte density too much 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 PFG 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.

How Ultimatte Mattes Are Displayed

The Ultimatte nodes operate by generating matte data from the RGB values of the image and adjusting various matte parameters, but the specific purpose and display of these mattes differ among the Ultimatte nodes.

The Ultimatte CSC, RSC, and GK nodes preprocess foreground images; they generate temporary mattes that are used only to perform the nodes' screen correction and grain killing functions. These mattes are not part of the node output image.

The Ultimatte PFG node, on the other hand, generates the matte that is used as the alpha channel in the output image. For the Ultimatte CC node, the matte controls where the color correction occurs.

Using the Ultimatte Work Monitors

All of the Ultimatte nodes feature a work monitor, which is accessed by clicking the plus icon on the node panel. You must use the work monitor that is provided for each of the Ultimatte nodes in conjunction with the node parameters.

Work monitors enable you to select the sample color that Ultimatte should use as a reference for the backing color; work monitors also display the node-specific mattes that are created by the Ultimatte CSC, RSC, and GK nodes.

The Ultimatte RSC work monitor provides rotospline tools for drawing and editing shapes (which define the subject areas).

Ultimatte Intelligence

Work monitors are used to activate Ultimatte Intelligence mode, in which you can interactively adjust various parameters by clicking specific areas in the image and letting Ultimatte automatically determine the optimum settings.

For example, if you click on an area of print-through in the matte display of an Ultimatte CSC node, Ultimatte Intelligence will automatically adjust the matte density level based on your selection. In the Ultimatte CC node, however, you click on two different colors you want to match and Ultimatte does the rest.

The Ultimatte Intelligence settings can be adjusted manually, although in most cases it isn't necessary or advised.

The work monitors feature the same toolbar options available in the regular node monitors. The following are especially useful with the Ultimatte work monitors:

- Use the Zoom arrows, or open the Magnify utility monitor, to help identify an individual pixel for selection, as when you want to sample an area of backing without clicking on a foreground detail such as a strand of hair.
- Display the Inspect panel to get RGBA values and other information for the pixel that is currently under the cursor. (The Inspect panel is displayed by selecting the Inspect option from the popup menu that you access by right-clicking anywhere in the image.)

The individual Ultimatte node descriptions that follow provide additional information about how to use the work monitor in conjunction with the parameter controls for each node.

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 perfectly 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, PFG, or CC nodes.

The Ultimatte CSC node requires two inputs:

- foreground imagery (top input)
- screen correction frame or clip (bottom input)

Clean Plate

The screen correction image, also referred to as a clean plate, is a clip of the lit backing shot in the absence of foreground objects, such as actors or props. The clean plate is used as the reference for the correction.

The screen correction input may be a single frame, or a sequence of frames that matches the number of frames in the foreground image input. The Ultimatte CSC node will use the single reference frame for comparison with each frame of the foreground image.

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. Ideally, however, the frame used for the reference image input should be an average of several frames to reduce the interference of film grain as much as possible in the reference 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.

Tip:

If you do not have a screen correction clip for a moving camera shot, use the Ultimatte RSC node instead of Ultimatte CSC.

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 (a shadow, e.g.), 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 origi-

nal foreground image, and transparent foreground objects are assigned a value that represents how they would appear over the blue color 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."

The Ultimatte CSC 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.

Then 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.

Using Classic Screen Correction

There are two steps to follow when using the Ultimatte CSC node: selecting a sample backing color and adjusting the matte.

Selecting a Sample Color

The first step to using Ultimatte CSC is to select a sample color for Ultimatte Intelligence to use as a reference in the screen correction process.

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. See also "How Ultimatte Generates Matte Data" for more information.

Open the CSC work monitor and click on a pixel in the RGB image display that represents the desired color to use for the backing. The RGB values of the selected color are displayed in the Sample Color parameter on the node panel.

The best blue to select from an uneven backing will depend on the image:

- Choosing too bright a blue can cause darker areas of unobstructed backing to be interpreted as being in shadow when the image is composited.
- Choosing too dark a blue 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 advised to select a bright, representative blue 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.

Adjusting the CSC Matte

The second step is to switch to Alpha channel display in the CSC work monitor to view a temporary matte that is generated by the node.

The CSC Alpha matte is used for the screen correction operation only, and is not part of the node output image. It represents how Ultimatte is interpreting the image: what is foreground subject and what is backing. In this matte image, foreground objects are black, and the backing area is white. The Matte Density and Black Gloss parameters enable you to control how the image data is defined for this purpose.

If there are patches of gray or white in foreground objects that should be entirely opaque, they represent pixels with RGB values that cause Ultimatte to interpret them as transparent. You can use Ultimatte Intelligence to adjust these values. Click on the brightest gray or white pixel in a foreground area and Ultimatte Intelligence will automatically adjust the RGB values of these pixels to correct this problem.

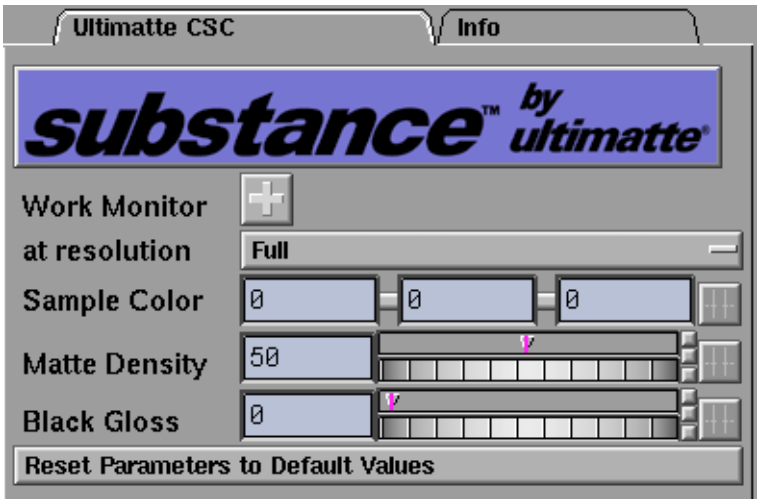
To adjust these values manually, refer to the Matte Density & Black Gloss parameter description in the next section.

Viewing the Final Result

To review the result of the screen correction, open a regular node monitor from the local node menu of the CSC node. The backing should now appear even. The final test, of course, is to view the foreground image over the background in a composite node.

Ultimatte CSC Parameter Tab

The Ultimatte CSC parameter tab enables you to select the backing color and adjust the matte.



Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node. By default, the monitor opens in RGB display. To display another channel, click the Channel Display button (cube icon) on the monitor toolbar and select the desired channel from the popup menu:

- In RGB display, this monitor enables you to select a sample color for the backing by clicking on the appropriate pixel.
- In Alpha display, this monitor displays a temporary matte image that enables you to adjust the Matte Density and Black Gloss parameters.

Sample Color

The Sample Color parameter displays the RGB values of the sample pixel you select by clicking on an area of backing in the RGB display of the work monitor. You can also key in RGB values for this parameter.

(See “Selecting a Sample Color” in the previous section of this node description for more information about how to choose the best color.)

Matte Density & Black Gloss

The Matte Density and Black Gloss parameters enable you to control the density of the matte channel data that is generated by the node as part of the screen correction process. (Be sure to select Alpha display in the work monitor to view the temporary matte.)

Both parameters function by minimizing patches of gray or white within the black foreground objects of the inverted matte. Click the brightest of these pixels and Ultimatte Intelligence will adjust the parameters as needed.

In most cases, you will want to use the parameter values set by Ultimatte Intelligence, as explained in the previous section, "Adjusting the CSC Matte." However, you can also set these parameters manually by keying in a numerical value in the range of 0–100 or by using the mouse to set the tuner to the desired value:

- To increase matte density in light or bright colored foreground objects, use Matte Density (the default value is 50).
- To increase matte density in dark, reflective foreground objects, use Black Gloss (the default value is 0).

Note:

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.

Reset Parameters to Default Values

Click this button to reset the Matte Density and Black Gloss parameters to their default values. This will not affect the Sample Color parameter values. (To reset all node parameter values to their default settings, select the "Defaults" option from the node's Presets menu.)

ULTIMATTE GK (GRAIN KILLER) NODE

The Ultimatte GK (Grain Killer) node enables you to remove film grain from the screen areas, 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" 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 to using the Ultimatte GK node is to specify a sample color of the backing area that represents the ideal color for Ultimatte Intelligence to use as a reference.

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. You can

- copy and paste the values from the Sample Color parameter entry fields.
- use the STED to export the Ultimatte CSC sample color values and then import them into Ultimatte GK.
- check Expressions and use the ch function to access the sample color data.

If your shot does not require the use of the CSC node, select a sample color by opening the GK work monitor to RGB display and clicking the appropriate pixel. See "Selecting a Sample Color" in the Ultimatte CSC node description for guidelines on selecting the best color.

Alpha Matte

The next step is preparatory to the actual grain filtering process. Switch the monitor to Alpha display, which shows foreground subjects as black and the backing as white. You can then use Ultimatte Intelligence to adjust the matte density, if necessary.

Solid foreground subject areas are not filtered by the GK node, and this process enables you to control what is defined as the subject by adjusting the Matte Density and Black Gloss parameters on the Advanced tab.

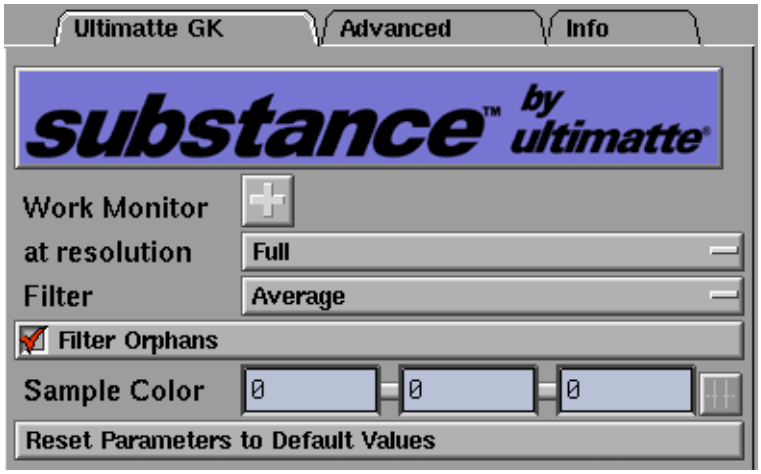
Filter Area Matte

The final step is to switch to Filter Area display in the GK monitor. This display shows foreground subjects in green, which indicates that they will never be filtered. White areas will be filtered and black areas will not; however you can change black pixels to white by using Ultimatte Intelligence.

Click on a black area that looks like grain and Ultimatte will adjust the Filter Area matte accordingly by setting the Screen Filter parameters on the Advanced tab (you can also adjust these parameters manually).

You can repeat this process as many times as necessary to achieve the best result. Be sure to review the GK node output as it will be composited. If you begin to blur important edge detail, you can increase the matte density to turn these areas green, so that they will not be filtered. Increasing the number of Filter Passes may improve the effectiveness of the grain removal, although it will also increase processing time.

Ultimatte GK Parameter Tab



Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node. By default, the monitor opens in RGB display mode. To display another channel, click the Channel Display button (cube icon) on the monitor toolbar and select the desired channel from the popup menu.

The following displays enable you to use Ultimatte Intelligence to make the indicated parameter adjustments:

- **RGB** display enables you to select a sample color for the backing by clicking on the appropriate pixel.
- **Alpha** display enables you to view a temporary matte image that is used to adjust the Matte Density and Black Gloss parameters on the Advanced tab.
- **Filter Area** display enables you to view a temporary matte image that is used to adjust the Screen Filter parameter on the Advanced tab.

Filter

The Filter parameter displays the type of filtering process used by the node, which is Average. This filter 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 by 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 by 5 square to be defined, 9 pixels (in a 3 by 3 matrix) are averaged.

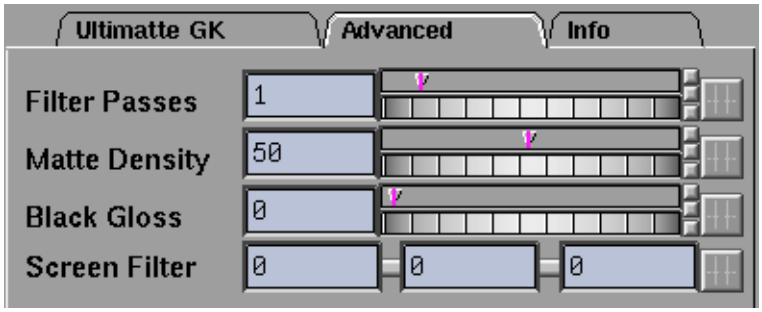
Filter Orphans
Check this box to eliminate stray single pixels, or "orphans," from the screen area. These orphans are pixels that failed to be filtered by the node.

Sample Color
The Sample Color parameter displays the RGB values of the sample pixel you select by clicking on an area of backing in the work monitor. You can also key in RGB values for this parameter.

(Refer to "Using Grain Killer" in this node description for more information about using this parameter.)

Reset Parameters to Default Values
Click this button to reset the parameters on the *Advanced tab* to their default values. This will not affect the parameters on the Ultimatte GK tab. (To reset all node parameter values to their default settings, select the "Defaults" option from the node's Presets menu.)

Advanced Parameter Tab



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 greater the number of passes, the longer the operation will take.

Matte Density & Black Gloss
The Matte Density and Black Gloss parameters enable you to control the density of the matte channel data that is generated by the node. (Select Alpha display in the work monitor to view this matte.)

Control of matte density means control of what portion of the image is defined as solid subject area. 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 Alpha 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.

In most cases, you will want to use the parameter values set by Ultimatte Intelligence, as explained in the "Using Grain Killer" section. However, you can also set these parameters manually by keying in a numerical value in the range of 0–100 or by using the mouse to set the tuner to the desired value:

- Use Matte Density to adjust light or bright colored foreground objects.
- Use Black Gloss to adjust dark, reflective foreground objects.

Screen Filter

This parameter displays the values that Ultimatte intelligence sets each time you click a pixel of grain in the Filter Area matte display. The higher the values, the greater the area of backing that will be filtered.

If these values get too high, you will begin to filter the edges of the foreground subject, resulting in a loss of detail. To ensure that these values are set properly for your shot, be sure to review the resulting image in another monitor and judge for yourself.

ULTIMATTE RSC (ROTO SCREEN CORRECTION) NODE

The Ultimatte RSC (Roto Screen Correction) node enables you to correct flaws in bluescreen backings when you do not have a reference clip (a clean plate) for the shot. For scenes with camera motion, you can use Ultimatte RSC to correct the screen area, or backing, even if you were not provided with a motion-control reference clip to match the scene frame for frame.

Note:

If you have an adequate reference clip, use the Ultimatte CSC (Classic Screen Correction) node instead of Ultimatte RSC. It will be faster and easier to produce an even backing area.

How Roto Screen Correction Works

The Ultimatte RSC node enables you to delineate the subject area at every frame by using the rotospline drawing tools provided in the RSC work monitor.

Every pixel located outside the boundaries of the shapes you draw is considered to be backing. Within each shape you have defined, Ultimatte RSC performs essentially the same screen correction operations as the Ultimatte CSC node to remove defects in the backing while retaining detail and transparency in the subject area. For more information about this process, see "How Classic Screen Correction Works."

However, accurate screen correction requires a reference frame that includes all the seams, smudges, and other flaws in the backing, so that these defects can be distinguished from transparent foreground elements such as shadows.

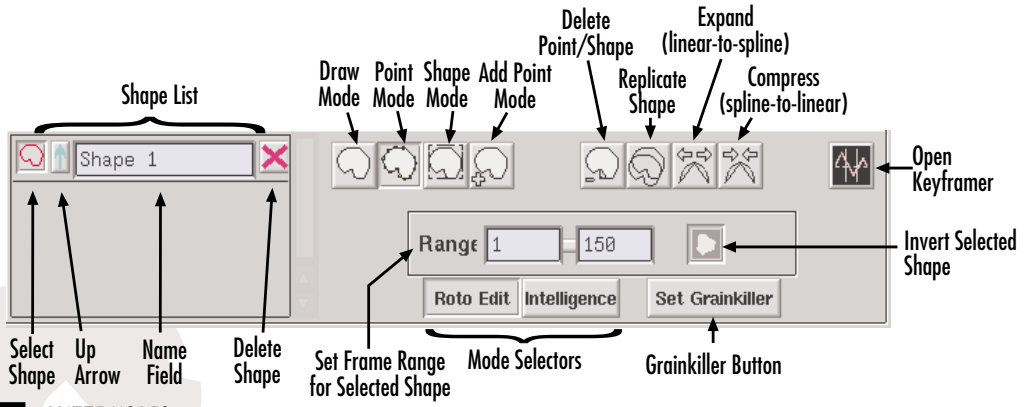
Ultimatte RSC simulates such a reference frame by extrapolating the characteristics of visible defects in the backing to the areas behind the subject. The transition area values are interpolated from the edge of the spline toward the center of the shape, using guidelines that you specify during the screen reconstruction process.

The RSC node also eliminates grain so that the output can be connected directly to an Ultimatte PFG node. You do not need to use an Ultimatte GK node after RSC, because the functionality of the Ultimatte GK node is included in Ultimatte RSC.

All of the necessary steps for roto screen correction are performed by using the RSC work monitor, in conjunction with the RSC node panel parameters.

RSC Work Monitor

The RSC work monitor provides the tools that enable you to draw and edit shapes that define the subject area. It also enables you to select a sample color to use for the backing, and to use Ultimatte Intelligence to adjust matte density and grain filtering.



Action Modes

This monitor operates in two distinct action modes: Roto Edit and Intelligence. Mode selectors on the monitor panel enable you to select the appropriate mode for the task.

Roto Edit Mode

Roto Edit mode is used for creating and editing the shapes that define the subject areas of the image. When Roto Edit mode is selected, the Shape Tools and Shape List become available for use.

The Roto Shape Tools work in the same way as the corresponding tools in the work monitor of the Rotospline node. (Refer to "Using the Rotospline Work Monitor" for a detailed description of these tools.)

You start by clicking the Draw button to create a shape, and then you use the various point manipulation tools to adjust it.

The Shape button enables you to move or rescale the shape as a whole, and the Keyframer button enables you to access the Keyframe Editor for the shape. The Range parameter is used to specify a frame range for the currently selected shape.

You can create as many shapes as you need and specify on which frames each shape will appear. In some cases it may be easier to outline the backing than to outline the subject. If so, outline the backing and then use the "Invert Roto Screen" option.

A key frame for a shape is created automatically whenever you move to a new frame in a sequence and modify the shape in any way. The node performs a linear interpolation of the shape characteristics for the frames between each key frame. You can click the Keyframer button to bring up the Keyframe Editor, which is explained in the Rotospline Node description.

Intelligence Mode

Intelligence mode is used to select a sample color and to automatically adjust the various matte and Grain Killer filter parameters.

Set Grainkiller Option

Set Grainkiller works by sampling the backing area along the perimeter of all shapes and in the area surrounding the pixel selected as the sample color to characterize the noise and then using this data to filter grain in the transition areas of the subject.

The RSC node automatically invokes Set Grainkiller when you choose a sample color (see step 2 of the next section, "Using Roto Screen Correction"), so in most cases you will not need to use this option. One exception might be if you have drastically revised the outlines of your rotospline shapes.

RSC Monitor Displays

By default, the monitor opens in RGB channel display. To display another channel, click the Channel Display button (cube icon) on the monitor toolbar and select the desired channel from the popup menu.

The Channel Display menu features several displays that are specific to the RSC node: SC Alpha, GK Alpha, and Filter Area. These three options display temporary matte image data that is used in Intelligence mode to adjust various parameters. (The RGB display is used in both Roto Edit and Intelligence modes.)

For screen correction:

- **RGB** display enables you to draw shapes to define the subject areas (when Roto Edit is selected) or to select a sample color for the backing by clicking on the appropriate pixel (when Intelligence is selected).
- **SC Alpha** display is used in conjunction with the SC Matte Density and SC Black Gloss parameters on the Advanced tab.

For grain killing:

- **GK Alpha** display is used in conjunction with the GK Matte Density and GK Black Gloss parameters on the Advanced tab.
- **Filter Area** display is used in conjunction with the Screen Filter Offset parameter on the Advanced tab.

Using Roto Screen Correction

The basic steps to using Roto Screen Correction are outlined in this section:

1. **Define the subject areas**
2. **Select a sample color**
3. **Adjust the SC Alpha matte**
4. **Adjust the screen reconstruction**
5. **Adjust the GK Alpha matte**
6. **Adjust the Filter Area**

It is important that they are performed in the order described to achieve the desired result, as each step may use parameter data that you specify in a preceding step.

Step 1: Defining the Subject Area

In the RSC work monitor, select RGB display and Roto Edit mode. Use the tools provided to outline the subject areas.

Be sure to include within the shape any shadows, wisps of hair, or other similar foreground elements that you want to appear in the final composite—all pixels outside of the shapes you draw are treated as backing.

The transition area inside of a shape is where the RSC node will perform the screen correction.

Step 2: Selecting a Sample Color

Once the subject areas have been defined, you need to specify the sample color that the node will use in the screen reconstruction, screen correction, and grain killing processes.

Note:

Be sure to define the subject area before selecting the sample color. The Grain Killer filter parameters are automatically set when the sample color is selected, and these settings are based on the sample color as well as the current shapes. Therefore, if the sample color is set before the shapes are created, the grain filtering parameters will not be set properly.

If you did select the sample color before you created the shapes, either reset the sample color, manually adjust the Filter Area, or click the “Set Grainkiller” button.

To select a sample color—the most representative blue in an unobstructed area of backing—switch to Intelligence mode and click on an appropriate pixel in the image. The RGB values of that pixel will appear in the Sample Color parameter on the RSC tab.

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. (See "How Ultimatte Generates Matte Data" for more information.)

The best blue to select from an uneven backing will depend on the image:

- Choosing too bright a blue can cause darker areas of unobstructed backing to be interpreted as being in shadow when the image is composited.
- Choosing too dark a blue 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 advised to select a bright, representative blue that is close to important foreground information, such as near a person's face. Be sure to avoid clicking on fine details, such as smoke, shadows, strands of hair, or mist.

Step 3: Adjusting the SC Alpha Matte

First you must switch the monitor display to SC Alpha. This will enable you to review the temporary matte data that is generated by the node for use in the screen correction process.

The matte represents how Ultimatte is interpreting the image: what is foreground subject and what is backing. The SC Alpha display enables you to use Intelligence, if necessary, to control how the image data is defined for this purpose.

In this matte image, foreground objects look black, and the backing area looks white. If there are patches of gray or white in foreground objects that should be entirely opaque, they represent pixels with RGB values that cause Ultimatte to interpret them as transparent.

Click on the brightest gray or white pixel in a foreground area and Ultimatte Intelligence will automatically adjust the RGB values of these pixels to correct this problem.

SC Alpha Intelligence can be repeated multiple times, if necessary. To adjust these values manually, refer to the "Matte Density & Black Gloss Parameters" description in the Parameters section.

Step 4: Adjusting the Screen Reconstruction

At this stage you will want to evaluate the how the screen area has been reconstructed. Check "Show Screen Reconstruction" on the RSC tab and switch to RGB display.

The Screen Reconstruction display represents the reference frame data that will be used (in lieu of an actual clean plate) to create a corrected foreground image for further processing in other nodes.

The goal, therefore, is to create a representation of what the screen would like if the foreground subject were not present, not to create a homogeneous field of blue.

Note:

The screen only needs to be reconstructed in the transition area which spans the narrow area between the solid parts of the foreground subject and the edge of the shape. When evaluating the reconstruction, you can ignore areas that you are sure fall completely within the solid part of the foreground subject.

If the reconstructed screen areas need adjustment, you can use the tracking parameters on the Reconstruct tab to control how edges, seams, and other discontinuities are extended into transition areas. (The parameters on the Reconstruct tab are explained in the Parameters section later in this node description.)

Tip:

If a dark or discolored smear appears to stain the screen reconstruction, it is usually an indication that the rotospline has accidentally touched the edge of a foreground subject (and the reconstruction process has treated it like a discontinuity and extended it into the transition area).

Return to Roto Edit mode temporarily and move the spline so that it no longer touches the foreground edge detail.

Step 5: Adjusting the GK Alpha Matte

Once you are satisfied with the screen reconstruction, you should turn off "Show Screen Reconstruction" and switch to GK Alpha display to view the temporary matte.

The GK Alpha matte is provided to enable you to use Ultimatte Intelligence, if necessary, to control what portion of the image is defined as solid subject area. This is a preparatory step to grain killing, because the areas defined as totally opaque foreground objects will not be grain-filtered. (Grain killing with Intelligence is performed in the Filter Area display.)

For example, if you boost the density values in GK Alpha 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.

The GK Alpha display shows the backing as white and the subject areas as black. If there are patches of gray or white in foreground objects that should be entirely opaque, click on the brightest gray or white pixel and Ultimatte Intelligence will automatically adjust the GK Matte Density or GK Black Gloss parameters.

These parameters, located on the Advanced tab, can also be adjusted manually. (The "Object Area (Green)" parameter on the RSC tab can also be used to manually adjust matte density.)

Step 6: Adjusting the Filter Area

After adjusting the GK Alpha matte, switch to Filter Area display to review how the transition areas inside of shapes will be grain-filtered.

This step is key to a good composite using RSC. If you leave excess grain or noise in the transition areas, you may be able to see the outline of the shape because it will stand out from the homogenous blue of the backing.

The Filter Area display shows the foreground subject in green (green areas are never filtered). The remainder of the area inside a shape is the transition area. White pixels within the transition area will be filtered; black pixels will not.

You can increase the area to be filtered by using Ultimatte Intelligence. If you click on an area of black that looks like grain or noise, Intelligence will adjust the Screen Filter Offset parameters (located in the Advanced tab).

All areas of black that meet similar criteria as the selected pixel will turn white and thus be subject to filtering. You can continue to click on black specks in the transition area until you are satisfied.

If you go too far, however, areas that should remain black will turn white, such as strands of hair and other edge detail. If these areas remain white, they will be filtered and thus blurred. Turn down the Screen Filter Offsets until these areas turn black.

Note:

As you adjust the filter area, be sure to view the RSC node output image by bringing up a regular Chalice monitor from the RSC node menu. You can compare the result against the source image by performing an A/B wipe in the monitor. You should also review the result composited over the background.

This is an essential part of the filter area adjustment process. If there is a problem, as when important edge detail has been accidentally filtered, this is how you will find out. The goal is not to make the Filter Area display look good, but to make the final composite look seamless.

If you are having trouble eliminating grain without also blurring important edge detail, try increasing the foreground subject area (green) to encompass more of the edge detail. (The green area is not affected by Screen Filter Offset adjustments.)

You can switch to GK Alpha display and use Intelligence to increase the green subject area, or you can adjust the GK matte controls manually. Then you can go back to the Filter Area display and try again to remove the grain.

You can also increase the number of filter passes to improve the grain removal. The Filter Passes parameter, located on the Advanced tab, is adjusted manually. However, the more you increase the number of passes, the longer the node will take to process.

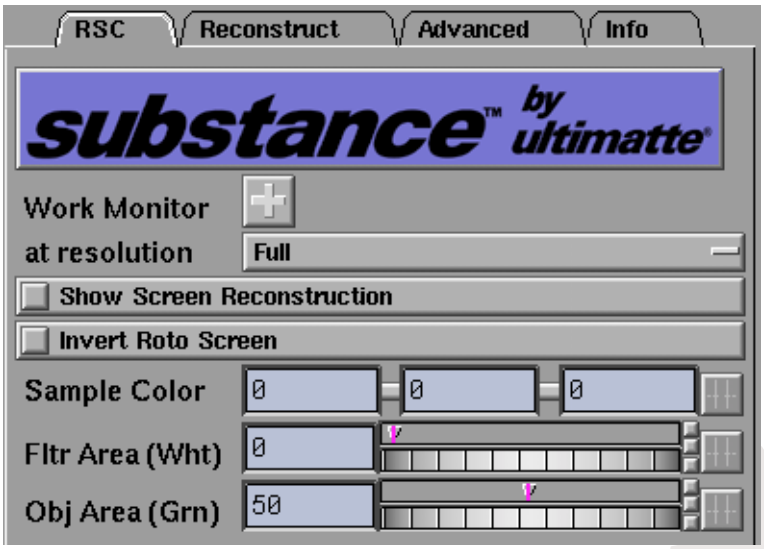
Tip:

If necessary, you can turn off filtering temporarily while you are adjusting the filter area by resetting the number of passes to zero. (The black and white areas of the display will still update properly.)

But you must remember to reset the Filter Passes parameter to a nonzero value when you are done, or the output will not be filtered!

RSC Parameter Tab

The RSC parameter tab enables you to access the work monitor and other parameters.



Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node. For more information, refer to the "RSC Work Monitor" description in the previous section.

Show Screen Reconstruction

Once you have defined the subject areas and chosen a sample color, check this box to display a reconstruction of the current reference frame. If the reconstruction of the backing under any of the subject areas needs adjustment, use the parameters provided in the Reconstruct parameter tab.

Invert Roto Screen

Normally the area inside any shape that you draw is considered foreground and all areas outside the shapes are considered backing. Check this box to invert this definition, such that the areas outside the shapes become foreground and the areas inside the shapes become backing.

Sample Color

The Sample Color parameter displays the RGB values of the sample pixel you select by clicking on an area of backing in the RGB display of the work monitor.

Intelligence mode must be selected (not Roto Edit mode) to select a sample color. The mode selector buttons are located in the RSC work monitor panel. You can also key in RGB values for this parameter.

Filter Area & Object Area Parameters

These two parameters can be used to manually adjust the grain filtering process, if necessary. They should be used while viewing the Filter Area display in the RSC work monitor. (For more information about this process, refer to Step 6: Adjusting the Filter Area.")

Note:

Always start by using Ultimatte Intelligence to set the Screen Filter Offset parameter (located on the Advanced tab). Then, if necessary, the "Filter Area (White)" parameter can be used to add small amounts of white.

Filter Area (White)

This parameter adjusts the transition area of the matte used for grain filtering. Increasing the value (the default is 0) will increase the white area of the Filter Area display; that is, the area that will be grain-filtered.

Object Area (Green)

This parameter increases or decreases the density of the matte used for grain filtering. Increasing the value effectively expands the green foreground area in the Filter Area display. (A corresponding expansion will also appear in the black area of the GK Alpha display.)

Reconstruct Parameter Tab

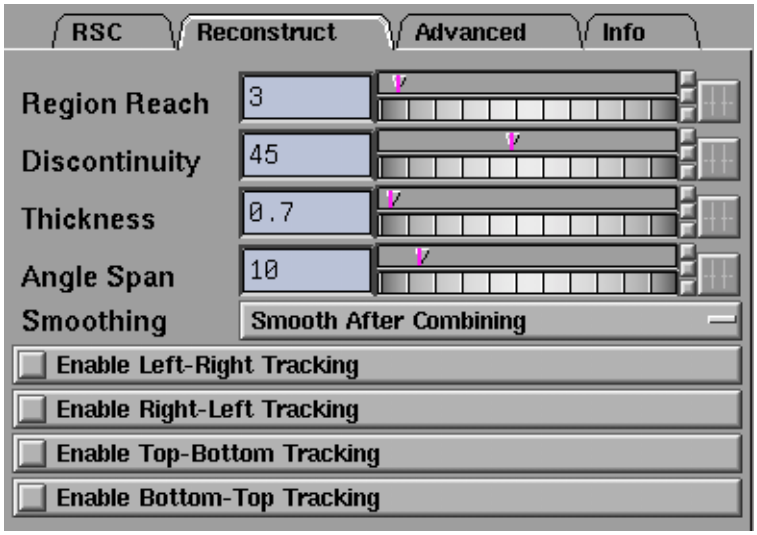
The Reconstruct parameter tab enables you to finetune the screen reconstruction process as needed for a particular scene by using directional tracking.

Directional tracking, in this context, is the process of identifying discontinuities (flaws in the backing area) and extrapolating them into the reconstructed screen area behind the subject. When directional tracking is enabled, the RSC node identifies those discontinuities that follow the same vector as the selected tracking direction and extends them into the screen area. To see the effect, you must turn on Show Screen Reconstruction (the checkbox is located on the RSC tab) and select RGB display in the work monitor.

You can specify how discontinuities are extrapolated by using the four parameters that modify directional tracking: Region Reach, Discontinuity, Thickness, and Angle Span.

Tip:

Select the "No Smoothing" option from the Smoothing menu while adjusting these parameters. This will make it easier to see and follow the extrapolations into the shape area as you change the parameter settings.



"Enable Tracking" Parameters

There are four directional tracking options available:

- Enable Left-Right Tracking
- Enable Right-Left Tracking
- Enable Top-Bottom Tracking
- Enable Bottom-Top Tracking

You can check as many options as needed for adequate screen reconstruction of the shot. For example, if a horizontal seam in the bluescreen is visible coming in from the left side of the image and going behind the subject, check Enable Left-Right Tracking.

Region Reach

The Region Reach parameter modifies how far into the subject area (inside the splines) the discontinuity will be extrapolated. The range is 0–100, which represents a percentage of the total width of the image.

Discontinuity

The Discontinuity parameter is a threshold that limits what features in the backing are defined as discontinuities that should be extrapolated into the screen reconstruction area.

The RSC node evaluates differences in pixel brightness to identify discontinuities, and only those discontinuities with values that exceed the specified threshold of this parameter are extrapolated.

The value is a function of pixel brightness and the range of 0–100 represents normalized brightness value.

Thickness

The Thickness parameter modifies the width, or spread about the vector, of the discontinuity as it is extended into the subject area. The range is 0–100, which represents a percentage of the total width of the image.

Angle Span (Degrees)

The Angle Span parameter enables you to control whether those discontinuities that deviate from the selected tracking direction will still be interpolated into the subject area.

For example, if Left-Right Tracking is selected and the Angle Span parameter is set at 10 (which is the default), the RSC node will interpolate not only those discontinuities that follow the X axis of the image exactly, but also any discontinuities that describe a slight angle to the horizontal—up to 10 degrees in either direction.

The parameter range is 0–90. A value of 0 specifies no deviation from the tracking angle, while a value of 90 specifies that a discontinuity can deviate up to 45 degrees in either direction (a total span of 90 degrees).

Smoothing

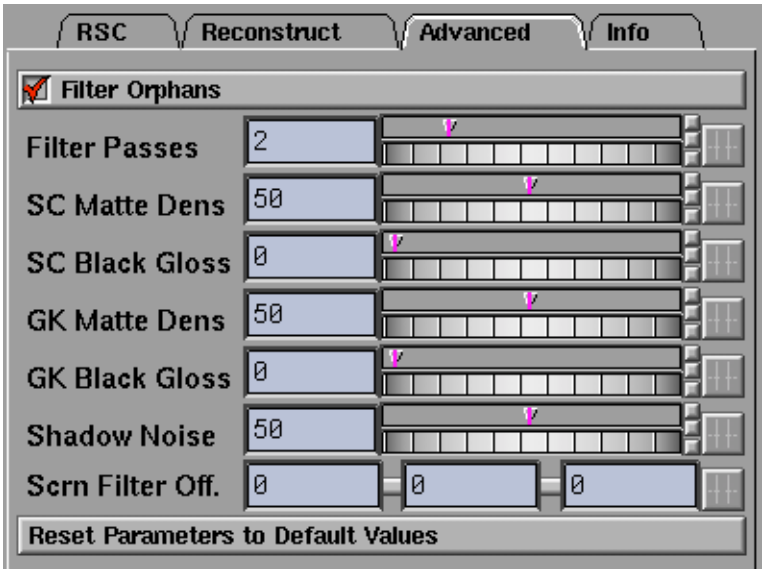
Smoothing refers to the process of filling in the shape areas during screen reconstruction; therefore, you should always use the default setting "Smooth After Combining" for the final output.

However, you will most likely want to turn off smoothing (select "No Smoothing" from the popup menu) while adjusting the Reconstruction parameters.

Advanced Parameter Tab

The Advanced tab provides parameters for making fine adjustments to the temporary matte data that the RSC node creates as part of the screen filtering and grain filtering processes.

In most cases you will want to use Ultimatte Intelligence to adjust these parameters, as described in "Using Roto Screen Correction." However, these parameters can be adjusted manually.



Filter Orphans

This option, which is on by default, eliminates stray single pixels, or "orphans," from the transition area. Orphans are pixels that were missed in the grain filtering process.

Filter Passes

The Filter Passes parameter enables you to specify the number of times the image data will pass through the grain filter. The default value is 2.

If necessary, you can increase this value; however, the greater the number of passes, the longer the process will take. (You can set this parameter to 0 to turn off filtering temporarily while adjusting the Filter Area.)

Matte Density & Black Gloss Parameters

The **SC Matte Density** and **SC Black Gloss** parameters are used in conjunction with the SC Alpha monitor display to enable you to control the density of the matte channel data that is generated by the node as part of the screen correction process. (Refer to Step 3: Adjusting the SC Alpha Matte of the previous section on Using Roto Screen Correction.)

The **GK Matte Density** and **GK Black Gloss** parameters are used in conjunction with the GK Alpha monitor display to enable you to control the density of the matte channel data that is generated by the node as part of the grain killing process. (Refer to Step 5: Adjusting the GK Alpha Matte of the previous section on Using Roto Screen Correction.)

All of these parameters function by minimizing patches of gray or white within the black foreground objects of the matte.

In most cases, you will want to use the parameter values set by Ultimatte Intelligence. However, you can also set these parameters manually by keying in a numerical value in the range of 0–100 or by using the mouse to set the tuner to the desired value. The associated monitor display will update to show you the result of any manual adjustments.

Try adjusting Matte Density for bright objects, and Black Gloss for dark, or black glossy, objects.

Shadow Noise

The Shadow Noise parameter is used to adjust the SC Alpha matte data to reduce noise in shadows and glare areas.

Screen Filter Offset

This parameter is used in conjunction with Filter Area display to remove grain from transition areas. Each time you click a pixel of grain in the Filter Area display, Ultimatte Intelligence will adjust these values accordingly. If these values get too high, you will begin to filter the edges of the foreground subject, resulting in a loss of detail.

This process is described in Step 6: Adjusting the Filter Area, in the previous section on Using Roto Screen Correction.

Reset Parameters to Default Values

Click this button to reset all of the parameters on the Advanced tab to their default values.

> **Note:**
>
> It will also reset the two Screen Filter parameters (Filter Area and Object Area) on the RSC tab to their default values.

ULTIMATTE PFG (PROCESSED FOREGROUND) NODE

The Ultimatte PFG (Processed Foreground) node enables you to create and output a clean, processed foreground image: that is, an image of the foreground subject with all 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 Chalice composite nodes.

Tip:

Before you use the Ultimatte PFG 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 with the Ultimatte PFG node.

Matte density values can be adjusted automatically or manually in the Ultimatte PFG node 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. See "Adjusting Matte Density" for a description of how Ultimatte performs this adjustment. The Ultimatte PFG node also provides flare controls that you can use to remove discoloration caused by spill from foreground objects.

The Ultimatte PFG node accepts two inputs: the three-channel (RGB) foreground image and an optional one-channel garbage matte. The garbage matte can be used to extend the backing in cases where the bluescreen was not large enough to fill the frame or to hide lighting equipment and other unwanted elements in the shot.

You can specify whether the node will output only the matte channel, only the processed foreground, or all four (RGBA) channels.

USING ULTIMATTE PFG

Step 1: Pick a Backing Color

Start by opening the PFG work monitor and clicking on a pixel in the backing area. Ultimatte Intelligence will use the backing color values as a reference for setting color logic and generating matte data. See "How Ultimatte Generates Matte Data" 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 PFG. Refer to the introduction, "The Ultimatte Nodes," 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 work monitor to view the matte channel. Unlike the Ultimatte CSC and GK nodes, which display an inverted matte, the Ultimatte PFG node displays a matte channel where transparent (backing) areas appear black and opaque (foreground) areas appear white. 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. Click on one of the gray or black pixels in the foreground and then click the Intelligence: Fix White Matte but-

ton. Ultimatte will adjust certain Matte Control parameters for you. Alternatively, you can adjust matte values manually by resetting the parameters on the Matte Controls tab.

The Intelligence: Fix Black Matte button is used the same way when you need to correction imperfections in the backing areas.

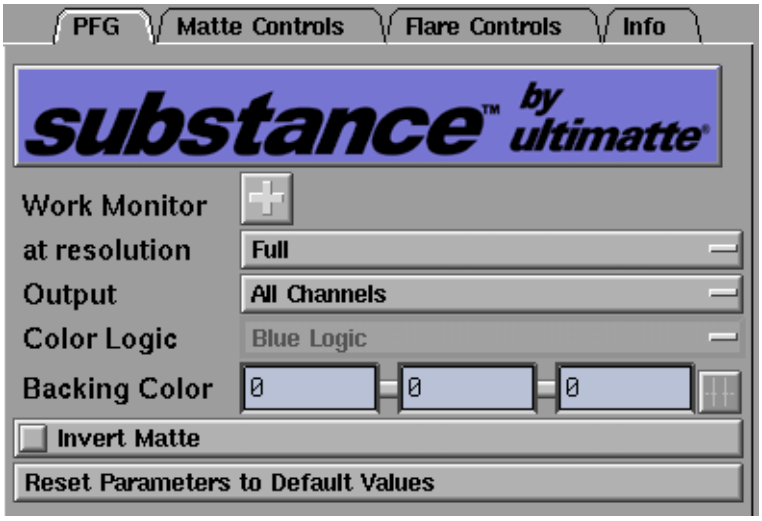
Step 4: Correct Discoloration

After you are satisfied with the matte density values, you can use the Flare Controls to correct discoloration, especially of magentas, yellows, and browns, that can be caused by spills and flares. Using an A/B wipe, 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 tab descriptions in the Parameters section. Adjusting any of these parameters too dramatically can adversely affect the realism of your composite.

PFG Parameter Tab



Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node. In RGB display mode, you can select a pixel for the Backing Color parameter by clicking on it. In Alpha display mode, you can click on areas that represent holes in the matte that you want Ultimatte Intelligence to fix for you.

Output

The Output popup menu enables you to specify which channels the node will output:

- All Channels (RGBA) — the default
- Matte Channel only (Alpha)
- Processed Foreground only (RGB)

Color Logic

The Color Logic popup menu displays the processing logic selected by Ultimatte PFG, based on the Backing Color parameter value: Blue Logic (for an image shot against a bluescreen); Green Logic (for a green screen shot), or Red Logic (for red screen).

Backing Color

The Backing Color parameter displays the RGB values of the backing color that you have selected by clicking on an appropriate pixel in the PFG work monitor. You can also key in RGB values for this parameter.

Invert Matte (Checkbox)

Click this checkbox to invert matte values, so that transparent (black) areas become opaque (white) and vice versa.

Reset Parameters to Default Values

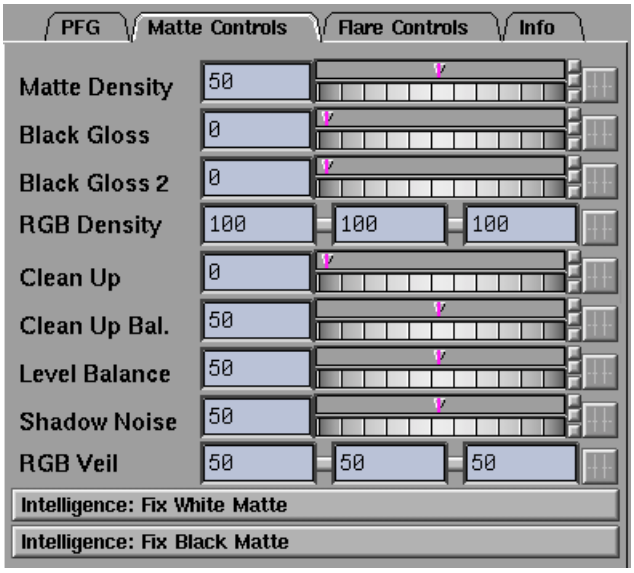
Click this button to reset all of the parameters on the Matte Controls tab and Flare Controls tab to their default values, whether these parameters have been adjusted manually or by clicking the "Intelligence: Fix" buttons on the Matte Controls tab.

This will not affect the Backing Color parameter values. (To reset all node parameter values to their default settings, select the "Defaults" option from the node's Presets menu.)

Matte Controls Parameter Tab

The Matte Controls parameter tab enables you to apply programmed intelligence to adjust matte levels (by using the "Intelligence: Fix" buttons) and to manually assign parameter values. Be sure to display the Alpha channel in the work monitor before using the Matte Controls.

You set each parameter by keying a value in the range of 0–100 into the associated data entry field or by using the mouse to set the tuner, if available, to the desired value.



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.

Start by using Ultimatte Intelligence (the "Fix White Matte" button). If further adjustment is needed, adjust Matte Density and Black Gloss interactively: 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.).

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.)

The Matte Density default value is 50.

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.

The default value is 0. Setting this parameter too high may alter the color of objects in the foreground.

RGB Density

The RGB Density parameter enables 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 RGB values 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. The default value is 100 for all three component colors.

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, reflections, etc. and ruin the realism of the composite.

Start by using Ultimatte Intelligence (the "Fix Black Matte" button) to set the values for this parameter and Clean Up Balance. If necessary, you can make further adjustments manually. Use Clean Up and Clean Up Balance interactively, and set Clean Up as low as possible; a compromise setting can usually be found. The Clean Up default value is 0.

Clean Up Balance

The Clean Up Balance parameter enables you to assign the degree of influence that the Clean Up parameter has on the foreground relative to the backing area of the image.

The Clean Up Balance 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, when Ultimatte suppresses the backing color to black a residue of the backing color may remain in some areas. This veil of color can discolor the background image in the composite.

The RGB Veil parameter enables you 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.

The default value is 50 for all three component colors.

Note:

If possible, you should use the Ultimatte CSC node to even the backing area before the image is input to the PFG node.

"Intelligence: Fix" Buttons

When you click one of the Intelligence: Fix buttons, Ultimatte Intelligence will automatically adjust the appropriate Matte Controls parameters.

This adjustment is based on the sample pixel value that you select by clicking on an imperfection displayed in the Alpha channel of the work monitor.

Fix White Matte (Fix Subject)

Intelligence: Fix White Matte is used to eliminate print-through in foreground objects. Click on a gray or black pixel in a foreground object and then click the Fix White Matte button. Ultimatte will adjust the Matte Density and/or Black Gloss parameter values based on the value of the pixel you selected.

Fix Black Matte (Fix Backing)

Intelligence: Fix Black Matte is used to eliminate imperfections in the backing area. Click on a gray or white pixel in a backing area and then click the Fix Black Matte button. Ultimatte will adjust the Clean Up parameter values based on the value of the pixel you selected.

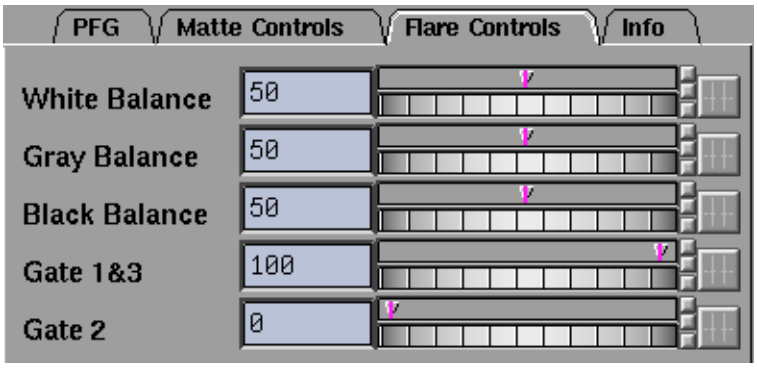
Flare Controls Parameter Tab

The Flare Controls parameter tab enables you 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 blue backing as an example (this suppression works similarly for green and red backings), 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.

Each of the Flare Control parameters can be set by keying in a numerical value in the range of 0–100 or by using the mouse to set the tuner to the desired value.

Balance Parameters

The Balance parameters enable you to make foreground colors warmer or cooler by overriding the spill/flare suppression logic which 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 type of logic (blue, green, or red) assigned to the Color Logic parameter on the PFG parameter tab. The default value for Gate 1 & 3 is 100; the default value for Gate 2 is 0.

Gate 1 & 3

The following table illustrates the spill suppression effect of the Gate 1&3 parameter on various color families:

	Blue Logic	**Green Logic**	**Red Logic**
	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

For example, if Blue Logic is specified for the Color Logic parameter, compare the original foreground and the composite. Look to see if any cyan objects have turned blue, or green objects cyan. If so, you can probably use the Gate 1&3 parameter to remove the excess blue.

Gate 2

The following table illustrates the spill suppression effect of the Gate 2 parameter on various color families:

	Blue Logic	**Green Logic**	**Red Logic**
	Magenta-Red	*Yellow-Red*	*Magenta-Blue*
0	Magenta turns Red	Yellow turns Red	Magenta turns Blue
100	Magenta unaffected	Yellow unaffected	Magenta unaffected

For example, if Blue Logic is specified for the Color Logic parameter, compare the original foreground 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.

Note:

The default value for Gate 2 depends on the color logic. For Green Logic, the Gate 2 parameter will be automatically be set to a value of 50. This will prevent hair and skin tones from losing their green component and taking on a purple or red cast.

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 PFG (Processed Foreground) node to match the background image over which it will be composited.

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 PFG 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.

USING COLOR CONTROL

Start by setting up an A/B wipe between the input image and a reference image in the Ultimatte CC work monitor. Refer to "Performing A/B Wipes" in chapter 8, "Monitors and Flipbooks," if you need instructions.

Click on a pixel in the A (input) image to select the sample color, and in the B (reference) image to select the reference color:

- To match highlights, click the pixel with the highest value in each image (the Inspect panel is helpful for this).
- To match flesh tones, on the other hand, you could click on a pixel in the same part of the face in each image.

Then click the appropriate button (Match Image Levels, Match Image Gamma, or Match Image Blacks) on the Ultimatte CC parameter tab and Ultimatte Intelligence will automatically match highlights, midrange tones, or blacks.

You can match more than one color attribute for an image. For example, you can match brightness levels using Match Image Levels and then select new sample and reference pixels to match flesh tones using Match Image Gamma.

You can use the Controls parameter tab to further adjust these settings. In fact, you can use the Controls parameters to manually adjust various color attributes of the input image without using a reference image.

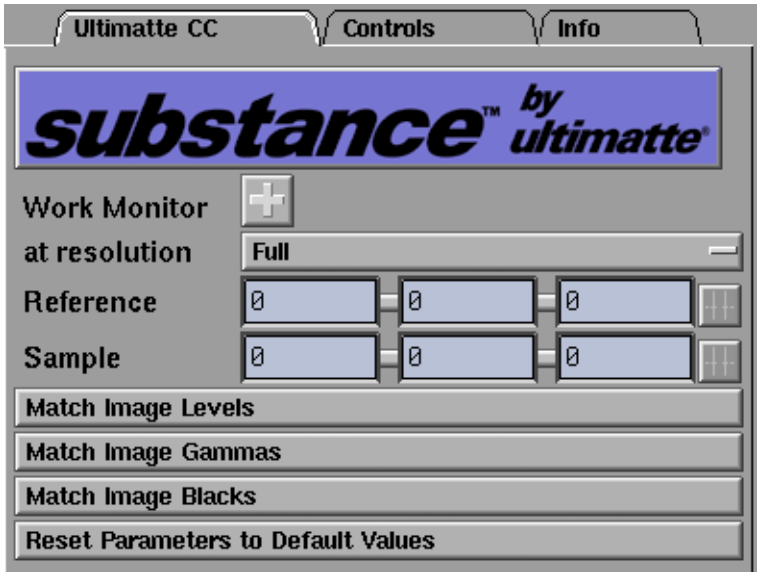
Tip:

For the best result, finetune your color adjustments based on viewing the final composited image, rather than the foreground image alone.

The Ultimatte CC accepts one input. (The reference image, if used, is accessed via an A/B wipe in the Ultimatte CC work monitor.)

Ultimatte CC Parameter Tab

The Ultimatte CC parameter tab enables you to use Ultimatte Intelligence to automatically adjust color parameters of the input image to match a reference image.



Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor, which is used to set up an A/B wipe between the input (sample) image and the reference image, as well as to select the sample and reference colors.

You click on an appropriate pixel in both the A and B images of the wipe and the corresponding RGB values are displayed in the Sample and Reference parameters of the Ultimatte CC tab.

Note:

Both images used for the A/B wipe must be cooked at the same resolution. For example, if you open the Ultimatte CC work monitor at Full resolution, you must select Full from the local node menu of the node you wish to use for the reference image when you cook it into a buffer.

Reference & Sample Parameters

The Reference parameter displays the RGB values of the selected pixel of the reference image (the image that contains the color values you want to match).

The Sample parameter displays the RGB values of the selected pixel of the input image, which you want to correct to match the reference image.

"Match Image" Buttons

When you click any of these buttons, Ultimatte Intelligence automatically makes the appropriate adjustments to the RGB values of the input image. Sample and Reference parameter values must be selected before invoking a Match Image operation.

Each Match Image button affects a different set of parameters, which can be viewed and adjusted manually on the Controls parameter tab.

Match Image Levels

Use Match Image Levels to match highlights. This is effectively a brightness control; pure blacks are not affected at all. (This adjustment is reflected in the Red Level, Green Level, and Blue Level parameters on the Control tab.)

Match Image Gamma

Use Match Image Gamma to match midrange tones, as when you need to match flesh tones or other like colors. Pure blacks and pure whites are not affected. (This adjustment is reflected in the Red Gamma, Green Gamma, and Blue Gamma parameters on the Control tab.)

Match Image Blacks

Use Match Image Blacks to match black levels. Pure whites are not affected. (This adjustment is reflected in the Red Black, Green Black, and Blue Black parameters on the Control tab.)

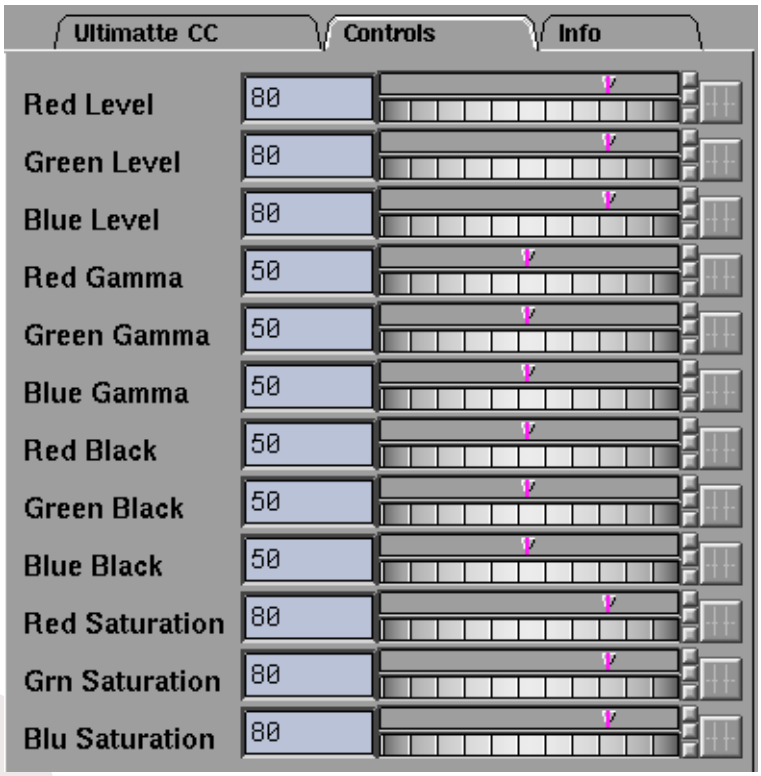
Reset Parameters to Default Values

Click this button to reset all of the Control tab parameters to their default values, whether these parameters have been adjusted manually or by clicking one or more of the Match Image buttons on the Ultimatte CC tab.

This will not affect the Sample or Reference parameter values. (To reset all node parameter values to their default settings, select the "Defaults" option from the node's Presets menu.)

Controls Parameter Tab

The Controls parameter tab enables you to manually adjust RGB values for Level, Gamma, Black, and Saturation parameters to facilitate or finetune the color correction process. (Any automatic adjustments to Levels, Gamma, or Blacks made by Ultimatte Intelligence will also be reflected here.)



You set each parameter by keying in a numerical value between 0 and 100 or by using the mouse to set the tuner to the desired value.

Level: Red, Green, Blue

These three parameters are used to control highlights. Use Level to adjust brightness; pure blacks are unaffected. The level parameters are effectively gain controls on the color components because they provide a multiplier for each pixel value. The default value for all three parameters is 80.

Gamma: Red, Green, Blue

These three parameters are used to control 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 all three parameters is 50.

Black: Red, Green, Blue

These three parameters are used to control black levels; pure whites are unaffected. Impure whites (less than 100 percent saturated) can be noticeably affected, however, if all three Black parameters are not adjusted equally. If the Black parameters are set below 50 (which is their default value), the darkest color components in the image will clip at 0.

Saturation: Red, Green, Blue

These three parameters are used to control saturation by proportionally replacing each color component with its luminance value (R * 0.299; G * 0.587; B * 0.114). If you set all three Saturation parameters to zero, the result will be a grayscale image. The default value for all three parameters is 80.

17 MOVEMENT NODES

The Movers menu contains nodes used for spatial transformations such as cropping, scaling, and rotating: **Crop**, **Scale**, **Rotate Flip**, and **Transform**. The Transform node is the most versatile mover—it translates, rotates, and scales, with or without motion blur and anti-aliasing.

Three nodes, **Corner Pin**, **Match Move**, and **Stabilize**, all use tracking data to control the transformations over time.

The Crop and Rotate Flip nodes accept one input. The rest all take an optional second input to provide a background reference image and/or to control the size of the output.

The Movers menu contains the following nodes:

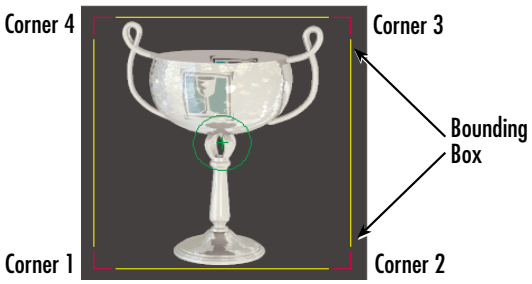
Corner Pin Node ..... p. 17.2
Crop Node ..... p. 17.6
Rotate Flip Node ..... p. 17.8
Scale Node ..... p. 17.9
Transform Node ..... p. 17.12
Match Move and Stabilize Nodes ..... p. 17.17

CORNER PIN NODE

The Corner Pin node is used to both distort and "fly" one image over another through a series of frames.

The Corner Pin node accepts one or two inputs: the foreground image and an optional background image used in the work monitor to establish size and provide a visual reference for the motion.

The corner positions can be controlled interactively in the work monitor, which features an adjustable bounding box for manipulating the foreground image interactively.



The interactive controls in the work monitor are tied to the four Corner parameters on the node panel and their range of motion can be constrained, if you so specify, by the Shape Limits menu.

All four corners can be animated, and most commonly the Corner Pin node uses position data from a Track node to control the corner values.

Using Track Data in Conjunction with the Corner Pin Node

You can use expressions to access data generated in a Track Node to control the position of each corner. Enter the following expressions in the *x* and *y* parameter fields of the Corner parameters:

Corner 1:	track("track1", XV, 1, $F)	track ("track1", YV, 1, $F)
Corner 2:	track("track1", XV, 2, $F)	track ("track1", YV, 2, $F)
Corner 3:	track("track1", XV, 3, $F)	track ("track1", YV, 3, $F)
Corner 4:	track("track1", XV, 4, $F)	track ("track1", YV, 4, $F)

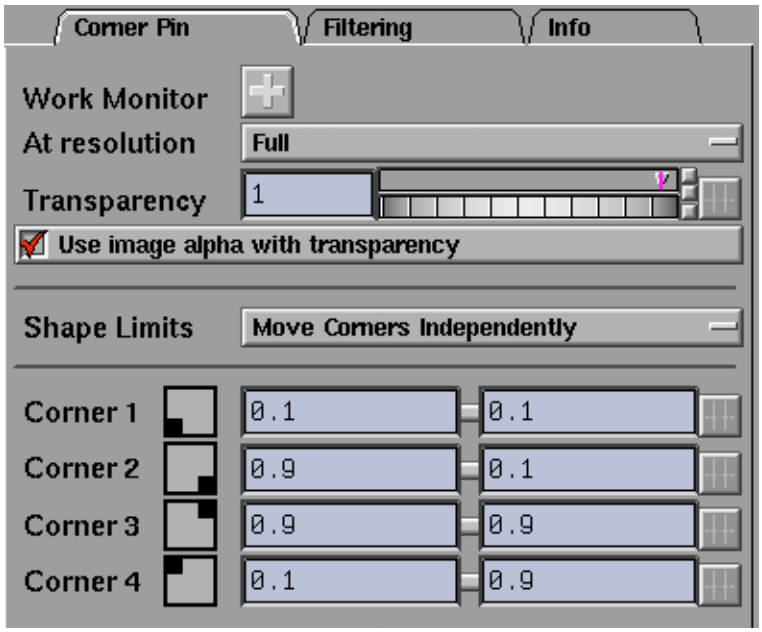
These expressions stipulate

- that the data should be accessed from a node called "track1." (You should use the exact name, in quotes, of the actual Track node that will be accessed.)
- whether the *x* or *y* value (XV or YV) should be used.
- which track point to use (1, 2, 3, or 4).
- that the data for the current frame number ($F) should be used.

Tip:

When placing the four track points in the Track node, position them so that their numbers correspond to the Corner parameter numbers in the Corner Pin node: place Track Point 1 in the lower-left corner of the object to be tracked, since the corresponding corner in the Corner Pin node is Corner 1; place Track Point 2 in the lower-right corner (Corner 2), and so on. This will ensure that you reference the correct point numbers in the above expressions.

Corner Pin Parameter Tab



Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node.

Transparency

The Corner Pin node enables you to input a background image for reference. The Transparency parameter is used to specify the level of transparency of the top input over the bottom input as it is viewed in a temporary composite in the Corner Pin work monitor. You set the transparency value of the top input by keying in a numerical value between 0 (transparent) and 1 (opaque) or by using the mouse to set the tuner to the desired value.

Use Image Alpha with Transparency

When this option is checked, the alpha channel value of each pixel of the primary input is multiplied by the Transparency parameter value to govern the overall transparency of the temporary composite image in the work monitor. (This is assuming that you have used the bottom input connector to the Corner Pin node to input a background image for reference.)

Shape Limits Menu

The Shape Limits parameter enables you to set certain constraints on the movement of the image being pinned:

Move Corners Independently

This mode, which is the default, is unconstrained. You can drag the corners of your image freely to move, scale, rotate, or distort their shape.

Right Angles with Rotation

In this mode, you can move, squeeze, and stretch your image and also rotate it against the background frame. The opposite sides of your image, however, will remain parallel.

Maintain Right Angles

In this mode, the borders of the image always remain parallel to the borders of the frames. You can move, squeeze, and stretch your image, but it will remain perfectly aligned.

Corners 1-4

You can key a pair of values into each of the Corner 1 through Corner 4 parameters. Each parameter accepts two values between 0 and 1.

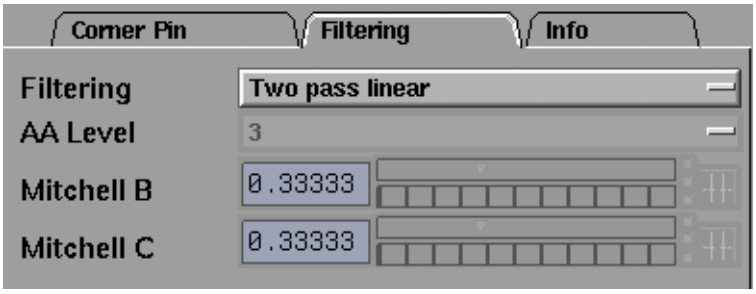
- Corner 1 represents the lower-left corner of the cropping box.
- Corner 2 represents the lower-right corner of the cropping box.
- Corner 3 represents the upper-right corner of the cropping box.
- Corner 4 represents the upper-left corner of the cropping box.

Pixel coordinates are produced using the following computation, using Corner 1 as an example.. If we define the Corner 1 values as X1 and Y1:

- X1 multiplied by the width of the source image will generate the X pixel coordinate.
- Y1 multiplied by the height of the source image will generate the Y pixel coordinate.

Filtering Parameter Tab

The Filtering tab enables you to select an appropriate filtering option for your imagery from the Filtering popup menu.



No Filtering

This option will give you the fastest result and may be adequate for previews; however, it is not recommended for final output.

Super Sampling

Super sampling is adequate if you are performing only a translation. But this form of filtering degrades the image when scaling to a significant degree. The associated AA Level parameter allows you to set the level of anti-aliasing.

AA Level

The AA Level parameter becomes active if Super Sampling is chosen from the Filtering menu. The AA Level controls the level of anti-aliasing in the image. Anti-aliasing is a filtering technique that creates the appearance of smooth lines and edges on a raster image display.

Select one of six levels of anti-aliasing using the popup menu. Level 1 provides the least amount of smoothing; level 6 provides the maximum. Level 3 is the default. The higher the level of anti-aliasing, the longer the node will take to complete its processing, but the sharper the image will be. You may want to use a low level of anti-aliasing to experiment, and then reprocess with a higher AA Level setting.

Two Pass Linear

Two pass linear and Mitchell filtering are both better choices than super sampling for major scaling or rotation operations. Two pass linear is preferable when scaling an image, as long as the image is not very noisy, because it is a bit faster than Mitchell.

Two Pass Mitchell

Two pass Mitchell is an excellent low pass filter for noisy images. It is likely to be the best choice for Jpegs or images with a lot of fine lines.

Mitchell B and C Parameters

You can make adjustments using the Mitchell B and C parameters, which become active when Two Pass Mitchell filtering is selected.

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.

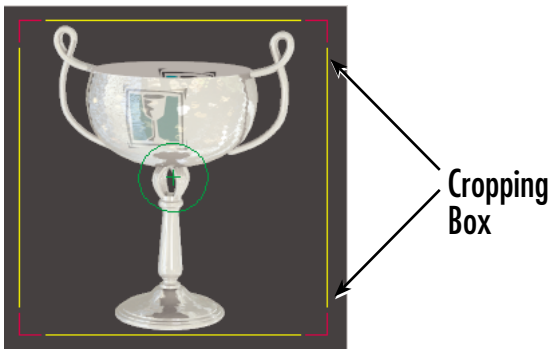
Two Pass Lanczos

The Lanczos filter provides a good compromise between sharpness, ringing and aliasing reduction. It is best used on images which are reduced or scaled down.

CROP NODE

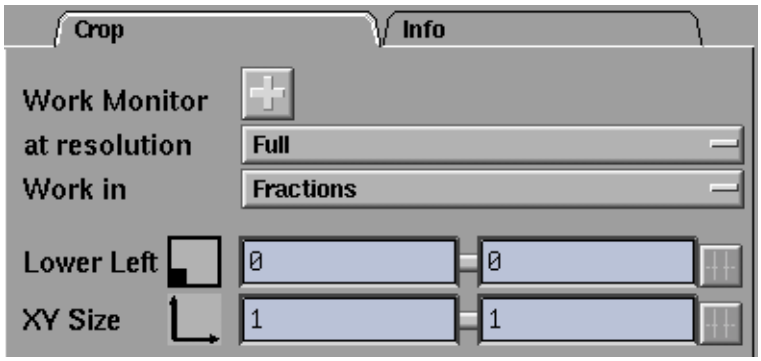
The Crop node enables you to crop, or trim, images to a smaller size or to a desired area of the input image. The position and dimensions of the cropped image can be defined in either of two ways:

- numerically, using the appropriate Crop tab parameter fields to specify a new XY Size and the location of the Lower Left Corner, or
- interactively, using the cropping tool in the work monitor to frame the image area to be output.



The cropping tool in the work monitor is tied to the corner and size parameter fields on the Crop tab; when one is changed the other is updated automatically to reflect the changes. In fact, it is usually most efficient to use both methods together.

Crop Parameter Tab



Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node, which displays the source image under a cropping box that defines the desired dimensions of the cropped image.

You adjust the size and position of the cropping box interactively:

- Use the mouse to click and drag the edges of the cropping box to resize it.
- Click on the circle-and-crosshairs icon in the center of the box to drag the box to a new position within the larger monitor image.

Work In...

Use this menu to specify whether the parameter values you enter for size and position represent pixels or fractional multipliers. The default is to work in fractional values—floating point values between 0 and 1.

Fractions

Working in fractions enables you to use proxies, where a lower resolution proxy input is eventually replaced by a full resolution input, without having to account for the change in the input resolution—a fractional value of 0.5 in the X size field will reduce the width of both the proxy and the full resolution image by half.

Pixels

On the other hand, working in pixels is useful when you know the exact pixel dimensions to which you want to crop as, for example, when converting imagery digitized from full aperture film format to standard Academy format.

> **Tip:**
>
> You can use pixel values to set the crop size values and then switch to fractions. Chalice will translate the pixel values to percentages, to enable the Crop node to successfully accommodate a later change from a lower resolution input to a full resolution input.

Lower Left Corner

These parameter fields are used to enter the *x,y* values that define the lower left corner of the crop area. Along with the XY Size parameters, this defines the crop; that is, the total area to be included in the output image.

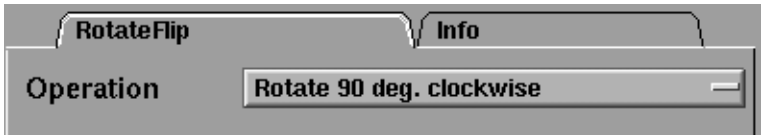
XY Size

These parameter fields are used to enter the X and Y resolution of the crop area, starting from the lower left corner (which is defined by the parameter above).

ROTATE FLIP NODE

The Rotate Flip node enables you to rotate an image about its axis in increments of one-quarter turn and to flip an image horizontally and vertically.

Rotate Flip Parameter Tab



Operation

The Operation parameter features a popup menu from which you can select any of the following rotate or flip operations:

- Rotate 90 degrees clockwise
- Rotate 180 degrees
- Rotate 90 degrees counterclockwise
- Horizontal flip
- Vertical flip

SCALE NODE

The Scale node enables you to enlarge or reduce the size of an input image. You can specify exact numerical values, in pixels, or use multiplier values to scale the image in the X and Y dimensions. The Scale node accepts one or two inputs. The top input is the image to be scaled. The optional bottom input is used as a size reference for the scale operation.

Tip:

If you are scaling an input to match the size of another image in the network, the fastest method will probably be to connect this reference image to the second Scale node input. This will automatically scale the top input image to match.

Scale Parameter Tab

The Scale tab enables you to specify either specific XY dimensions in pixels or the factor by which the input image will be scaled in X and Y. First you choose whether to work in Pixels or Fractions, and then you enter values into the parameter that is activated by the choice.

Work In...

Use this menu to specify whether the parameter values you enter represent pixels or fractional multipliers. The default is to work in Fractional values—floating point values between 0 and 1.

Tip:

Always work in fractions (or switch to fractions after entering specific pixel values) if you are working with proxies, where a lower resolution proxy input is eventually replaced by a full resolution input.

This saves you from having to account for the change later by going back into the Scale node to adjust pixel values manually—a fractional value of 0.5 in the X and Y fields will scale both the proxy and the full resolution image by half.

Fractions

First, the Fractions option must be selected from the "Work In" menu to activate this parameter. Then you can key a pair of values into the data entry fields: the left column specifies the multiplier value for the X dimension; the right column specifies the Y dimension.

Pixels

First, the Pixels option must be selected from the "Work In" menu to activate this parameter. Then you can key a pair of values into the data entry fields that represent the exact dimensions, in pixels, to which you want to scale the image: the left column specifies the number of pixels you desire for the X dimension; the right column specifies the Y dimension.

Filtering

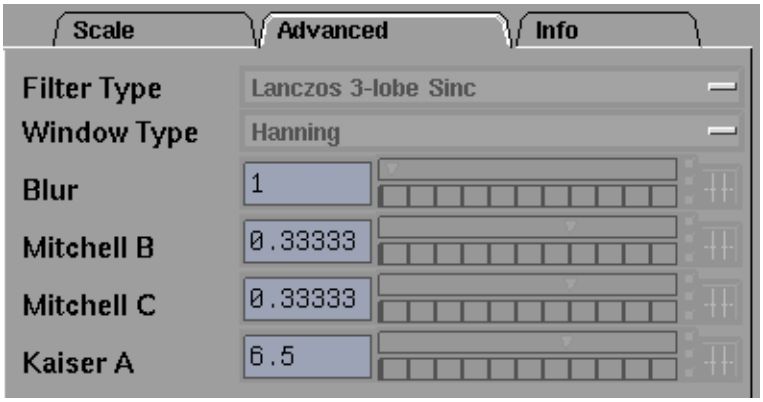
The Filtering menu offers filtering options for the scale operation: The default is "Best (Dynamically Chosen)," which means that Chalice will choose between Mitchell and Lanczos

filtering. However, you can also select “Good (Bilinear Interpolation)” to speed processing, or the fastest option, “Fast (No Filtering),” when image quality is not a top concern, as when reviewing your initial settings.

The “Advanced - User Set” option activates the parameters on the Advanced tab, which provide control over the type of filtering to use for the current imagery.

Advanced Parameter Tab

This parameters on this tab are activated when the “Advanced - User Set” option is chosen from the Filtering menu on the Scale tab.



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 function curve 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 operation the node is performing.

The Mitchell and Lanczos filters are among the most popular filter options. Mitchell filtering is often preferred for Jpegs or images with a lot of fine lines. Lanczos filtering provides a good compromise between sharpness, ringing and aliasing reduction. It is best used on images which are reduced or scaled down.

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

This parameter is activated only when the Sinc or Bessel filter type is selected. Choices are Hanning, Hamming, Blackman, or Kaiser.

Kaiser A

This parameter is activated only when Kaiser is chosen in the Window Type parameter.

Blur

The Blur parameter can be used with any filter option when heavy blurring is needed. The default value of 1 adds no blurring to the image; values greater than 1 represent increased blurring, up to the maximum of 4.

TRANSFORM NODE

The Transform node enables you to rotate, translate, or scale an image, or insert a smaller image into a larger image.

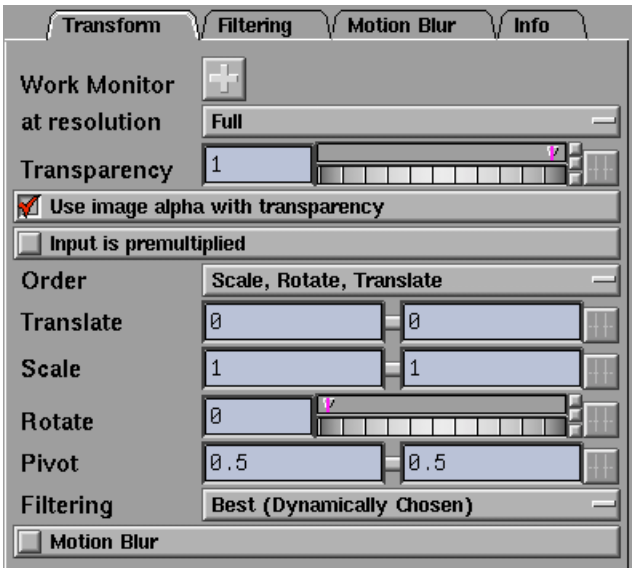
You can use the Transform node to animate the motion of one image relative to another or to control the specific placement of composited imagery. You can also simulate camera moves such as pans or zooms, or wildly complex shots that integrate pans, zooms, translations, or rotations.

The Transform node accepts one or two inputs:

- The top input is the image to be transformed.
- The optional bottom input defines the dimensions of the output and serves as a visual reference in the Transform work monitor.

Transform Parameter Tab

The Transform tab provides two groups of parameters: the first group controls the work monitor display; the second group governs the actual transformations.



Work Monitor Display Parameters

The first set of parameters in the Transform tab control work monitor display:

Work Monitor / At Resolution

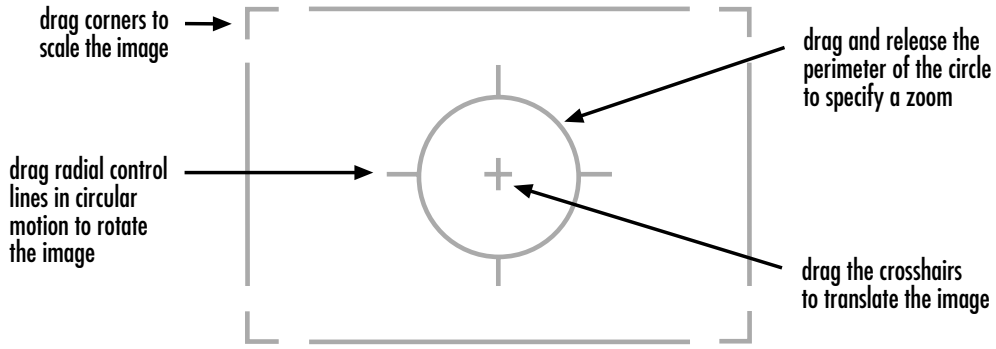
First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node.

Note:

Assuming that you have used the optional bottom input to the Transform node, the work monitor will display both input images in a temporary composite, with the input to be transformed (the top input) over the background image (the bottom input). This enables you to use the second input image as a reference while you operate on the primary image.

However, only the primary image will be output from the node. To actually composite the imagery, you would use a composite node such as Mcomp.

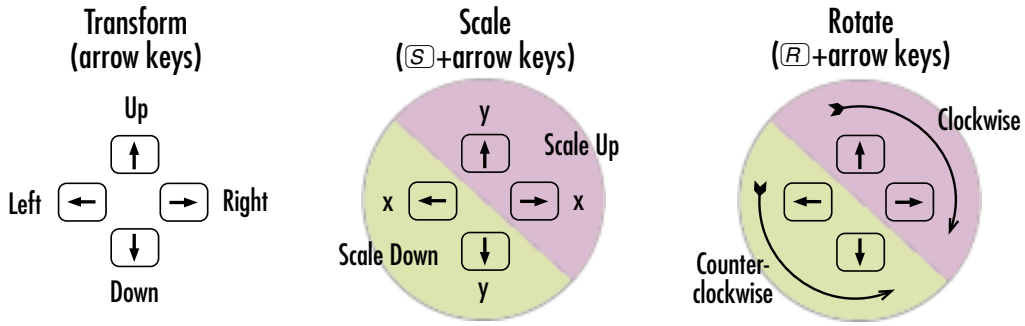
The Transform work monitor enables you to interactively control the Rotate, Scale, and Translate parameters using a boundary box and circle-with-crosshairs device that includes four radial control lines around the outside of the circle.



Hot Keys:

You can use the arrow keys on your keyboard to Translate, Scale, and Rotate in small increments. Each nudge will update both the work monitor display and the associated parameter fields.

The default mode for the arrow keys is Translate. To scale or rotate, you hold down a modifier key while pressing the arrow keys: to scale, hold down the "s" key; to rotate, hold down the "r" key.



Translate is a 1-pixel nudge. Rotate is a 1-degree nudge. The scale nudge is a fraction of image size: 10 / max(xres,yres)

Transparency

This parameter is used to set the level of transparency of the top input over the (optional) bottom input as it is viewed in the temporary composite displayed in the Transform work monitor.

Use Image Alpha with Transparency

When this option is checked, the alpha channel value of each pixel of the primary input is multiplied by the Transparency parameter value to govern the overall transparency of the temporary composite image in the work monitor.

Input Is Premultiplied

Check this box to inform Chalice when the primary input to the Transform node is a premultiplied image. For more information About Premultiplication, refer to chapter 15.

Transformation Parameters

The Rotate, Scale, Translate, and Pivot parameter fields update to reflect any modifications you make using the interactive crosshairs device in the work monitor. You can also set these parameters by entering values directly into the parameter fields and you can use hot keys to transform, scale, or rotate in small increments (refer to the diagram in the previous section).

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 most cases, where the pivot point will be at the center of the image being transformed. But for transformations where the pivot will be located at a corner (a swinging arm motion, for example), then the "Translate, Rotate, Scale" option would be preferable.

Rotate

The Rotate parameter enables you to rotate the top input image. Values are expressed in degrees (0–360). A positive value rotates the image in a counterclockwise direction. Alternatively, you may enter a negative value to rotate the image in a clockwise direction. The rotation is centered around a pivot point that you specify by using the Pivot parameter.

Scale

The Scale parameter enables you to scale, or proportionally adjust the total size of, the top input image. The scale is centered on a pivot point that you specify in the Pivot parameter.

The Scale parameters are simple multiplier values for the X and Y coordinates. The 0,0 point is the lower-left corner of the source image. For example, full size (no change to the input) is expressed by values of 1,1 in the data entry fields. And entering 0.5 in both fields will create an image that is half as high and half as wide as the source image.

Translate

The Translate parameter enables you to shift the position of the top input image along the X and Y axes.

You set the Translate parameter by keying in a pair of numerical values. The first value shifts the image along the X axis (horizontally). The second value shifts the image along the Y axis (vertically). The distance is measured as a percentage of the X or Y axis within a range of 0 to 1. For example, values of 0.5, 0.0 will shift the image to the right a distance equal to one-half the width of the image.

Pivot

The Pivot parameter enables you to specify a pivot point to be used in conjunction with the Rotate and Scale parameters (see above). The pivot point is the center of the rotation or scale.

You set the Pivot parameter by keying in a pair of values in the range of 0 to 1. These values represent an *x,y* location specified as a percentage of the image width and height. The lower-left corner of the image is the 0,0 point; the upper-right corner of the image is located at 1,1; the center of the image is 0.5, 0.5.

Filtering

This menu enables you to select an option for filtering the transformation operations. The default is "Best (Dynamically Chosen," however, you can select "Good (Bilinear Interpolation)" to speed processing, or the fastest option, "Fast (No Filtering)," when image quality is not a top concern, as when reviewing your initial transformation settings. The fourth option, "Advanced - User Set," enables the parameters on the Filtering tab, which provide control over the type of filtering to use for the current imagery and operation.

Motion Blur

Check this box to enable motion blur for the transformation. To control various aspects of the motion blur, you can then adjust the parameters on the Motion Blur tab.

Filtering Parameter Tab

The parameters in the Filtering tab are enabled when the "Advanced - User Set" option is selected from the Filtering menu on the Transform tab.

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 function curve 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.

The Mitchell and Lanczos filters are among the most popular filter options. Mitchell filtering is often preferred for Jpegs or images with a lot of fine lines. Lanczos filtering provides a good compromise between sharpness, ringing and aliasing reduction. It is best used on images which are reduced or scaled down.

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, values above 0.5 for Mitchell B may produce unnecessary blurring. Values above 0.5 for Mitchell C 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. Choices are Hanning, Hamming, Blackman, or Kaiser.

Kaiser A

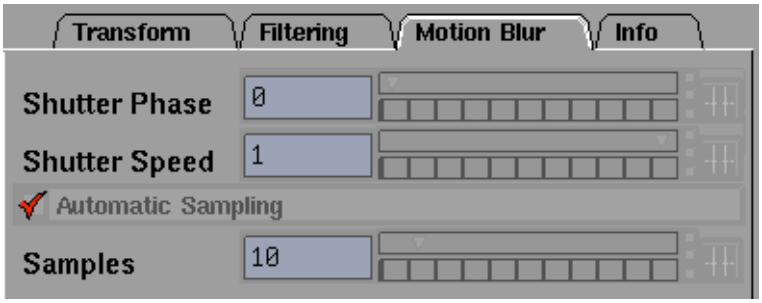
When Kaiser is chosen in the Window Type parameter, this parameter is activated.

Blur

The Blur parameter can be used with any filter option when heavy blurring is needed. The default value of 1 adds no blurring to the image; values greater than 1 represent increased blurring, up to the maximum of 4.

Motion Blur Parameter Tab

These parameters are activated Motion Blur is enabled on the Transform tab.

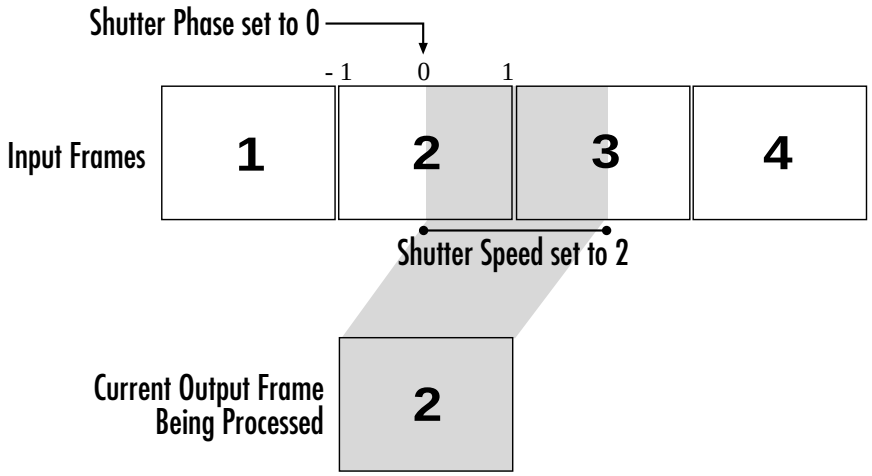


Shutter Phase

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 center 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.



Automatic Sampling

Automatic Sampling is enabled by default, to let Chalice set this value for the operation. However, you can turn off this feature to set a specific value in the Samples parameter.

Samples

The Samples parameter enables you to specify the number of subframes to examine when computing the motion, when you have disabled Automatic Sampling. The faster the motion in the sequence being blurred, the greater the number of samples needed. Larger values mean longer processing times.

MATCH MOVE AND STABILIZE NODES

The Match Move and Stabilize nodes enable you to apply track data to the control of translation, rotation, and scaling operations. As their names imply, the Match Move node is used to match the movement of a foreground element to a background and the Stabilize node is used for image stabilization.

Both nodes provide popup menus to select the Track node to reference—it is *not* necessary to enter mathematical expressions to use them.

Note:

Match Move and Stabilize do not create tracking data. Refer to the Track Node description in chapter 13, "Tool Nodes," for detailed instructions on generating track data.

Both nodes feature the same set of parameter tabs for specifying the operations to perform and any filtering or motion blurring to use; therefore both nodes are described together here.

Match Move and Stabilize each accept one or two inputs. The second, optional input is used to resize the primary input and as a visual reference in the work monitor.

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 (one that doesn't move off camera, e.g.).

For match moves, follow these steps:

1. **In the Apply Track tab, specify a Track node from which the Match Move node should access *x,y* position data.**
   You can enter the name of the track node into the Track Node parameter field, or you can use the associated popup menu to select any existing Track node in the file.

   Once a Track node is selected, other relevant parameters on the Apply Track tab become active

2. **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 doing Scaling or Rotation only.)

3. **Specify the Translation Point to use.**
   The Track node being accessed may include data for multiple track points. Therefore, you should specify which track point to use by selecting it from the Translation Point popup menu, which automatically lists all track points in the node specified in the Track Node parameter.

4. **Connect the background image to the second input of the Match Move node.**
   This changes the output resolution of the foreground image to match the size of the background input.

5. **Open the work monitor.**
   This will enable you to view the foreground over the background image as a reference while setting parameters.

6. **Optionally, you can create an Offset for the position data.**
   You can either key *x,y* coordinate values directly into the Offset parameter fields or drag the image in the work monitor to reposition it by eye.

7. **Optionally, you can modify scaling and rotation.**
   Refer to "Matching or Undoing Scaling and Rotation" for details.

Note:

The Match Move operation is the equivalent of entering the following expressions in the Translation fields of the Transform node, in which values are accessed for *x* (XV) and *y* (YV) positions, respectively, at the current frame ($F) of track point 1 (1) in a node named "track1":

```
track("track1",XV,1,$F)
track("track1",YV,1,$F)
```

Using the Stabilize Node

To stabilize imagery, for instance to remove camera jitter, 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. **In the Apply Track tab, specify a Track node from which the Stabilize node should access *x,y* position data.**
   You can enter the name of the track node into the Track Node parameter field, or you can use the associated popup menu to select any existing Track node in the file.

   Once a Track node is selected, other relevant parameters on the Apply Track tab become active.

2. **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 doing Scaling or Rotation only.)

3. **Specify the Translation Point to use.**
   The Track node being accessed may include data for multiple track points. Therefore, you should specify which track point to use by selecting it from the Translation Point popup menu, which automatically lists all track points in the node specified in the Track Node parameter.

4. **Open the work monitor.**
   This will enable you to view the image while setting parameters.

5. **Optionally, you can create an Offset for the position data.**
   You can either key *x,y* coordinate values directly into the Offset parameter fields or drag the image in the work monitor to reposition it by eye.

6. **Optionally, you can modify scaling and rotation.**
   Refer to "Matching or Undoing Scaling and Rotation" for details.

Note:

The Stabilize Image operation is the equivalent of entering the following expressions in the Translation fields of the Transform node, in which the position coordinates of a track point at the current frame are subtracted from the values of the same coordinates at the first frame:

```
track("track1",XV,1,1)-track("track1",XV,1,$F)
track("track1",YV,1,1)-track("track1",YV,1,$F)
```

Matching or Undoing Scaling and Rotation

You can match or undo scaling and rotation in both the match move and stabilize operations. Additionally, both nodes provide a "No Translation" option to use tracking data to match or undo scaling and rotation alone.

Note:

The scale and rotate operations in the Match Move and Stabilize nodes 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 (Scale or Transform, e.g.) between the source image and these nodes.

1. **Select an option from the Rotation and/or Scale parameters for how the track data should be applied to the operation.**
   For example, you can choose to match or undo the rotation in the original track. You can match or undo both size and distance scaling. Refer to the "Rotation and Scale Parameters" description for more detailed information.

2. **Select a pair of tracking points to use from the Rotate Point and/or Scale Point menus, which list all track points in the Track node being accessed.**
   The Rotation and Scale operations each require the specified Track node to have at least two points tracked because the scale and rotation values are controlled by changes in the length or relative position of the vector between the two track points selected.

Track and Image Data Filtering

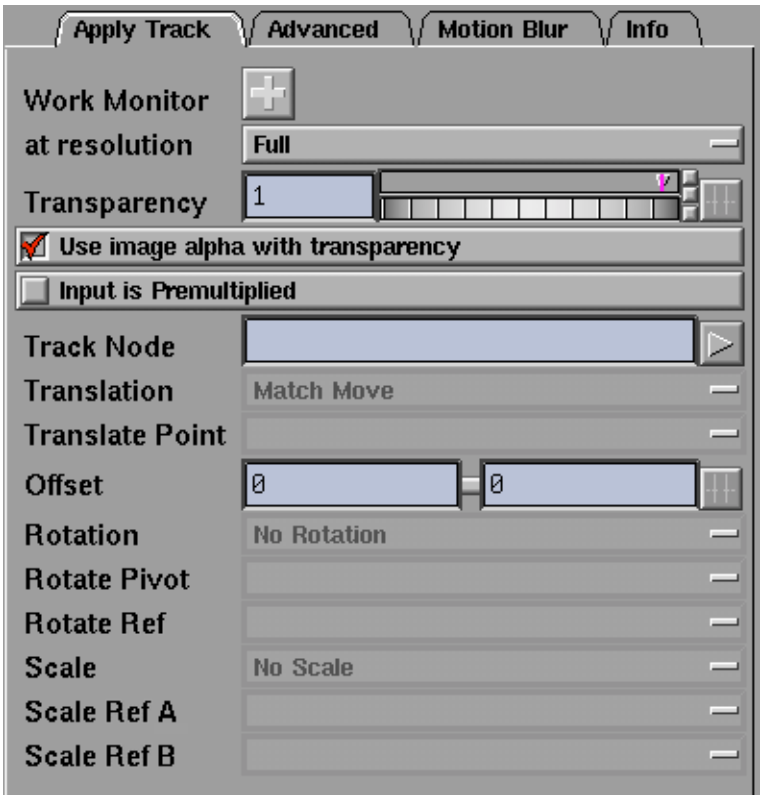
The Advanced parameter tab of the Match Move and Stabilize nodes features parameters you can use to filter both

- the track data, if you need to smooth the motion curve over time, and
- the image data, to reduce aliasing and other image artifacts that can be caused by scaling and rotation operations.

You can also specify motion blur for an operation.

Whether to use these options, and which options to use, is specific to the imagery and track data being accessed and the decision requires an examination of the initial result. For more information, refer to the descriptions of the Advanced and Motion Blur parameter tabs.

Apply Track Parameter Tab



Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node, which is used to specify an offset interactively by dragging the image in the monitor. It also enables you to view the placement of the image over a reference image in a temporary composite.

Transparency

If you connect a reference image to the second input of the node, you can view the primary image superimposed over this reference image. The Transparency tuner enables you to change the overall opacity of the foreground image in the work monitor by specifying a multiplier for the alpha channel data. By default, the Transparency parameter is set to 1, to use the full opacity of the input alpha channel.

Use Image Alpha with Transparency

When this option is checked, the alpha channel value of each pixel of the primary input is multiplied by the Transparency parameter value to govern the overall transparency of the temporary composite image in the work monitor. Otherwise, Chalice will assume an alpha value of 1 for each pixel.

Track Node

Use this parameter to specify the Track node from which to access tracking data. You can use the popup menu, which lists all existing Track nodes in the grail file, or you can key in the name of the Track node to use.

Translation

This parameter offers a popup menu from which you choose the type of translation operation to be performed:

- In the Match Move node, the choices are Match Move or No Translation.
- In the Stabilize node, the choices are Stabilize or No Translation.

No Translation

The No Translation option should be selected only if you are using the Match Move or Stabilize node strictly to match or undo rotation or scaling, without any other translation.

Translate Point

This parameter enables you to specify which track point to use for the translation. The Translate Point popup menu will display all track points available from the specified Track node.

Offset

The Offset parameter enables you to offset the position data used for translation, rotation, and scaling. You can key specific *x,y* values into the Offset data fields, or you can use the work monitor to drag the image to the offset position you desire. The Offset parameter values will update accordingly.

Rotation and Scale Parameters

The Rotation and Scale parameters enable you to select from several options for the type of operation. 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 is used to calculate the values.

Rotation

The Rotation popup menu enables you to select No Rotation (the default), Undo Rotation, or Match Rotation.

The associated Rotate Pivot and Rotate Reference parameters become active whenever Undo or Match Rotation is selected. Use the popup menus to select the track point data to use for each parameter.

For **Rotate Pivot**, use the track point that represents the pivot of the rotation; for **Rotate Ref**, use the point that represents the position of the moving end of the rotation vector.

Scale

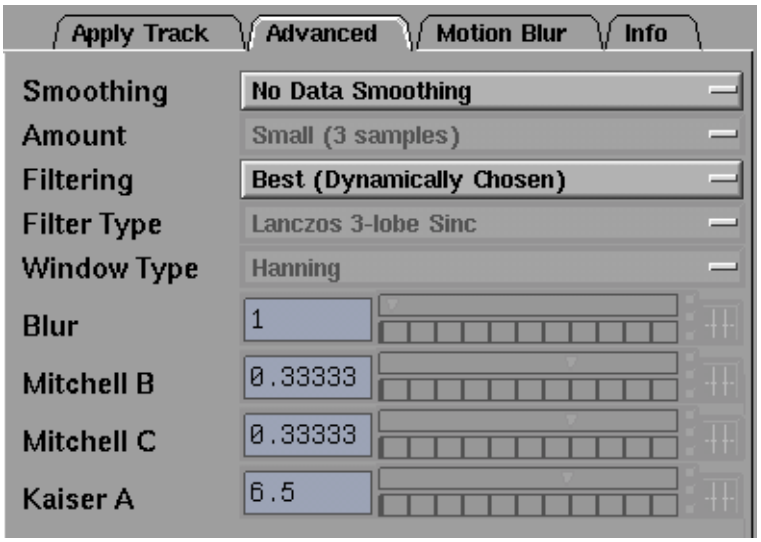
The Scale popup menu enables you to select No Scale (the default), Undo Size Scale, Undo Distance Scale, Match Size Scale, or Match Distance Scale:

- The **Size Scale options** enable you to match or undo a scale up or down in size, such as a zoom.
- The **Distance Scale options** enable you to match or undo relative changes in distance between objects.

The associated Scale Reference parameters become active whenever any option other than No Scale is selected. Use the **Scale Ref A** and **Scale Ref B** popup menus to select the track point to use to represent each end of the vector defining the scale value.

Advanced Parameter Tab

The Advanced parameter tab enables you to choose from several types of data smoothing and image filtering operations. The Smoothing parameters modify the track data, while the Filtering parameters modify the imagery.



Smoothing

This parameter menu enables you to apply a smoothing filter to the track data, to effectively smooth out excessive jerkiness in the motion curve from frame to frame. Instead of using the raw track point data, smoothing will average the values using the function you specify in the popup menu: No Data Smoothing (the default), Weighted Smooth (using a triangle filter), or Average Smooth (a box filter).

Amount

This parameter is activated whenever Weighted or Average Smooth is selected from the Smoothing menu. It enables you to specify the sample size, in frames, to use for the Smoothing filter: Small (3 samples), Medium (5 samples), or Large (7 samples).

Filtering

This menu enables you to select an option for filtering. The default is "Best (Dynamically Chosen)," which means that Chalice will choose between Mitchell and Lanczos filtering. However, you can also select "Good (Bilinear Interpolation)" to speed processing, or the fastest option, "Fast (No Filtering)," when image quality is not a top concern, as when reviewing your initial settings.

The "Advanced - User Set" option activates the Filter Type parameter, which enables you to select a specific type of filter operation.

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 function curve 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.

The Mitchell and Lanczos filters are among the most popular filter options. Mitchell filtering is often preferred for Jpegs or images with a lot of fine lines. Lanczos filtering provides a good compromise between sharpness, ringing and aliasing reduction. It is best used on images which are reduced or scaled down.

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. Choices are Hanning, Hamming, Blackman, or Kaiser.

Kaiser A

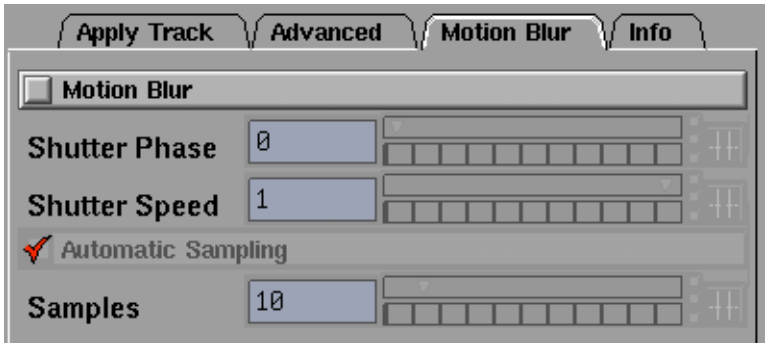
When Kaiser is chosen in the Window Type parameter, this additional control parameter is activated.

Blur

The Blur parameter can be used with any filter option when heavy blurring is needed. The default value of 1 adds no blurring to the image; values greater than 1 represent increased blurring, up to the maximum of 4.

Motion Blur Parameter Tab

These parameters are activated when Motion Blur is enabled on the Apply Track tab.

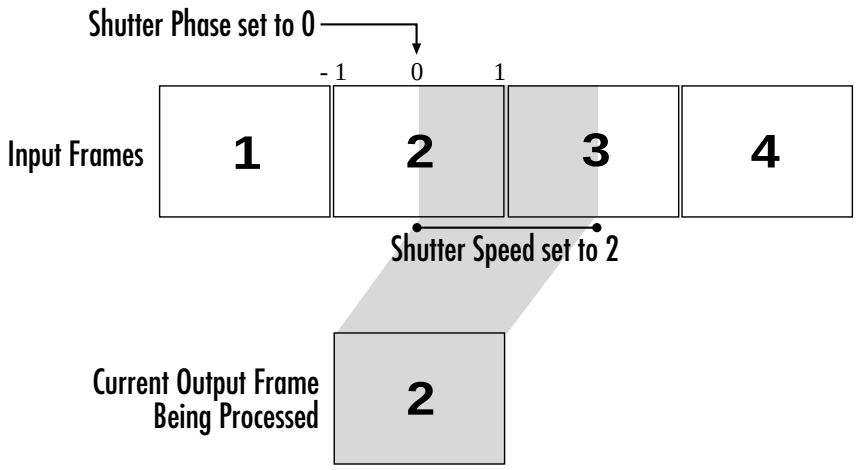


Shutter Phase

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.



Automatic Sampling

Automatic Sampling is enabled by default, to let Chalice set this value for the operation. However, you can turn off Automatic Sampling to set a specific sample value in the Samples parameter.

Samples

The Samples parameter enables you to specify the number of subframes to examine when computing the motion, when you have disabled Automatic Sampling. The faster the motion in the sequence being blurred, the greater the number of samples needed. Larger sample values mean longer processing times.

18 FILTER NODES

The Filter nodes modify imagery (as opposed to combining it with other imagery like the Composite nodes). They take two inputs, the primary input and an optional control image input, as explained below in "Using Control Images with Filter Nodes."

Note:

The Convolve node takes a second, optional input for use as a graphical kernel in the convolution, and a third, optional input for use as a control image.

The Filter nodes include color adjustment and area operation nodes; therefore they are discussed in two groups:

About the Color Filters ..... p. 18.3
- Color Correct Node ..... p. 18.4
- Brightness Node ..... p. 18.14
- Contrast Node ..... p. 18.16
- Gamma Node ..... p. 18.18
- Hue Adjust Node ..... p. 18.19
- Inverse Node ..... p. 18.21
- Monochrome Node ..... p. 18.22

About the Area Filters ..... p. 18.23
- Blur Node ..... p. 18.24
- Convolve Node ..... p. 18.26
- Degrain Node ..... p. 18.31
- Edge Node ..... p. 18.34
- Erode/Dilate Node ..... p. 18.36
- Grain Node ..... p. 18.38
- Noise Node ..... p. 18.41
- Rank Node ..... p. 18.42
- Sharpen Node ..... p. 18.46
- Unsharp Mask Node ..... p. 18.49

Using Control Images with Filter Nodes

The filter nodes accept an optional input image that is used to control which pixels of the primary input image will be processed in the filter operation.

Two control image parameters in the node panel enable you to specify which channel of the control image to use and how to use the channel data: the Control Action menu and the Control Channel menu.

Control Action Menu

The Control Action menu lets you specify how the information in the selected control channel will be used. The two methods are On/Off and Mix, and in both cases the value of each pixel in the control channel governs how the corresponding pixel in the primary input image

will be used. (The "corresponding pixel" refers to the pixel with the same *x,y* image coordinates.)

On/Off Method

The On/Off method specifies whether a pixel will be operated on by the filter:

- For any control 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 control channel pixel with a nonzero value, the corresponding pixel in the primary image will be filtered by the node, just as it would if there was no control image input.

Mix Method

The Mix method uses the value of each control channel pixel to determine whether the corresponding pixel in the primary input will be filtered, and the extent to which it will be affected by the node operation.

As with the On/Off method, a control channel pixel with a value of 0 will turn off filtering, and a pixel with a value of 1 will turn filtering on. However, pixels with values in between 0 and 1 will be filtered, and the output will be a mixture of the filtered and unfiltered value of the primary input pixel, with the ratio being determined by the floating point value of the control channel pixel.

For example, a control channel pixel with a float value of 0.32 will turn on the filter 32 percent, so that the corresponding output pixel will be a mixture of 0.32 of the filtered input pixel value and 0.68 of the original input pixel value.

Control Channel Menu

This menu enables you to specify which channel in the control image will be used to govern the control action, when the control image has more than one channel.

ABOUT THE COLOR FILTERS

The array of color filters in Chalice give you the option to perform multiple concatenated operations to an image within a single node, or to select a single-purpose node for each type of color adjustment, based on your needs and preference.

The Color Correct node is the most comprehensive and powerful color adjustment node in Chalice. It can be used to make a single type of adjustment, or you can choose as many other types of adjustment as necessary for an image. Each operation is adjusted in a separate, selectable layer in the Color Correct work monitor.

The Color Correct node is the best choice when multiple types of correction are needed, or when you need more precise control over an individual operation. For example, brightness, contrast, and gamma can be adjusted separately in the shadows, midtones, and highlights for each channel individually, and you can change the definition of shadows, mids, and highs for any image:

Color Correct Node ..... p. 18.4

You can also make fast adjustments to a specific image attribute using an individual node designed for that one purpose, including hue-based color correction in the Hue Adjust node:

Brightness Node ..... p. 18.14
Contrast Node ..... p. 18.16
Gamma Node ..... p. 18.18
Hue Adjust Node ..... p. 18.19
Inverse Node ..... p. 18.21
Monochrome Node ..... p. 18.22

COLOR CORRECT NODE

The Color Correct node is a comprehensive tool for performing a wide variety of adjustments. It provides seven common color manipulation tools in one layer-based interface for fast, simple color correction.

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 HSL-based color adjustments
- operate on individual channels
- swap channels
- invert colors
- desaturate colors
- simulate printer lights and stops

In the Color Correct node, adjustments are made directly in the work monitor, using the tools provided. For each type of adjustment you wish to make, you create a separate layer. 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.

Speed and Accuracy

Using the Color Correct node to perform multiple color correction operations on an image is both simpler and more accurate than using a separate node for each adjustment.

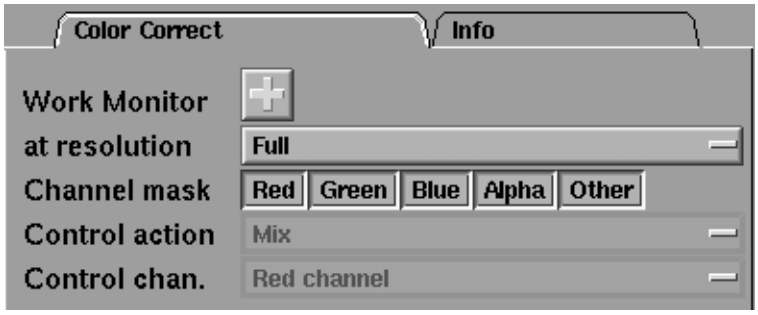
The result is faster and more accurate because each successive operation is cumulative. Regardless of the bit depth of the input image, the node calculates each operation in floating point. All of the data in the result of the first operation is preserved and used in the next operation—even those values that go out of bounds and would have to be clamped on final output.

Note:

Three operations, by their nature, are exceptions to this rule: HSL, Channel Swap, and Monochrome. Whenever you create a layer of one of these three types you break the chain and potentially lose some color data.

Color Correct Parameter Tab

This tab gives you access to the Color Correct work monitor. The parameters available for each type of adjustment, which are accessed from within the work monitor, are described in the following section, "Using the Color Correct Work Monitor."



Work Monitor / At Resolution

First, select a display resolution for the work monitor from the At Resolution popup menu. Then click the plus icon to display the work monitor for this node.

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel buttons.

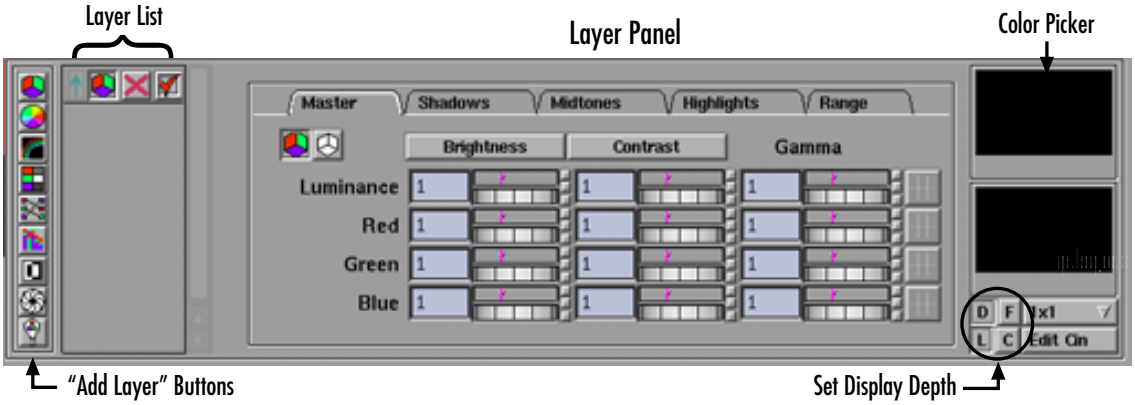
Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

Using the Color Correct Work Monitor

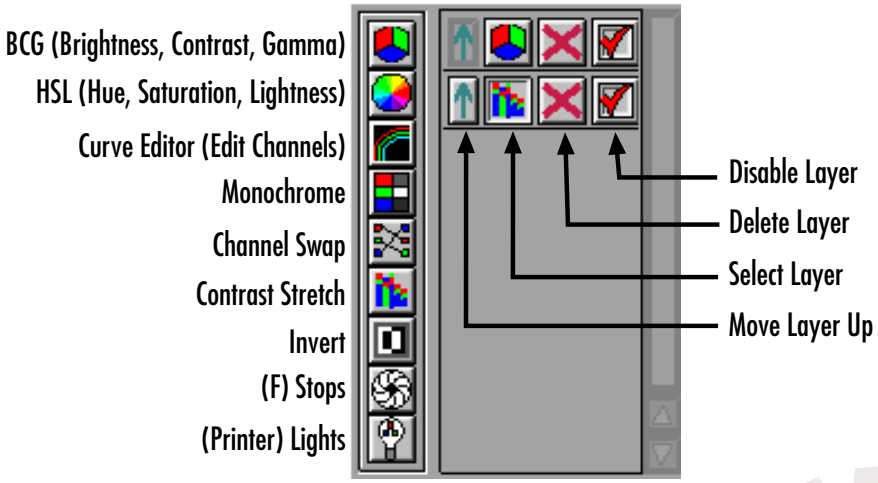
The Color Correct work monitor provides a unique panel where you create a layer with specific parameters for each type of operation you wish to perform.

Color Correct Monitor Panel



Working with Layers

Click any layer button to create a correction panel for that operation. A corresponding entry will be created in the layer list.



Each entry in the layer list consists of four icons:

- The **Move Layer** icon (up arrow) is used to move a layer up one position in the list. Each layer is calculated in order from the top of the list to the bottom. (Refer to "Speed and Accuracy" in the introduction of this node description for more information about why the order of layers is important.)
- The **Select Layer** icon is used to select a layer and display its associated parameters in the layer panel.
- The **Delete Layer** icon (red "X") is used to delete a layer.
- The **Disable Layer** icon (checkbox) is used to temporarily disable a layer. When the box is checked, the layer is enabled and will contribute to the result; when the box is not checked, the layer is disabled and will remain so until you check the box again.

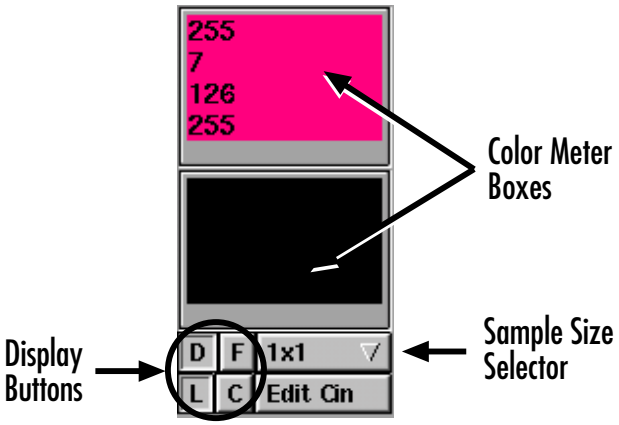
Selecting a Display Depth

The D and F buttons at the lower right corner of the layer panel enable you to control how values are displayed in the Color Picker and in the graph displays available in some layers: select "D" to use the Default bit depth in the graph displays; select "F" to use Floating Point (0–1) in the graph displays. This option enables you to display input values in whatever unit is most appropriate to the task.

For example, if you are working in 8-bit, choosing Floating Point display of input values would enable you to reduce the brightness of the image to 80 percent (0.8 on the graph) without calculating which value between 0 and 255 is equivalent. On the other hand, if you are working in Cineon linear and you know that your white point is 4095, you would probably prefer to use Default input display (0-65535).

Using the Color Picker

The Color Picker can display the RGBA value of any pixel, or pixel area, in an image. It is especially useful for comparing color values in two different images using A/B Wipe.



To use the Color Picker, simply click the mouse in one of the two Color Meter boxes. The cursor will change into an eye dropper—this is the Color Picker tool.

Then click on any pixel in the image that you want to meter. The color values of that pixel will appear in the Color Meter box and the cursor will revert to normal shape. (If you are sampling an area of pixels, their average value will be displayed.)

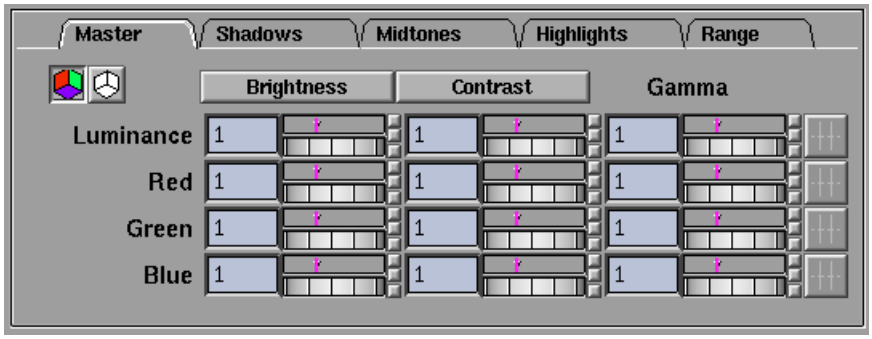
To sample an area of pixels, click the Sample Size Selector to select the appropriate size pixel grid from the popup menu: 1x1, 3x3, 5x5, 7x7, 9x9, or 11x11.

BCG (Brightness, Contrast, Gamma) Layer

The BCG panel enables you to adjust Brightness, Contrast, and Gamma values using tuners. The BCG panel features five tabs: Master, Shadows, Midtones, Highlights, and Range.

BCG Master, Shadows, Midtones, and Highlights Tabs

The first four tabs provide the same set of parameters, which enable you to specify Brightness, Brightness Offset, Contrast, Contrast Offset, and Gamma for each channel (RGBAO) and for Luminance.



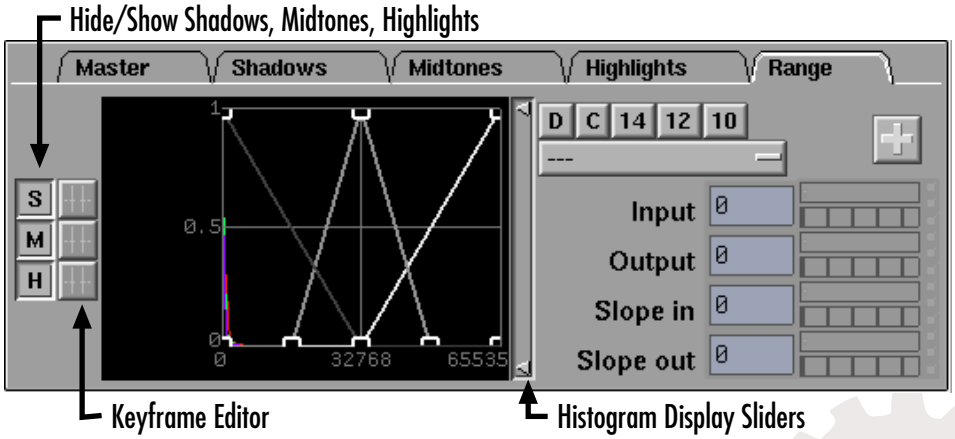
You can adjust the entire image in the Master tab, and/or adjust the Highlights, Midtones, and Shadows by selecting the appropriate tab, all of which feature the same set of parameters.

The default value for all of the Brightness, Contrast, and Gamma parameters is 1, which represents no change from the input image. Offsets for Brightness and Contrast can be reset for each channel:

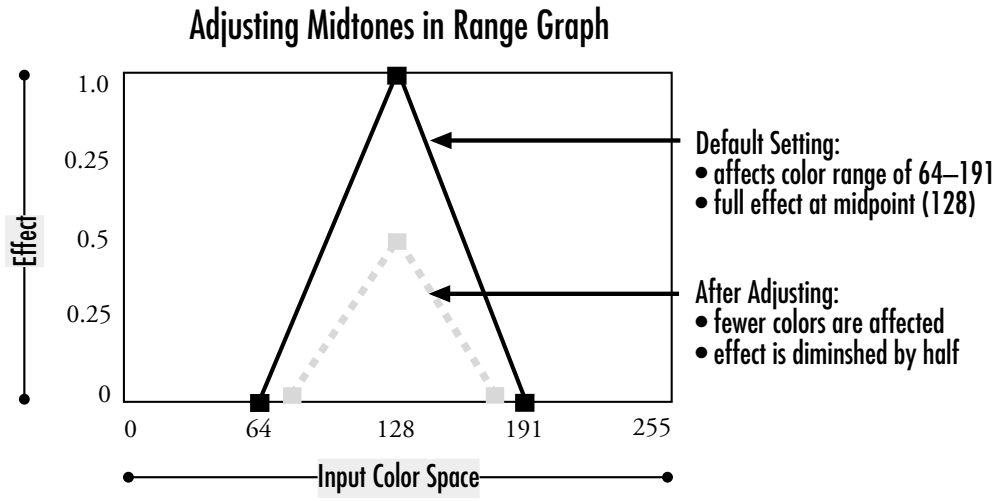
- Click the “Brightness” button to toggle to “Brightness Offset” (the default brightness offset is 0).
- Click the “Contrast” button to toggle to “Contrast Offset” (the default contrast offset is 0.5).

BCG Range Tab

The fifth BCG tab, Range, enables you to control the effect of any adjustments you make in the Shadows, Midtones, and Highlights tabs. You can specify the range of values that will be affected, as well as the magnitude of the effect.



The following illustration demonstrates how the Range tab can be used. If you made this adjustment to the Midtones curve in the Range tab, it would affect any changes in parameter values that you made in the Midtones tab. A smaller range of colors would be affected, and the effect would be diminished by half.



You can adjust the graph by manipulating the graph display, or by keying specific values into the Input and Output tuners:

- The Input value represents a point on the colorspace.
- The Output value represents the level of effect at that point (in a range of 0–1).

You can turn the display of shadows, midtones, or highlights on and off in the graph by clicking the S, M, and H buttons on the left side of the Range panel. To bring up a Keyframe Editor for each, click the associated button.

The BCG Range tab interface works exactly like the interface provided in the Curve Editor layer (see the description of the Curve Editor Layer for more information). You can add and delete points and apply different functions to the distribution curves.

The only difference is that the BCG Range panel displays a histogram of the current image in the graph. You can adjust the display of the histogram, if necessary, to make it easier to see, by dragging the vertical sliders on the right side of the graph.

HSL (Hue, Saturation, Lightness) Layer

The HSL layer enables you to perform hue-based color adjustments using the HSL color model.

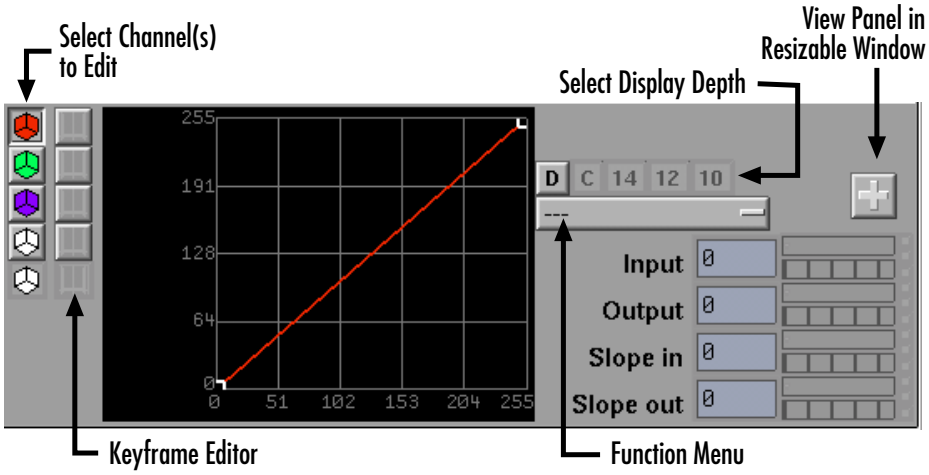
The **Hue Shift** parameter enables you to rotate the color distribution vectors of the image around the axis of the colorspace. You can perform the hue shift visually by dragging the color wheel or you can use the tuner to enter a value in the range of -180 to 180 degrees. A value of 0 represents the input data.

The **Saturation** parameter enables you to adjust the chrominance level by keying a value in the range of -1 to 1 into the data entry field. A value of 0 represents the input data value, -1 represents complete desaturation, and 1 creates maximum saturation.

The **Lightness** parameter enables you to adjust relative intensity, or brightness, by keying a value in the range of -1 to 1 into the data entry field. A value of 0 represents the input data value (i.e., no change), -1 will result in a completely black image, and 1 will result in a completely white image.

Curve Editor Layer

The Curve Editor enables you to modify the color attributes of an image by editing curves that represent color channels 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 hide or display any channel, and you can modify channels separately or together. You can bring up a Keyframe Editor for each channel to animate values over time.

Display Depth

When working with 16-bit (or float) imagery, you can choose to display the data in the graph in Default view (0–65535), Cineon view (0–4095), 14-bit (0–16383), 12-bit (0–4095), or 10-bit (0–1023) by clicking the appropriate Display Depth button: D, C, 14, 12, or 10.

When working in 8-bit space, only the Default view (0–255) is available.

Editing Curves

To work with one channel at a time, select that channel by clicking its icon, and then deselect any other selected channels.

To select a point, click on it. To select overlapping points on different curves, hold down the Alt key and drag a bounding box around the points in the graph.

To add a point, simply click on the curve and a point will be created. To delete a point, click on the point to select it and then press the Delete key on the keyboard.

Tip:

The Plus icon (+) enables you to bring up an identical Curve Editor panel in a resizable window. Increasing the size of the graph can make it easier to select and manipulate points and curves.

Input and Output Parameters

The Input and Output parameters display the current values to which the selected point is mapped. As an alternative to dragging a point in the graph to reposition it, you can key specific values into the data entry fields to remap values.

Slope In and Out Parameters

The Slope In and Slope Out parameters display the current value of the slope of the curve at the selected point. If the Equal Slopes or Different Slopes function has been selected from the Slopes menu, the fields become active and you can key specific values into the data entry fields to define the slopes.

Function Menu

The Function menu enables you to select Linear, Continuous, Equal Slopes, or Different Slopes options for the selected point. The menu displays the type of curve used for the currently selected point. If no point is selected, the menu displays a row of dashes.

Monochrome Layer

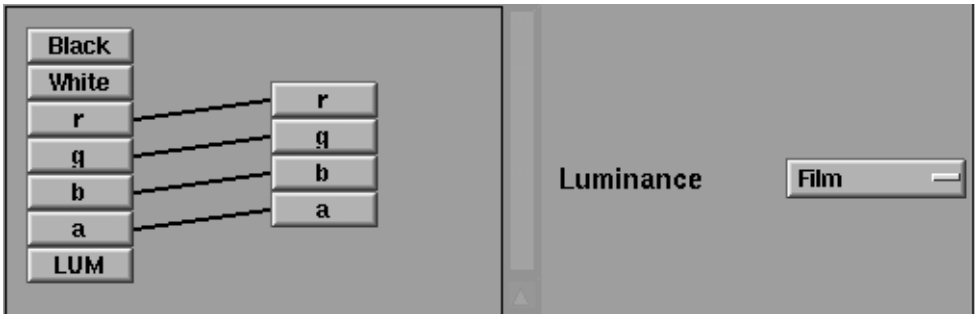
The Monochrome layer desaturates an image to the degree you specify. This value, which is calculated using luminance, can be animated over time.

The monochrome tuner range is 0–1, with 0 representing no change in the image and 1 representing a totally monochromatic (grayscale) image.

You can use the Luminance popup menu to specify whether Film, NTSC, or PAL luminance should be used in the calculation. For more information about the formulas used to compute luminance, refer to the Channel Swap Node description in chapter 13, “Tool Nodes.”

Channel Swap Layer

This layer enables you to swap channel values or to fill any channel with black, white, or luminance.



The left-hand column represents the channel data available for use, and the right-hand column represents the output channels. The connecting lines drawn between the columns indicate what channel data is being used for each output channel.

To replace the existing data in a channel, simply click on a channel in one column, and then click on a channel in the other column. A line is drawn automatically to connect them. For example, you could click on Black in the left column and then on the alpha channel in the right column to replace the existing alpha channel with an entirely transparent alpha.

If you choose to fill a channel with luminance, use the Luminance popup menu to specify whether Film, NTSC, or PAL luminance should be used in the calculation.

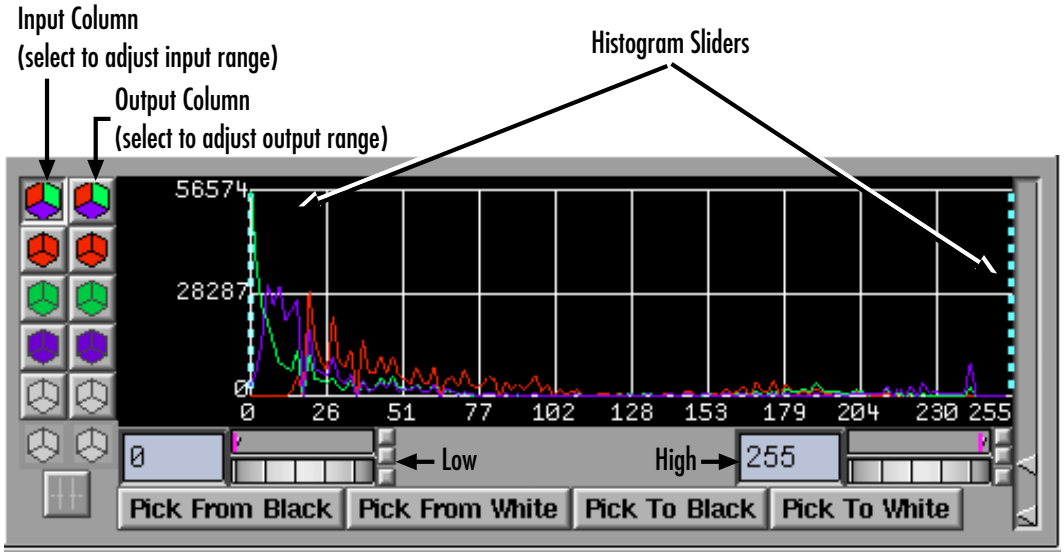
Contrast Stretch Layer

The Contrast Stretch layer enables you to increase or decrease the contrast of an image or image channel. You start by specifying a low value and a high value, which define the range of colors in the input image that will be remapped to the output colorspace.

By default, the selected range is remapped to the entire colorspace. For example, working in 8 bits per channel, the selected range will be remapped to 0-255 and any values in the input that fall outside of the selected range will be clipped.

However, you can specify a new low and high value in the output colorspace to use for the remapping. This will remap the selected input range to the new low and high, and any input values falling outside the range will not be clipped but will gradually fall off in a linear distribution to the minimum or maximum value of the output colorspace.

By specifying a smaller range for the output than the input, you can effectively decrease the contrast of the image.



Adjust Input Range with "Pick From" Adjust Output Range with "Pick To"

—use "Pick" buttons to specify black (low) and white (high) values by clicking on pixels in image—

Adjusting the Input Range

To specify the range of the input to use, be sure that the input column is selected by clicking the RGB cube in the leftmost column—it should be selected by default. (To adjust a channel or channels individually, select the Red, Green, Blue, Alpha, or Other cube in the leftmost column.)

With the input column selected, the low and high values can be adjusted numerically or visually, by referencing the histogram data or clicking in a portion of the image.

To specify the low and high values numerically, key specific values into the data entry fields, or drag the associated tuner dial to change the values. When referencing the frequency distribution data, you can drag the vertical bars (dashed lines) in the histogram to redefine the range. The values in the data entry fields will update accordingly.

The input low (black) and high (white) values can also be selected visually, by clicking a "pick from" button. The cursor will change to an eye dropper tool, which you can use to select a white or black pixel in the monitor image. You can select the average of an area of pixels by adjusting the Sample Size Selector (refer to "Using the Color Picker" if you need more information).

Tip:

If you are not satisfied with the result of a "Pick From" operation, you can disable the layer and the Pick buttons will remain active. This enables you to see the "before" image in the monitor again and use it pick a new black or white point. When you enable the layer again, Chalice will recalculate using the new pick value.

Adjusting the Output Range

The same methods described above for the input range can also be used to adjust the output range. First, you must select the output column by clicking the RGB (or individual channel) cube in the column next to the input column. Then specify the range.

Note:

You can use an A/B wipe to access another image and then use this reference image to pick pixels that represent the black and white point for the output range. For the output range, be sure use the "pick to" buttons instead of the "pick from" buttons.

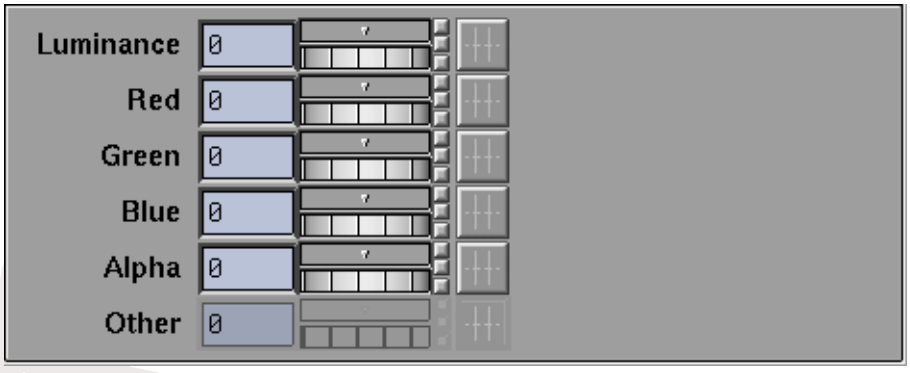
Invert Values Layer

The Invert Values layer 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 panel enables you to specify which channels should be inverted. By default, all available channels are inverted. To deselect a channel, click on its icon in the panel.

Stops Layer

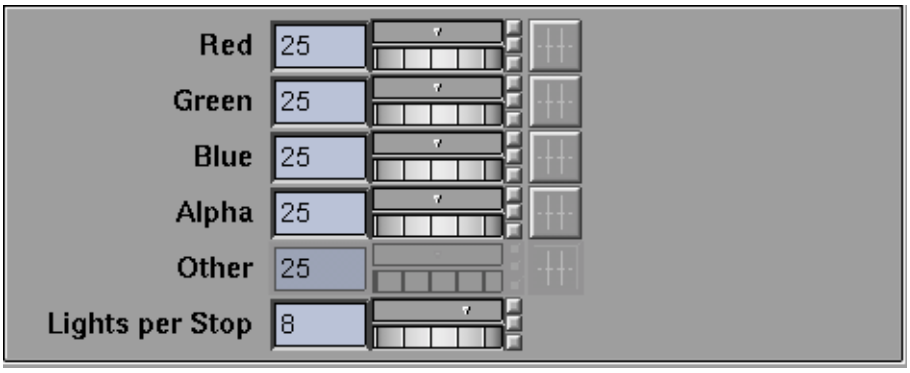
The Stops layer enables you to control the color values at any channel in an image using photographic stops.



Increasing a channel by one stop will double the color values at each pixel. This effect is cumulative; bringing a channel up two stops will increase the amount of light by a factor of four. Similarly, bringing a channel down one stop will reduce the color values by a factor of two.

Printer Lights Layer

The Printer Lights Layer enables you to control the color values at any channel in an image using printer lights.

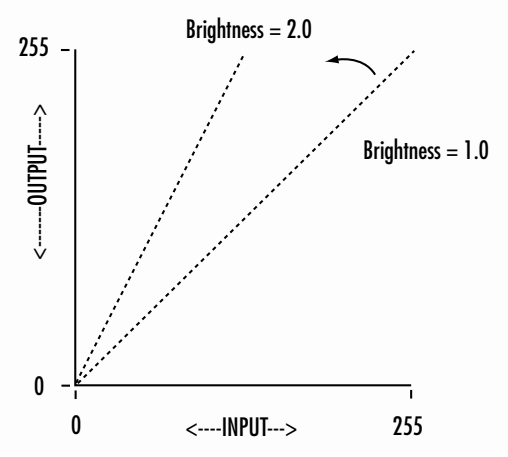


The **Lights per Stop parameter** specifies how many Printer Lights must be added to a channel to double the color values. The default values of 25 represent no change to the image. At eight lights per stop, an addition of eight printer lights will double the color values. At four lights per stop, an increment of eight printer lights will increase the color values fourfold.

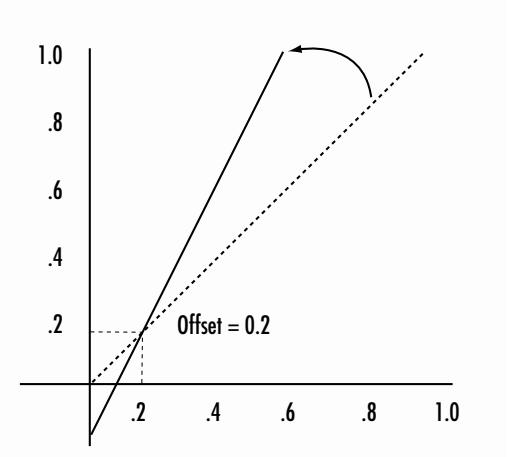
BRIGHTNESS NODE

The Brightness node enables you to vary the brightness of an image by assigning a value to the Brightness parameter.

The Brightness node allows you to specify an offset for the brightness value. The following illustration shows the change to the color range when the Brightness parameter is changed from 1.0 to 2.0 and the Offset parameter is changed from 0 (the default) to 0.2.



BRIGHTNESS = 2.0; OFFSET = 0 (default)



BRIGHTNESS = 2.0; OFFSET = 0.2

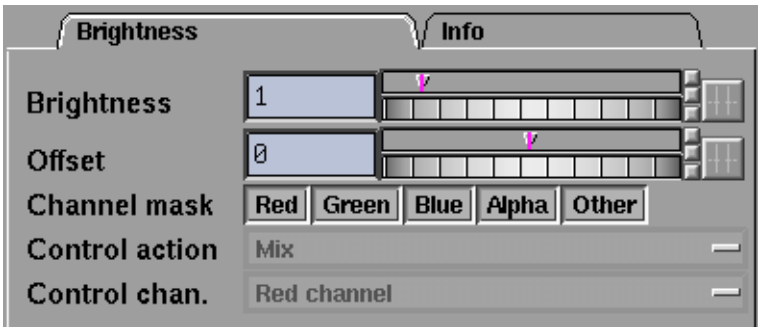
The Brightness node performs the following computation:

V = ((V - Offset) * Brightness) + Offset

For Brightness settings greater than 1, the Brightness node boosts values above the offset and attenuates values below it. For Brightness settings less than 1, the Brightness node attenuates values above the offset and boosts values below it.

For integer data, the Brightness node clamps values at the maximum for the colorspace (255 or 65535). Float values are not clamped.

Brightness Parameter Tab



Brightness

You set the Brightness parameter by keying in a numerical value between 0 and 10.0 (inclusive) or by using the mouse to set the tuner to the desired value. The default value is 1.0, indicating no change to the image.

Offset

You set the Offset parameter by keying in a numerical value between -1.0 and 1.0 (inclusive) or by using the mouse to set the tuner to the desired value. This value determines the offset point for the brightness adjustment. (See the illustration at the top of this node description).

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

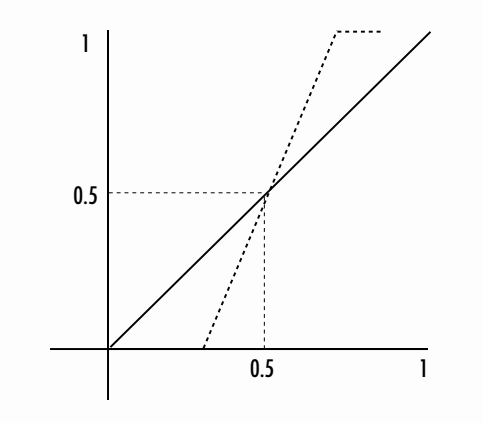
Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

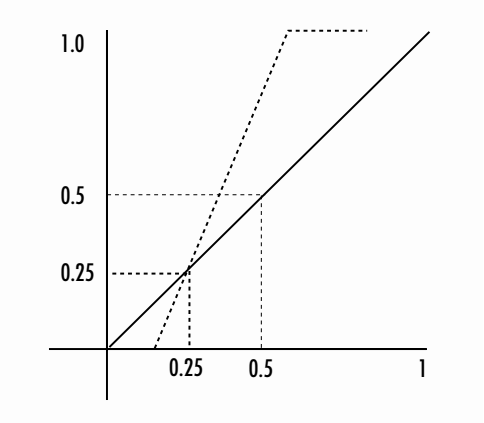
CONTRAST NODE

The Contrast node enables you to vary the contrast of an image, or individual image channels, effectively compressing or expanding the color range. The Contrast node includes a Pivot parameter that allows you to specify a pivot point about which the compression or expansion of pixel values will fluctuate.

The following illustration shows a graph in which the Contrast parameter is set to 1.5 and the Pivot parameter is set to 0.5 (default, left) and to .25 (right).



CONTRAST = 1.5, PIVOT POINT = 0.5 (default)



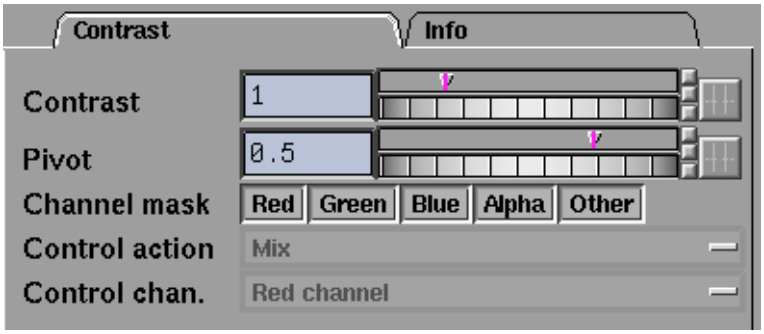
CONTRAST = 1.5, PIVOT POINT = 0.25

The Contrast node performs the following computation:

V = ((V - Pivot) * Contrast) + Pivot

This formula has the effect of increasing or decreasing the range of color values in the image.

Contrast Parameter Tab



Contrast

You set the Contrast parameter by keying in a numerical value in the preferred range of 0 to 5.0 (though the parameter is unconstrained) or by using the mouse to set the tuner to the desired value. The default value is 1.0.

Pivot

You set the Pivot parameter by keying in a numerical value in the preferred range of -1.0 to 1.0 (though the parameter is unconstrained) or by using the mouse to set the tuner to the desired value. The default value is 0.5.

The Pivot parameter allows you to determine the pivot point about which the compression or expansion of pixel values will fluctuate. It lets you tune the contrast adjustment according to values of a particular image. (See the illustration at the top of this node description).

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

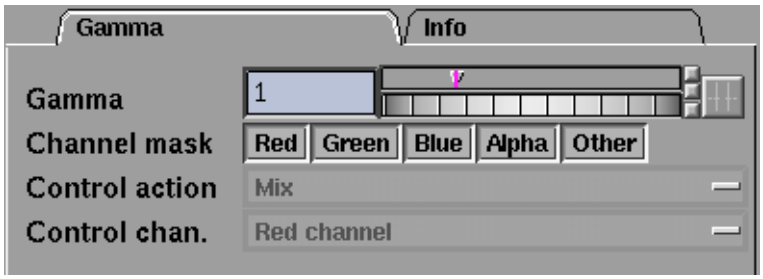
GAMMA NODE

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):

$$V = V^{1.0/Gamma}$$

Gamma Parameter Tab



Gamma

You set the Gamma parameter by keying in a numerical value in the preferred range of 0.1 to 4.0 (though the parameter is unconstrained) or by using the mouse to set the tuner to the desired value. The default value is 1.0.

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

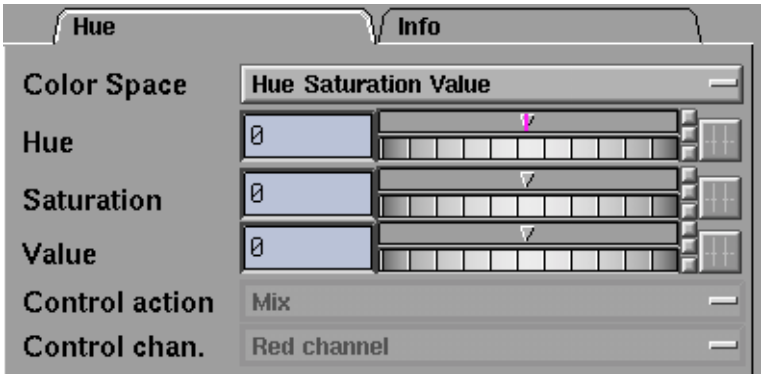
Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

HUE ADJUST NODE

The Hue Adjust node enables you to perform hue-based color adjustments to an image using the HSV and HSL color models. (To adjust RGB color values, use the Color Correct Node.)

Hue Parameter Tab



Color Space

The Color Space menu allows you to choose the colorspace to use for the adjustment:

- Hue Saturation Value (HSV)
- Hue Saturation Luminance (HSL)

The most significant difference between the two colorspaces is that, for HSV, maximum saturation values for any hue are on the same plane in the colorspace as maximum brightness (Value) levels; whereas for HSL, maximum saturation values are on the same plane as mid-range brightness (Luminance) levels.

Hue

The first parameter, Hue, allows you to rotate the color distribution vectors of the image 360° around the axis of the colorspace by selecting a value in the range of -180 to 180. A parameter value of 0 represents the hue data of the source image.

You can key a numerical value into the data entry field or use the mouse to set the tuner to the desired value.

Tip:

A vectorscope display of the data is available by clicking the Monitor Scope button on the Hue Adjust Monitor Toolbar (for more information, see the section on Using the Utility Monitors in chapter 8, “Monitors and Flipbooks”).

If Monitor Auto Update (in the Options menu) is turned on, changes to the parameter settings will be displayed interactively in the vectorscope as you make them.

Saturation

The second parameter, Saturation, allows you to adjust the chrominance level of the image by keying a numerical value in the range of -1 to 1 into the data entry box or by using the mouse to set the tuner to the desired value.

A value of 0 always represents the input image data; -1 results in complete desaturation of the image (monochrome); and 1 creates maximum saturation of the hues.

On the Monitor Scope, the degree of saturation is represented by the amplitude of the vector signal, with high saturation indicated by vectors near the outer edge of the scope and low saturation near the center.

Value/Luminance

The third parameter is either Value, if HSV is chosen from the Color Space menu, or Luminance, if HSL is chosen. Both Value and Luminance represent the relative intensity, or brightness, of the color data.

Value and Luminance both use the same range of -1 to 1. A value of 0 represents no change to the input image; -1 results in the minimum brightness; and 1, the maximum brightness.

A parameter setting of -1 will always result in a completely black image for both HSV and HSL colorspaces. A setting of 1, however, will result in blown-out colors for HSV, whereas the same brightness setting for HSL will result in a completely white image.

Note:

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 will be as great between numerical values of .10 and .11 as between .50 and .55.

Control Image Parameters

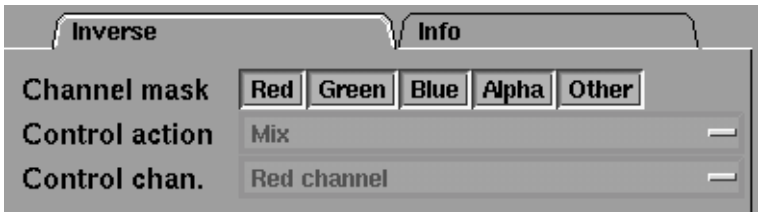
The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

INVERSE NODE

The Inverse node enables you to invert the color values in an image, effectively creating a negative of the image. The Inverse node performs the following computation, where Z is the maximum color value and V is the color value of a pixel:

$$V = Z - V$$

Inverse Parameter Tab



Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by de-selecting the Red, Green, Blue, Alpha, or Other channel icons.

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

MONOCHROME NODE

The Monochrome node enables you to desaturate the color component of an image to the extent that you specify. The Monochrome node performs the following computation:

R = mN + R(1-m)
G = mN + G(1-m)
B = mN + B(1-m)

(where R, G, and B are the red, green, and blue color values; N is the result of the Luminance computation; and m is the Monochrome Factor parameter value).

Monochrome Parameter Tab

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 popup menu. For the formulas used by Chalice to compute the three types of luminance, refer to the Channel Swap Node description in chapter 13, "Tool Nodes."

Monochrome Factor

The Monochrome Factor parameter enables you to control the degree of desaturation of the image. You set the Monochrome Factor parameter by keying in a numerical value between 0 and 1.0 or by using the mouse to set the tuner to the desired value.

A value of 0 yields no change to the image. A value of 1.0 completely desaturates the image, reducing it to a fully monochrome image (grayscale).

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

ABOUT THE AREA FILTERS

This section describes the filter nodes that operate on a pixel by taking into account the values of surrounding pixels, to perform filtering operations such as edge-detection, blurring and sharpening, noise reduction, etc.

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** and **Sharpen** nodes can be used simply by setting a value for the operation, or advanced filtering options in each node (low pass filter controls in Blur, and high frequency filter controls in Sharpen) can be accessed to precisely control a number of variables.

The **Noise** node adds video-type noise to an image, and the **Grain** node includes presets for various film stocks, as well as offering parameters for custom grain pattern matching. The **Degrain** node offers comprehensive control over the process of removing grain.

The edge operators include the **Edge** node, for edge detection, and the **Erode/Dilate** node, for shrinking or expanding edges with subpixel accuracy.

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 Filters menu includes the following area filters:

Blur Node ..... p. 18.24
Convolve Node ..... p. 18.26
Degrain Node ..... p. 18.31
Edge Node ..... p. 18.34
Erode/Dilate Node ..... p. 18.36
Grain Node ..... p. 18.38
Noise Node ..... p. 18.41
Rank Node ..... p. 18.42
Sharpen Node ..... p. 18.46
Unsharp Mask Node ..... p. 18.49

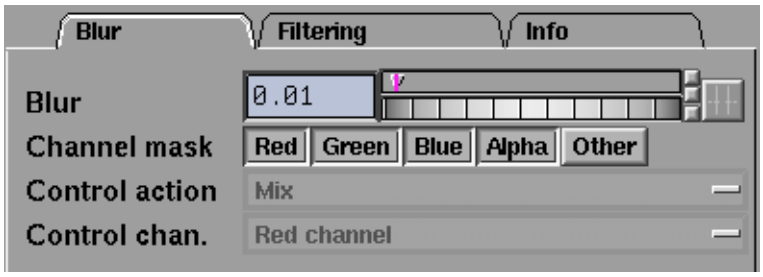
BLUR NODE

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.

To use the Blur node, specify a value in the Blur parameter tab. If necessary, you can customize the kernel used in the convolution by modifying the parameters on the Filtering tab.

Blur Parameter Tab



Blur

The Blur parameter enables you to specify the number of adjacent pixels whose values will be used to modify the current pixel. This value is expressed as a percentage of the total number of pixels in the image.

You set this parameter by keying a value in the range of 0 to 1 into the data entry field or by using the mouse to set the tuner to the desired value.

A value of 0 would leave the image unmodified by the convolution. A value of 1 would specify a kernel the size of the image, so that every pixel in the image would contribute to the modification of the current pixel. In most cases you will select values within a narrow range of the low end of the scale.

Channel Mask

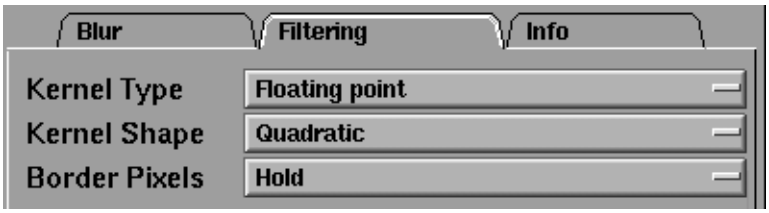
The Channel Mask parameter enables you to inhibit the modification of any channel to the blur effect by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

Filtering Parameter Tab

The convolution used for the blur modifies each pixel in the image, in turn, based on the values of surrounding pixels. The matrix that delineates this pixel area is known as a kernel. The Filtering tab parameters enable you to define the kernel attributes to control the blur.



Kernel Type

The Kernel Type parameter features a popup menu that allows you to specify Integer or Floating Point math for the calculation.

Note:

Integer is the default, unless you choose Cubic for the Kernel Shape parameter, in which case the Kernel Type parameter defaults to Floating Point. Integer precision restraints make it inefficient to compute the Cubic function using integer math.

Kernel Shape

The Kernel Shape parameter enables you to determine the extent to which each pixel in the kernel will contribute to the convolution by choosing a function that describes the distribution curve used to weight the sample. The following functions are available in the Kernel Shape popup menu: Constant, Linear, Quadratic, Cubic, or Gaussian.

The Constant function, for example, is equivalent to a box filter and applies the same weight to every cell (pixel) value in the kernel when it is factored. The weighting distribution becomes more complex as you select the other functions: Linear is equivalent to a triangle filter, and so on, with Gaussian being the most complex and computationally intensive.

The optimal choice will depend on the nature of the source image, the effect you wish to achieve, and how much time you are willing to devote to processing.

Border Pixels

By the nature of the convolve operation used to blur the image, the pixel currently being modified is in the center of the cell matrix that composes the kernel. Therefore, when convolving a pixel that borders the image (or when the kernel is large), there will not be adjacent pixels on all sides to contribute values to the convolution.

The Border Pixels parameter enables you to assign pixel values to these "empty" kernel cells by selecting one of the following options from a popup menu:

- **Hold:** This option, which is the default setting, assigns the values of the edge pixels to the kernel cells that fall outside the image borders.
- **Use Black:** This option assigns a value of 0 to each channel for kernel cells that fall outside the image borders.
- **Mirror Data:** This option assigns the values of the inner adjacent pixels to the corresponding kernel cells that fall outside the image borders.

CONVOLVE NODE

The Convolve node is a convolution filter. It can sharpen, soften, or emboss an image, detect its edges, and perform a variety of other functions. It is a very powerful node, but one that may take a little time to learn how to control and apply effectively.

The Convolve node operates on one pixel at a time, beginning with the first pixel in your image and proceeding sequentially through the image until every pixel has been filtered. The filtering process uses a convolution function to apply kernel cell values to each pixel. (The Convolution Kernel is explained in more detail below.)

You can select from a number of predefined convolution kernels (using the Library popup menu on the Convolve tab). You can also create your own filters by entering values into the cells of a kernel. You can modify the values of one of the predefined kernels in the library to suit your needs, or you can select a "generic" kernel (using the Kernel Size parameter) and modify the values in its cells. Another option is to add a control image to define the kernel, as explained in "Using an Input as a Kernel."

The Convolution Kernel

A kernel is a matrix of cells. Convolve offers three kernel sizes: 3 x 3, 5 x 5, or 7 x 7. In a nine cell (3 x 3) matrix, one central cell is surrounded by eight adjacent cells.

A	B	C
D	E	F
G	H	I

The central cell (E) represents the pixel to which the filtering process will be applied. The eight adjacent cells represent the eight adjacent pixels in the image. The adjacent pixels will not be modified when the filtering process is applied to the central cell; only the central pixel will change during this iteration of the filter. The adjacent cells will be modified when each of them in turn becomes the central pixel and is subjected to the filtering process.

Controlling Changes in Brightness

Unless you wish to modify the brightness of an image, the sum of all cell values must be equal to the number of cells in the matrix. In the following example, which uses integer data, all cells sum to nine (for floating point data, all cells must sum to 1). In this case, the overall brightness of the image being processed will not be affected.

1	1	1
1	1	1
1	1	I

If the sum is less than nine, the image will be darker after processing; if the sum is greater than nine, the image will be brighter after processing.

Blurring

The Convolve node will use the color values of the adjacent pixels to modify the color value of the center pixel. The extent to which each adjacent pixel will contribute information to the processing of the center pixel is represented by the values in the adjacent kernel cells.

The Convolve node will begin by looking at Pixel A and Pixel E. Since the value in each kernel cell is 1, the Convolve node will modify Pixel E so that it becomes 1/9 of its original value, plus 1/9 the value of Cell A, 1/9 the value of Cell B, etc.

In other words, each of the nine cells in the kernel will contribute its color value in equal measure to the resultant pixel; they will be "averaged" together, or in effect "blurred."

If you modified this matrix as follows, the resultant pixel would contain 2/9 of the color values of cells A, D, and G; 1/9 the color values of B, E, and H; and none of the color values of cells C, F, and I.

2	1	0
2	1	0
2	1	0

If you placed a 9 in the center of the matrix, surrounded by a zero in each adjacent cell, the Convolve node would have no effect on the image. It would process each pixel, but leave it unchanged (returning 9/9ths of its original value).

0	0	0
0	9	0
0	0	0

If you placed a 9 in an adjacent cell, as shown below, the Convolve node would effect a one-pixel shift, or offset, of the image to the right.

0	0	0
0	0	9
0	0	0

By creating a kernel with intermediate values that both blur and offset the image, you can effectively create a motion blur.

As mentioned above, blurring an image may be thought of as averaging the values of a pixel with its surrounding pixels. The more adjacent pixels you include in the computation, the blurrier the image will become.

Edge Detection

The inverse of a blurring process would entail detecting the edges of an object in an image and making the edges sharper. The Convolve node is also an effective edge-detection tool.

Three of the predefined matrices, "3x3 Laplacian," the "7x7 Prewitt," and "7x7 Canny," are effective edge detection paradigms. And of course, you may craft your own, depending on your requirements.

Convolve Node Inputs

The Convolve node accepts one, two, or three inputs:

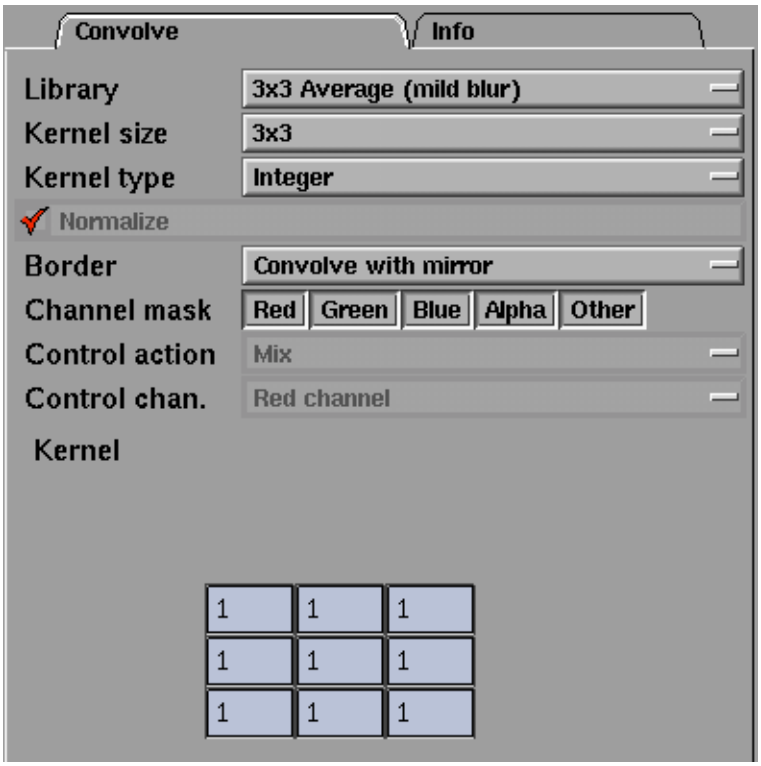
- **Input 1:** The top input is the image to be convolved.
- **Input 2:** The middle (optional) input is used as a convolution kernel.
- **Input 3:** The bottom (optional) input is used as a control image.

Using an Input as a Kernel

The middle, optional, input image is treated as a kernel in the convolution. That is, the value of each pixel represents a cell value in the kernel matrix.

Therefore, the kernel input must be one-channel image that is an odd-size square less than 256 pixels (a 15 x 15 pixel grid, for example, which consists of 225 pixels).

Convolve Parameter Tab



Library

The Library parameter enables you to choose from a menu of predefined cell matrices to perform various types of filtering processes, including several options for blurring and edge detection. The best choice will depend on the nature of the input image. The kernel you select is displayed in the node panel, where you can evaluate and modify cell values.

3 x 3 Average (Mild Blur)

This filter effects a mild blur.

3 x 3 Laplacian (Edges)

This filter is an edge-detection tool. The edges in the resultant image will be emphasized, appearing as bright lines.

3 x 3 High Pass (Sharpen)
This filter is used to sharpen the image.

5 x 5 Low Pass (Stronger Blur)
This filter is stronger than the 3 x 3 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)
This filter is an alternative to the low pass blur.

5 x 5 Quadratic (Smooth)
This filter is a subtle blur that removes noise to effectively smooth the image.

7 x 7 Prewitt (Edges)
This filter is an edge-detection tool that operates on a broader range of pixels than the Laplacian filter.

7 x 7 Canny (Edges)
This filter is an alternative edge-detector to the Prewitt filter.

Kernel Size
The Kernel Size enables you to select the size of the input matrix. You can choose from 3 x 3 (9 values), 5 x 5 (25 values), or 7 x 7 (49 values). The input matrix is displayed at the bottom of the Convolve parameter tab.

Kernel Type
The Kernel Type parameter allows you to choose either Integer or Floating Point data values.

Normalize
This option is applicable to floating point operations only. When checked, Normalize 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.

Border
The Border parameter enables you to specify how the Convolve node will handle the pixels that border the image, since these pixels do not have adjacent pixels on all sides to contribute values to the kernel matrix.

- **Do not convolve** leaves the values of the border pixels unchanged.
- **Convolve with black** assigns a value of 0 to kernel cells that fall outside the image borders.
- **Convolve with mirror** convolves each border pixel as if there were adjacent pixels on all sides by mirroring the values of the inner adjacent pixels, as illustrated below.

4	3	4	5	
1	0	1	2	
4	3	4	5	
7	6	7	8	

The white boxes represent actual pixels in the upper-left border of an image to be convolved; the gray boxes represent the mirror values that would be used in the top and left rows of cells of the kernel matrix.

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

DEGRAIN NODE

The Degrain node is used to help remove grain and other noise from imagery. The default Degrain method is essentially automatic—just connect the Degrain node and you're done. The other available Degrain node operations 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.

Choosing a Degrain Method

The default Degrain method, referred to as Spatial Analysis, performs an area operation not dissimilar to those used for some types of blurs. This averaging of pixel values will reduce the appearance of grain, but it can also eliminate fine image detail. This means that the Spatial Analysis method may make some imagery unacceptably soft.

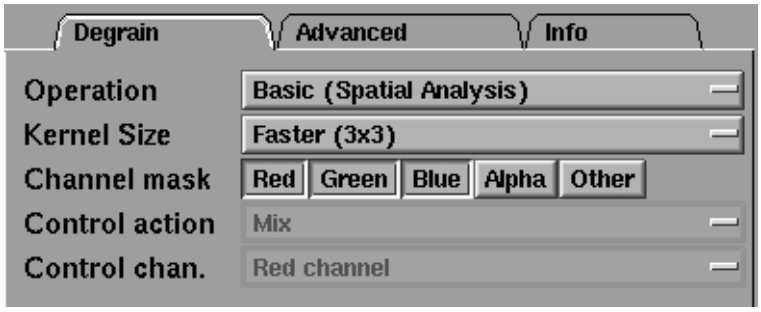
The other two Degrain operations you can choose, both of which use a method called Frequency Analysis, come optimized for film grain or for video noise.

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, such as 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.

These methods are described in detail below.

Degrain Parameter Tab



Operation

The Operation parameter enables you to specify the type of analysis used for the degrain operation: spatial or frequency analysis. You can choose between two types of frequency analysis, one optimized for film grain, the other for video noise.

Spatial Analysis

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 middle value in the ranking. (For more detailed information about this operator, refer to the Rank Node description. The Rank node includes a number of noise reduction presets.)

Spatial analysis is the default operation because it is much simpler to use than the other method, frequency analysis. However, the results of spatial analysis are more likely to look soft, or blurry, when compared with the frequency method.

Frequency Analysis

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 tab, is removed from the signal. Then this "cleaned up" signal is added back into the original in order to get some or all of the detail back into the image.

The frequency analysis approach requires that a number of parameter values be set with care. These parameters are described in the Advanced tab section.

Kernel Size

The Kernel Size menu is used to specify the size of the kernel used by the spatial analysis operation. As the menu options indicate, the smaller 3 x 3 kernel is faster, while the 5 x 5 kernel takes longer to process but produces better results.

The frequency analysis method also references 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.

Channel Mask

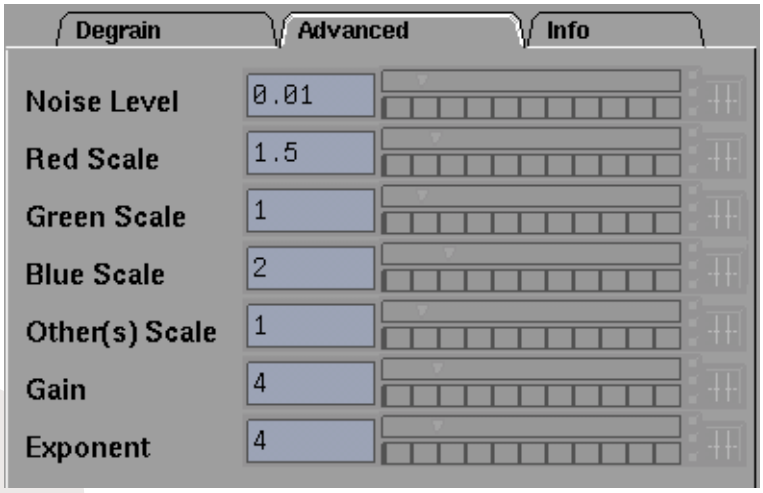
The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

Advanced Parameter Tab

The parameters on the Advanced tab become available when a frequency analysis option is selected from the Operation parameter on the Degrain tab. However, the channel-specific Scale parameters are disabled when Advanced Video is the selected operation.



Tip:

If you are unfamiliar with frequency analysis, read the parameter guidelines below, open a monitor on the Degrain node, and experiment. Your eye will tell you when you have a satisfactory result.

Noise Level and Scale Parameters

These parameters enable you to set a threshold that determines how much of the signal should be subject to filtering. The Noise Level parameter applies to all channels equally, whereas the Scale parameters are used to weight the influence of the noise level value differently for each channel:

Noise Level

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.

Scale (Red, Green, Blue, Other)

For film grain analysis, the Scale parameters enable you to adjust 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 values are a good starting point, as they reflect a ratio typical for many film stocks. (For video, however, the Scale controls are not active, because video signals do not exhibit noise differences between the red, green and blue components.)

Gain and Exponent Parameters

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.

EDGE NODE

The Edge Node enables you to select from several effective methods for performing edge-detection. Typically, the input to the Edge node is a foreground element that will subsequently be composited. The output of the edge node is often used as a control image in blurs, dissolves, and other operations to improve the realism of composites by softening or otherwise adjusting foreground edges.

The Edge node identifies edges by comparing each pixel with surrounding pixels. The pixel currently being processed is the center pixel in a kernel (a matrix, or grid) that you specify using the Edge node parameters.

Edge Parameter Tab

Method

This parameter enables you to choose from the following edge-detection methods:

- Roberts (2 x 2 Cross)
- Sobel (Two Direction)
- Kirsch (Eight Direction)

These options are listed in order of computation speed, from fastest (Roberts) to slowest (Kirsch). The best method to use will depend on your imagery; however, the default is Sobel, which usually provides excellent results without resorting to the much more computationally intensive Kirsch operation.

Filter Width/Pixels

The Filter Width and Pixels parameters enable you to specify the size of the kernel, or operation area, used by the Sobel or Kirsch methods. When Roberts is the selected method, the Filter Width tuner is grayed out and only the Pixels field is used to adjust the effect.

The Filter Width parameter and the Pixels parameter are tied; that is, when you adjust one, the value of the other updates accordingly. The value in the Filter Width tuner is expressed as a percentage of the width of the image, in a range of 0–1. The value in the Pixels field is expressed as an odd number of pixels (if you do enter an even number, Chalice automatically adjusts the value to the next lower odd value).

Border

The pixel currently being modified is in the center of a grid of pixels that define 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 to the calculation.

The Border parameter enables you to specify how border pixels should be treated by selecting one of the following options from the Border popup menu:

- **Mirror Data:** This option, which is the default setting, will process border pixels by assigning the values of the inner adjacent pixels in the grid to the corresponding locations in the grid that fall outside the image borders.
- **Use Black:** When this option is selected, a value of 0 is assigned to locations in the grid that fall outside the image borders.
- **Do Not Process:** When this option is selected, border pixels are not processed; that is, their original values are passed through unchanged.

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

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 Erode/Dilate is used primarily to modify matte edges for better composites.

Note:

If you want to modify the edges of an Ultimatte matte in an "over" composite, use the Ultimatte AdvantEdge node, which is described in chapter 15, "Composite Nodes," rather than Erode/Dilate.

The Erode/Dilate node accepts one or two inputs. The optional second input is used for a control image (the output of an Edge node, for example) for the effect.

Erode/Dilate Parameter Tab

Operation

The Operation popup menu enables you to select Erode or Dilate for the operation.

Magnitude

The Magnitude parameter controls the size of the erode or dilate effect. The Magnitude value is expressed as a percentage of the total width of the input image. The range is 0–1.

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, however, 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 Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

GRAIN NODE

The Grain node enables you to simulate the effects of film grain. The Library menu offers presets for several popular film stocks. If desired, however, a number of adjustment parameters are available to provide precise control over the characteristics of the grain added to the input image. This is particularly useful for visually matching grain to different images for a later composite.

The grain node works on the Red, Green, and/or Blue channels by assigning a value to each pixel, based on a Gaussian distribution. These values are then scaled according to the node controls and added back into the input image. The overall visual effect will depend on the parameter specifications and the brightness of the input image.

The principal differences between the Grain node and the Noise node are the library of presets offered in the Grain node, the fact that the RGB channels can be controlled independently in the Grain node, and the way the grain patterns are distributed (Noise uses a constant distribution). In general, a Gaussian distribution will provide a better match for film grain.

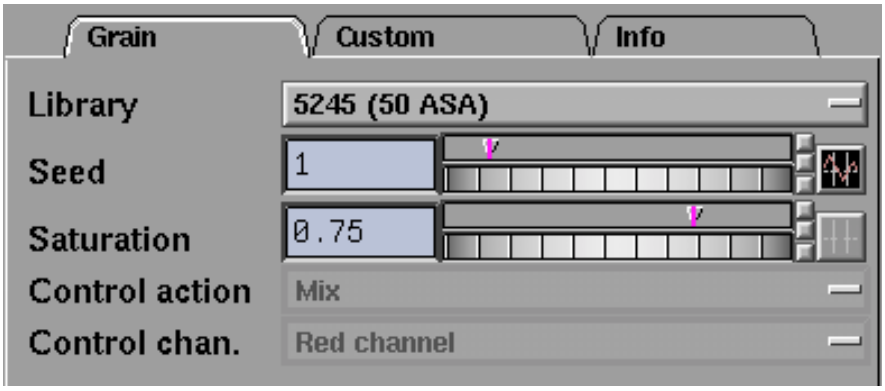
Matching Grain Patterns

Start by selecting a specific type of film stock, if applicable, from the Library menu in the Grain tab. The other Grain node parameters will default to settings optimized for the selected film stock.

If the result is not as exact a match as desired, follow these steps to visually match the node output to an existing grain pattern:

- Open two monitors—one from the Grain node, and one from the node containing the image with the pattern you wish to match. (Or use an A/B wipe to view the Grain node image and the desired match side by side.)
- Use the Size and Amount parameters in the Custom tab to match the grain in the **bright areas**.
- Use the Falloff parameters in the Custom tab to match the grain in the **darker areas**.

Grain Parameter Tab



Library

The Library menu enables you to select preset values optimized for various film stocks, as well as a Custom option to set the parameters manually by matching grain patterns visually:

- 5245 (50 ASA)
- 5248 (100 ASA)
- 5279 (500 ASA)
- 5298 (500 ASA)
- Custom

Seed

The Seed value is used to generate a sequence of random numbers for the grain operation. Varying the Seed parameter will generate a different grain pattern for each frame.

By default, this parameter has been animated using the $F variable—which represents the current frame number—to create a new value for every frame. (Note that the STED button is in its active state, which indicates an animated parameter.) To change this value, check the Expressions box and type in a new expression, or Alt-click the STED button to deactivate it and type in any integer.

Saturation

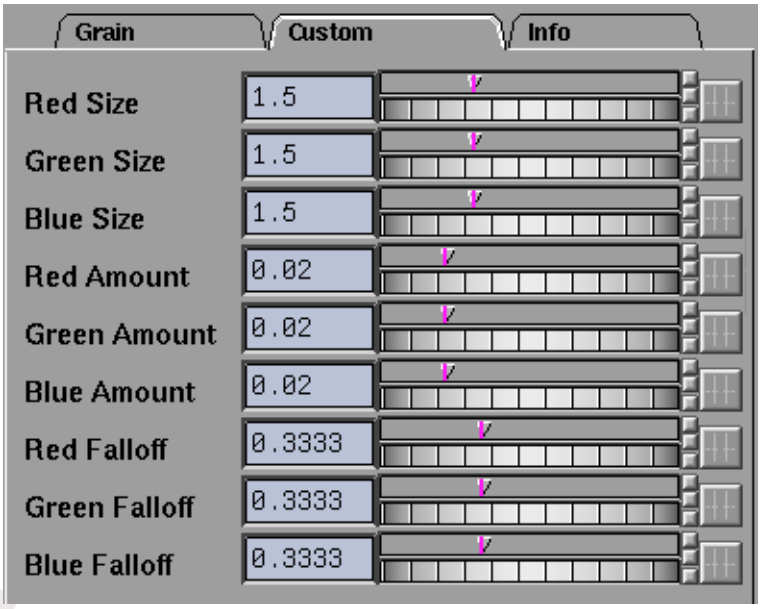
The Saturation parameter controls the color saturation of the grain. A value of 0 will result in monochrome grain and a value of 1 will result in completely independent red, green, and blue grain.

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

Custom Parameter Tab

The Custom tab provides parameters for making precise adjustments to the grain pattern produced. The Size, Amount, and Falloff of the grain can be controlled for each channel (RGB) independently.



Size (Red, Green, Blue)

The Size parameters set the average pixel size of the grain added to the image in the Red, Green, and Blue channels. The precise size of each individual kernel will vary.

Amount (Red, Green, Blue)

The Amount parameters can be used to increase or decrease the apparent amount of grain added to the image in the Red, Green, and Blue channels. Larger values will result in a more substantial addition or subtraction from each pixel in the input image, leading to the appearance of more grain.

Falloff (Red, Green, Blue)

The Falloff parameters enable you to increase or decrease the concentration of grain based on the brightness of the imagery in the Red, Green, and Blue channels. Higher falloff levels reduce the prominence of grain in darker areas of an image and increase the prominence of grain in the brighter areas.

NOISE NODE

The Noise node enables you to add video noise to imagery. It provides you with extensive control over the amount and character of noise you wish to add.

Note:

To create film grain, use the Grain Node instead of the Noise node. The Grain node is optimized for film grain simulation.

Noise Parameter Tab

Noise

You set the Noise parameter by keying in a numerical value between 0 and 1.0 or by using the mouse to set the tuner to the desired value. A value of 0 adds no noise to the image; a value of 1.0 represents maximum noise.

Seed

The Seed value is used to generate a sequence of random numbers for the noise operation. By default, this parameter has been animated using the $F variable—which represents the current frame number—to create a new value for every frame. Varying the Seed parameter will generate a different noise pattern for each frame.

Constant Hue/Sat

Check this parameter to keep hue and saturation constant; only the luminance value changes.

Value Relative

Check this parameter to keep the luminance value of the noise relative to that of the image.

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel to the noise effect by deselecting a channel icon.

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

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. Essentially, the Rank node ranks the pixels in order of importance based on the operation being performed.

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 Width parameter)
- the shape of operation area: box or cross (Filter Coverage parameter)
- how pixels that border the image should be treated (Border parameter)

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.

Note:

To remove film grain, use the Degrain Node instead of Rank. Another option for reducing noise in still imagery, such as a background plate, is the Time Convolve Node.

To remove spot noise, "Threshold Replace (with avg)" is effective. For more information about this option, see the "Threshold (%)" 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.

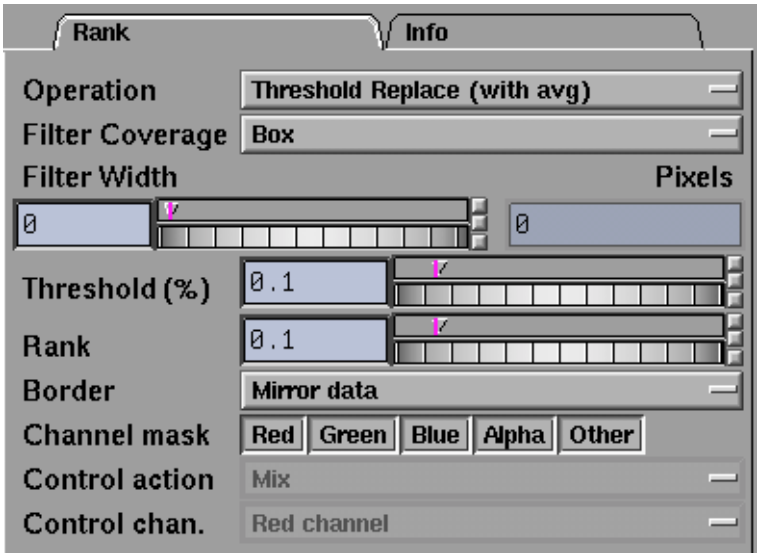
Note:

For the best result with noise reduction filters, specify Cross for the filter shape (see the Filter Coverage 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)." However, if you need to shrink and grow edges with subpixel accuracy, use the Erode/Dilate Node instead.

Rank Parameter Tab



Operation Menu

The Operation parameter enables you to specify the type of operation to be performed by selecting one of the following options from the popup menu:

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 Width parameter.)

Box

This option, which is the default, specifies a square grid of pixels. Box is the recommended option 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. 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 Width/Pixels

The Filter Width parameter enables you to specify the size of the operation area, either as a percentage of the width of the image (using the Filter Width tuner) or as a specific number of pixels (using the associated Pixels data entry field).

You can set the Filter Width tuner by entering a value into the data entry field or by using the mouse to set the tuner to the desired value. When you set the Filter Width tuner, the value in the Pixels field will update accordingly.

When entering specific values into the Pixels field, you must enter an odd number (if you do enter an even number, Chalice automatically adjusts the value to the next lower odd value). A value of 3, for example, specifies a 3 x 3 pixel grid, assuming that the Box option is selected in the Filter Coverage parameter.

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.

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.

The default Threshold value is 0.1, or 10 percent.

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, expressed as a number in the range of 0–1, which you specify in the Rank parameter. For example, if you specify 0.5 in the Rank parameter, the median pixel value in the ranking is assigned as the output pixel value.

Note:

The "Median (Rank 0.5)" filter in the Operation parameter menu is essentially the same operation as when you select the "Specify Pixel Rank" operation and use 0.5 as the Rank parameter value. However, "Median (Rank 0.5)" has been optimized to run faster with the most commonly used Filter Width parameter settings, which are 3 x 3, 5 x 5, and 7 x 7.

Border

When using a neighborhood operator, the pixel currently being modified is in the center of a grid of pixels that define 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 parameter enables you to specify how border pixels should be treated by selecting one of the following options from the Border popup menu:

- **Mirror Data:** This option, which is the default setting, processes border pixels by assigning the values of the inner adjacent pixels in the grid to the corresponding locations in the grid that fall outside the image borders.
- **Use Black:** When this option is selected, a value of 0 is assigned to locations in the grid that fall outside the image borders.

- **Do Not Process:** When this option is selected, border pixels are not processed; that is, their original values are passed through unchanged.

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

SHARPEN NODE

The Sharpen node enables you to enhance the detail in an image, effectively sharpening it.

You can choose from spatial analysis or frequency analysis. The default is spatial analysis, because it is the simpler method. Spatial analysis uses a convolution operation for sharpening.

Note:

If you need more information about using a convolution filter for sharpening, refer to the Convolve Node description.

Frequency analysis, on the other hand, performs several filtering steps and requires that the parameters on the Advanced tab be set appropriately for the image being sharpened.

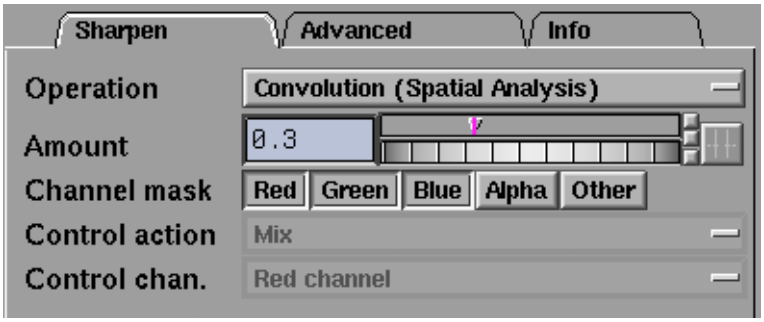
Using Frequency Analysis

The results of Frequency Analysis are well worth the effort of learning the controls involved. Start by reading the descriptions of the parameters on the Advanced tab. Then decide if the image needs prefiltering before sharpening; that is, if the image is especially noisy or grainy.

For the actual sharpen operations, start by setting the Noise Level and Scale parameters. (The Scale parameters are only applicable to film.) These parameters set 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 second part of the sharpen operation is to control how much of the "cleaned up" high frequency signal gets added back into the original image. Open the Sharpen node monitor and adjust the 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.

Sharpen Parameter Tab



Operation

The Operation menu enables you to select the method to use for the sharpen operation:

Spatial Analysis

This is a 3 x 3 convolve kernel that sharpens by assigning greater weight to the value of the central pixel in the convolution than to the surrounding pixels in the kernel.

Frequency Analysis

This method treats the image as a continuous signal of changing intensity. To sharpen the image, the high-frequency part of the signal—which tends to include important image detail, like edges or small features—is separated out, smoothed if possible to remove extra noise, and then added back into the original image.

There are two frequency analysis methods, one optimized for video imagery and one for film. When either is selected, the parameters on the Advanced tab are enabled. These parameters must be set knowledgeably to get the best results, so it is recommended that you review the Advanced tab parameter descriptions below.

Amount

This parameter enables you to specify the amount of sharpening. With either method, lower values produce less sharpening, and higher values, more (from no sharpening at 0 to a very harsh result at 1):

- For spatial analysis, the value you set is used in the convolution kernel.
- For frequency analysis, this value specifies the amount of high-frequency detail to be added back into the original image.

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

Control Image Parameters

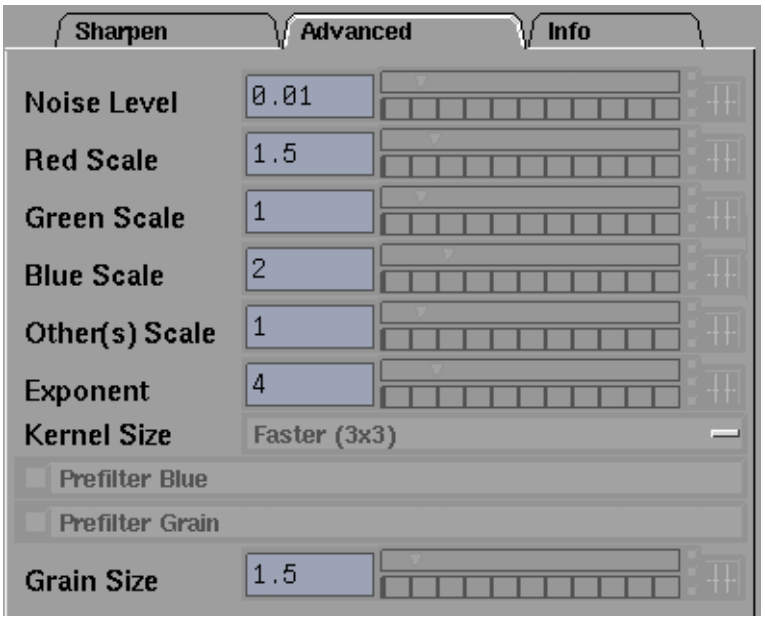
The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

Advanced Parameter Tab

The parameters on the Advanced tab become available when a frequency analysis option is selected from the Operation menu on the Sharpen tab.

Note:

Enabling the prefiltering parameters is optional and they may be used singly or together. In either case, any prefiltering is done before the sharpen operation.



Noise Level and Scale Parameters

These parameters enable you to set a threshold for noise and grain filtering of the high frequency signal before it is added back into the original image. The Noise Level parameter ap-

plies to all channels equally, whereas the Scale parameters are used to weight the influence of the noise level value differently for each channel:

Noise Level

Large 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.

Scale (Red, Green, Blue, Other)

For film grain analysis, the Scale parameters enable you to adjust 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 values are a good starting point, as they reflect a ratio typical for many film stocks. (For video, however, the Scale controls are not active, because video signals do not exhibit noise differences between the red, green, and blue components.)

Exponent

The Exponent parameter, in conjunction with the Amount parameter on the Sharpen tab, modifies how much of the high frequency signal falling above the threshold levels (set in the Noise Level and Scale parameters) will be added back into the original image.

The Exponent differs from the 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 Amount tab) 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.

Kernel Size

The Kernel Size menu is used to specify the size of the convolution kernel used by a blur operation that is performed initially to help identify the high frequency component of the signal. (The high frequency component is the difference between the original image and the blurred image.)

Prefiltering Parameters

The Prefiltering parameters give you the option to specify that a prefiltering operation be done before the actual sharpen operations:

Prefilter Blue

In film, the blue component is subject to more noise than either red or green. Therefore, you can check the Prefilter Blue option to instruct Chalice to prefilter the blue channel only, before sharpening. Prefilter Blue is a basic 3 x 3 convolution similar to a blur.

Prefilter Grain

The Prefilter Grain option performs grain filtering on all three image channels before sharpening, using the same frequency analysis method performed in the Degrain node.

Grain Size

If Prefilter Grain is checked, the Grain Size parameter becomes active. It enables you to control how much of the new, degrained signal is added back into the original image during the Prefilter Grain operation. Higher values enhance detail up to a point; beyond that, however, the image will begin to look harsh.

UNSHARP MASK NODE

The Unsharp Mask node sharpens an image by increasing the contrast between adjacent pixels in those areas with the greatest visual detail.

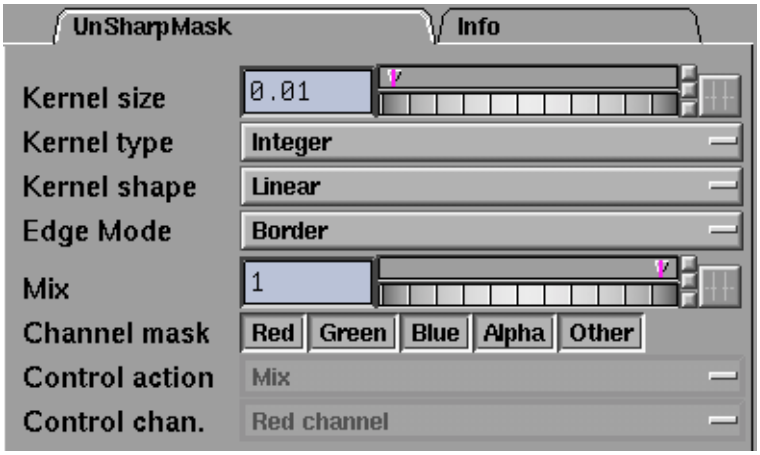
The process involves three steps:

- First, a low-pass convolution filter is used to blur the source image.
- Second, 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").
- Third, the result of the unsharp mask operation is added to the source image.

The convolution used for the blur modifies each pixel in the image, in turn, based on the values of surrounding pixels. The matrix that delineates this pixel area is known as a kernel.

The Unsharp Mask node provides parameters that enable you to define the kernel attributes and thus control the blur. You can control the contribution of the unsharp mask to the source image by using the Mix parameter.

Unsharp Mask Parameter Tab



Kernel Size

The Kernel Size parameter enables you to specify the number of adjacent pixels whose values will be used to modify the current pixel. The size parameter is expressed as a percentage of the total number of pixels in the image.

You set this parameter by keying a value in the range of 0 to 1 into the data entry field or by using the mouse to set the tuner to the desired value. A value of 0 would leave the image unmodified by the convolution. A value of 1 would specify a kernel the size of the image, so that every pixel in the image would contribute to the modification of the current pixel. In most cases you will select values within a narrow range of the low end of the scale.

Kernel Type

The Kernel Type parameter features a popup menu that allows you to choose whether Chalice should use Integer or Floating Point math for the calculation. Integer is the default, unless you choose Cubic for the Kernel Shape parameter, in which case the Kernel Type parameter defaults to Floating Point. Integer precision restraints make it inefficient to compute the Cubic function using integer math.

Kernel Shape

The Kernel Shape parameter enables you to determine the extent to which each adjacent pixel in the kernel will contribute to the convolution by choosing a function that describes the distribution curve used to weight the sample.

You can select one of the following functions from the Kernel Shape popup menu: Constant, Linear, Quadratic, Cubic, or Gaussian.

The Constant function, for example, is equivalent to a box filter and applies the same weight to every cell (pixel) value in the kernel when it is factored. The weighting distribution becomes more complex as you select the other functions: Linear is equivalent to a triangle filter, and so on, with Gaussian being the most complex option and therefore the most computationally intensive.

The optimal choice will depend on the nature of the source image, the effect you wish to achieve, and how much time you are willing to devote to processing.

Edge Mode

By the nature of this convolve operation, the pixel currently being modified is in the center of the cell matrix that composes the kernel. Therefore, when convolving a pixel that borders the image (or when the kernel is large), there will not be adjacent pixels on all sides to contribute values to the convolution.

The Edge Mode parameter enables you to assign pixel values to these "empty" kernel cells by selecting one of the following options from a popup menu:

- **Border:** This option, which is the default setting, assigns the values of the edge pixels to the kernel cells that fall outside the image borders.
- **Black:** This option assigns a value of 0 to kernel cells outside the image borders.
- **Mirror:** This option assigns the values of the inner adjacent pixels to the corresponding kernel cells that fall outside the image borders.

Mix

The Mix parameter enables you to specify the extent to which the source image is modified by the unsharp mask by keying a value in the range of 0 to 1 into the data entry field or by using the mouse to set the tuner to the desired value.

The range represents the percentage of the unsharp mask you wish to be added to the source image. A value of 0 would result in no modification. A value of 1 would add 100 percent of the value of the mask to the source image.

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation, as explained in "Using Control Images with Filter Nodes."

19 EFFECTS NODES

The Effects menu comprises "special effects" style operators. Loosely defined, this category includes operations such as embossing and posterization, as opposed to core operations related to compositing such as color correction or edge detection.

The first group of Effects menu nodes described here are related functionally—they all contribute to embossing, rippling, and other distortion type effects. The **Surface Normal** and **Bump** nodes both create two-channel floating point outputs that are then used by other nodes, including the **Emboss** and **Vector Warp** nodes, which require a two-channel float image for the second input.

Bump Node ..... p. 19.2
Surface Normal Node ..... p. 19.3
Emboss Node ..... p. 19.4
Vector Warp Node ..... p. 19.6

The other Effects menu nodes include **Posterize**, which reduces colors to an indexed palette; **Time Convolve**, which averages pixels from different frames rather than pixels in an area of the same frame, as other convolve operations do; **Vector Blur**, which is often used for motion blurs; and Xpresso.

The **Xpresso** node is perhaps the most versatile node in Chalice. To use Xpresso, you enter expressions into a text field, which enables you to essentially design your own filter, composite, or other operator. The Xpresso node could have been placed with equal logic in several other menus, including the Tools menu and the Composite menu. Xpresso is a multi-input node; like Mcomp, every time you add an input to Xpresso a new connector is created.

Posterize Node ..... p. 19.7
Time Convolve Node ..... p. 19.8
Vector Blur Node ..... p. 19.11
Xpresso Node ..... p. 19.15

BUMP NODE

The Bump node enables you to create a bumpmap that you can use for embossing or other rendering effects. A bumpmap is a pseudo-3D image that contains a surface normal (a vector perpendicular to the surface) for each pixel of a selected image channel. During the Bump computations, the surface normal of each pixel is encoded as two floating point channels:

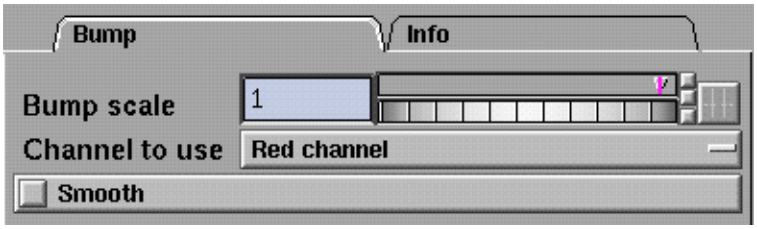
- a float value for the X component of the normal in the red channel
- a float value for the Y component of the normal in the green channel

The value of the normal is based on the rate of change between the current pixel and neighboring pixels of the selected plane.

The Bump node accepts one input, which can have from one to 10 channels. For each pixel the Bump node examines the values of adjacent pixels on a single selected image plane (channel) and creates the bumpmap. You can apply the bumpmap perturbation

- over another image using the Emboss node to create an embossed effect,
- over the source image using the Emboss node to give the image an appearance of greater dimensionality, or
- over another image using the Vector Warp node to create a displacement effect such as a ripple or other distortion.

Bump Parameter Tab



Bump Scale

The Bump Scale parameter enables you to specify the degree of relief of the bumpmap by keying in a numerical value or by using the mouse to set the tuner to the desired value. The higher the value, the greater the effect.

Channel to Use

The Channel to Use parameter enables you to specify which channel of the source image the Bump node will use to calculate the surface normal for each pixel. You set this parameter by selecting a channel from the popup menu.

Smooth

Check the Smooth parameter box to soften the bumpmap's edge transitions to create a softer slope or more "rounded" effect when compositing. The Bump node creates the smoothing by taking an average of the pixels surrounding the current pixel on the left and below it before calculating the bump for the current pixel.

SURFACE NORMAL NODE

The Surface Normal node creates an image that can be used as a bumpmap, a displacement map, etc., to control the perturbation of an image in another node, such as the Vector Warp node.

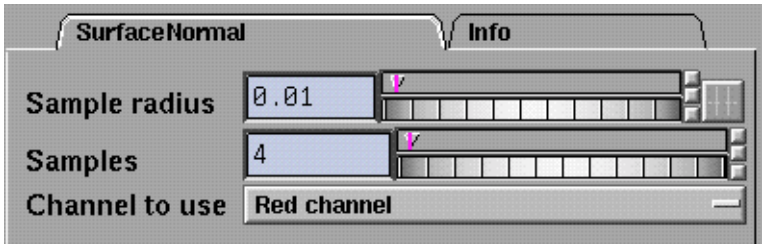
The Surface Normal node uses a single, user-selectable channel of the input image. A vector perpendicular to the surface is calculated for each pixel of the selected channel by taking a weighted average of a sample of surrounding pixels.

The 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.

Note:

The Surface Normal node is similar to the Bump node, but Surface Normal offers more control over the output by enabling you to specify the number of surrounding pixels to sample and the radius of the sample area (refer to the parameter descriptions below).

Surface Normal Parameter Tab



Sample Radius

Pixels are sampled along the circumference of a circle surrounding the pixel currently being processed. The Sample Radius parameter enables you to set the size of the radius that defines this circle. The radius size is expressed as a percentage of the total image size.

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. You can key a value into the data entry field or use the mouse to set the tuner to the desired value.

Samples

The Samples parameter enables you to specify the number of surrounding pixels to be sampled. The minimum is 4. Larger sample values take longer to process but produce a more accurate result. You can key a value into the data entry field or use the mouse to set the tuner to the desired value.

Channel to Use

This parameter enables you to specify which channel of the input image should be used to calculate the surface normal for each pixel by selecting the desired channel from the Channel to Use popup menu.

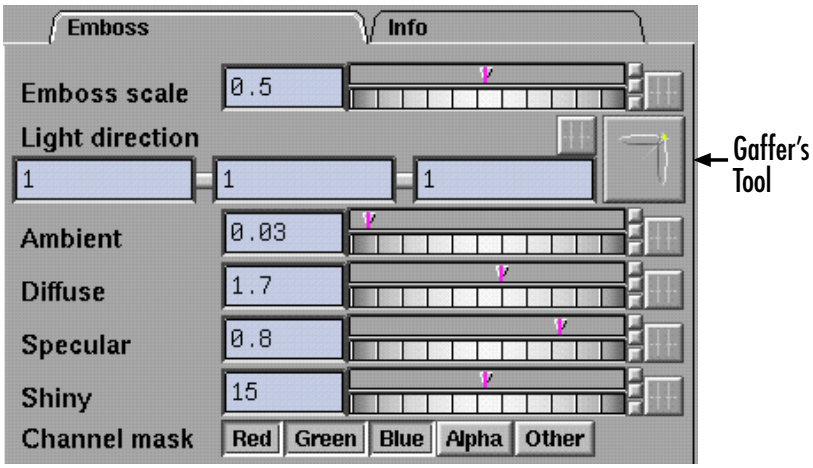
EMBOSS NODE

The Emboss node embosses the primary (top) image input with a bumpmap (bottom input), in effect raising in relief certain graphic elements of the primary image. A bumpmap is a pseudo-3D image that contains a perturbation of the surface normal for each pixel.

The Emboss nodes takes two inputs:

- The primary image can have any number of 8-bit, 16-bit, or floating point channels.
- The bumpmap input must be a two-channel floating point image such as that created by the Bump or Surface Normal node.

Emboss Parameter Tab



Emboss Scale

The Emboss scale parameter controls the depth of the embossing. As you increase or decrease this value, the Emboss node lengthens or shortens the vectors of the bumpmap within the following range:

1 = maximum embossing
0 = no embossing
-1 = maximum negative embossing

Light Direction

The Light Direction parameter provides you with two ways of controlling the lighting direction in three dimensions around the image:

- You can key specific *x,y,z* coordinates into the three Light Direction boxes.
- You can use the Gaffer's tool icon to control the light source. Click and drag the tool to "move" the light source. The *x,y,z* coordinates in the Light Direction boxes will update accordingly.

Tip:

You can create a negative emboss effect simply by changing the lighting direction (without adjusting the Emboss scale parameter).

Ambient

The Ambient parameter controls the non-directional or surrounding illumination. You set this parameter by keying in a numerical value or by using the mouse to set the tuner to the desired value within the range of 0 to 1.

Diffuse

The Diffuse parameter controls the level of reflected light. You set this parameter by keying in a numerical value or by using the mouse to set the tuner to the desired value within the range of 0 to 3.

Specular

The Specular parameter controls bright highlights from shiny surfaces, such as brilliant reflections on metallic objects. You set this parameter by keying in a numerical value or by using the mouse to set the tuner to a value within the range of 0 to 1.

Shiny

The Shiny parameter represents the specular exponent and controls the size of specular highlights. You set this parameter by keying in a numerical value or by using the mouse to set the tuner to the desired value within the range of 0 to 30.

Channel Mask

The Channel mask parameter enables you to inhibit the modification of any channel in the primary input by deselecting a channel icon. By default, the Red, Green, and Blue channels are active; the Alpha and Other channels are masked.

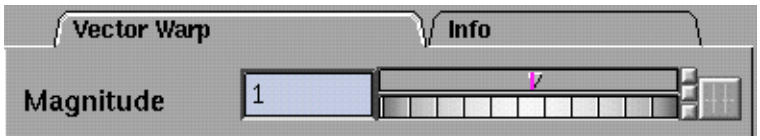
VECTOR WARP NODE

The Vector Warp node enables you to 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 bottom input must be a two-channel floating point image, such as the output produced by the Surface Normal node. The top input, on the other hand, may be one or more 8-bit, 16-bit, or floating point channels.

The vector data in the bottom input is used to control the displacement of corresponding pixels in the top input image. The output image of the Vector Warp node will be the same number of channels as the top input image.

Vector Warp Parameter Tab



Magnitude

The Magnitude parameter enables you to adjust the magnitude of the effect by specifying a multiplier value for the vector data in the bottom input. You set the Magnitude parameter by keying a value in the range of -20 to 20 into the data entry field or by using the mouse to set the tuner to the desired value. The default value is 1, which represents no change from the input image.

Larger positive and negative values both result in more pronounced distortion. Negative values, however, reverse the direction of the vector to create warping in the opposite direction of positive values.

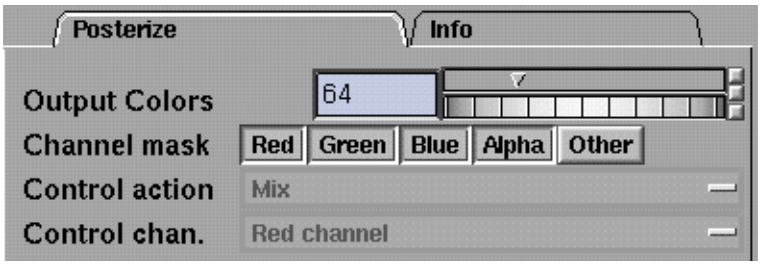
POSTERIZE NODE

The Posterize node remaps the pixels of an input image to a reduced palette of colors to create large areas of solid color.

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.

You can specify the number of colors per channel to use, as well as which channels of the input to modify. The Posterize node accepts 8- or 16-bit linear or floating point data. (For floats, the Posterize node allows a maximum of 65536 colors.)

Posterize Parameter Tab



Output Colors

This parameter enables you to specify the number of colors per channel to use for the output. The default is 16 colors per channel for 8-bit inputs and 4096 for 16-bit.

You can specify any value in the range of 1–256, for 8-bit images, or 1–65536, for 16-bit and floating point images, by keying a number into the data entry field or by using the mouse to set the tuner to the desired value.

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

Control Image Parameters

The Control Action and Control Channel menus enable you to specify how an optional control image input will govern the node operation.

The **Control Channel menu** enables you to specify which channel in the control image will be used to govern the control action.

The **Control Action menu** lets you select a control method: On/Off or Mix. When the On/Off method is selected any control pixel with a nonzero value will "turn on" filtering of the corresponding pixel in the primary input. The Mix method, on the other hand, uses the specific value of each control pixel to determine the extent to which the primary pixels will be affected by the node operation.

If you need more information, refer to Using Control Images with Filter Nodes at the beginning of chapter 18.

TIME CONVOLVE NODE

The Time Convolve node enables you to modify each pixel in an image by averaging the values of corresponding pixels across a range of frames. Time Convolve only affects those pixels that change from frame to frame.

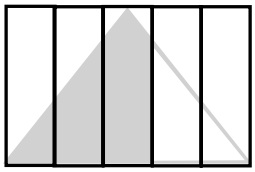
Time Convolve can be used to create various trailing or strobe effects. Depending on the nature of the movement in a shot, this node can also create an effective motion blur because it blurs moving elements without blurring the background. This characteristic means that Time Convolve is excellent for noise reduction on background images (images without moving elements).

You can specify the following options for the Time Convolve operation:

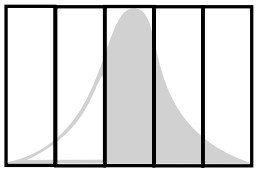
- the width of the filter, in frames (how many frames will contribute to the operation for each frame convolved)
- the shape of the filter (the type of function curve, which specifies how the values will be weighted)
- the direction (from current frame) in which the filter will operate
- an offset from the current frame (so that the frame being modified is no longer the central frame of the filter)

The Time Convolve node panel displays graphically how the current settings affect the filter:

Filter Width: 5
Filter: Triangle
Direction: Past Only



Filter Width: 5
Filter: Gaussian
Direction: Future Only



You may want to experiment with different settings to get a feel for the effect they will have on your imagery. For example, using the Box filter, so that each frame in the operation contributes equally to the result, can produce a strobelike or a banding effect.

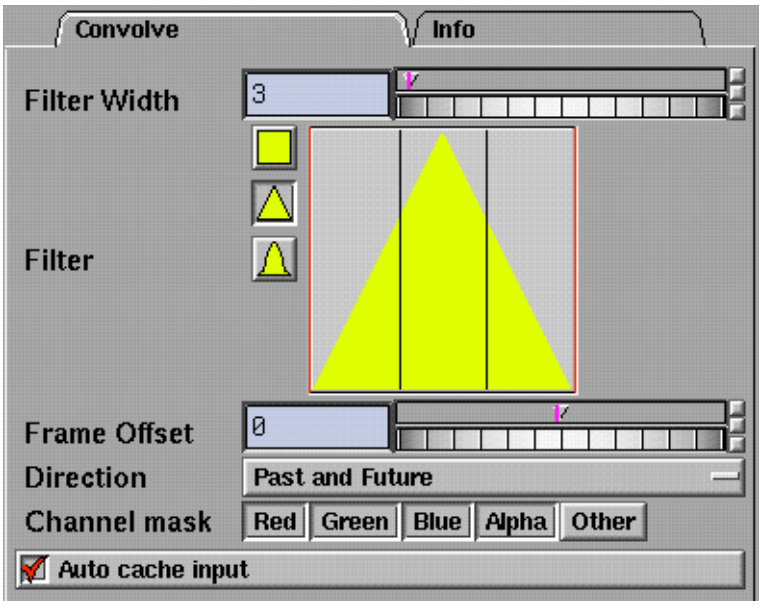
On the other hand, using the Triangle filter with the Past Only setting in the Direction parameter can create a diminishing trail behind the moving subject.

Note:

Due to the nature of the time convolve operation, you may encounter a problem rendering if the Time Convolve is part of a flow that connects to a multilayer-input node (such as Mcomp) downstream. This problem occurs only if the background (top) input to the Mcomp node is a Color node.

The solution is to insert a Sequence node between the background input and the Mcomp to "hold" the single frame until it matches the length of the other inputs.

Convolve Parameter Tab



Filter Width

The Filter Width parameter enables you to specify the number of segments into which the filter is divided, where each segment is a frame of the input image. The default value is 3, which is the minimum size.

You can enter any odd integer value in the range of 3–31. (If you enter any other value, it will be automatically replaced by the nearest allowable value.) 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 Filter Width 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.

Start and End Frames of the Input Sequence

When the **first frame** of the input image 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.

Filter

The Filter parameter enables you to select one of three function curves, each of which applies a different weight to the distribution of the effect:

- **Box filter:** The box filter (constant function) averages the values of each frame evenly, so that each contributes in equal measure to the result.

- **Triangle filter:** The triangle filter (linear function) performs a weighted average of the values in each frame, with the center frame contributing most to the result.
- **Gaussian filter:** The Gaussian filter uses a bell-shaped function curve to determine the contribution of each frame to the result.

Frame Offset

This parameter allows you to specify an offset from the center of the filter. If you set a Frame Offset value other than 0, the current frame (the frame being modified) is no longer the center segment of the filter.

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:

Valid Offset Values

The range of valid Frame Offset values is tied to the Filter Width parameter value, and a value of 0, which specifies no offset, is always the center value of the offset range. At the default Filter Width value of 3, for example, you can only specify a Frame Offset of -1 or 1. At a Filter Width value of 5, the Frame Offset range is -2 to 2, and so on.

Direction

The Direction parameter enables you to specify that only part of the filter be used in the convolve operation. The Direction popup 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 from the current frame.

You can see the effect of each setting on the filter in the graphic display in the node panel. (Two examples are illustrated in the introductory section of this node description.)

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by deselecting the Red, Green, Blue, Alpha, or Other channel icons.

Auto Cache Input

When checked, the Auto Cache Input option affects the cache setting of whatever node is serving as the input node to the Time Convolve node. The Auto Cache Input option resets the frame cache of the input node to match the Filter Width setting in Time Convolve.

Auto Caching will greatly improve the speed of rendering; however it requires substantial storage space (in RAM or on disk) to cache the needed input frames.

VECTOR BLUR NODE

The Vector Blur node enables you to specify precise values to control the blurring of an image. The Vector Blur node 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 Convolve Node description in chapter 18.

To control 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.

How Border Pixels Are Treated:

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.

You can control the following, using the indicated parameter on the Vector Blur node panel to do so:

- the length of the vector, which affects the overall magnitude of the blur ("Magnitude" parameter)
- the angle of the vector, ("Direction" parameter)
- the type of distribution curve used to weight the contribution of each cell in the kernel ("Filter" parameter)
- the portion of the curve used in the computation ("Extents" parameter)

Using a Control Image with Vector Blur

The Vector Blur node accepts one or two inputs. The second, optional input is used as a control image for the blur. The value of each pixel in the control image determines how the corresponding pixel (the pixel with the same *x,y* coordinates) in the primary input image will be processed by the Vector Blur node.

Note:

The control input must be a one-channel image if the "on/off" control method is specified or a two-channel floating point image if the "direction" control method is specified.

On/Off

The On/Off setting turns the blur operation on and off at each pixel in the primary image. The control image used for On/Off must be a one-channel image of the same resolution as the primary image input.

If a pixel in the control image has a value of 0 (i.e., is black), the corresponding pixel in the primary image will not be processed, and the original value of the pixel in the primary image will be passed through unchanged. However, if a pixel in the control image has a value other than 0, the corresponding pixel will be processed.

Direction

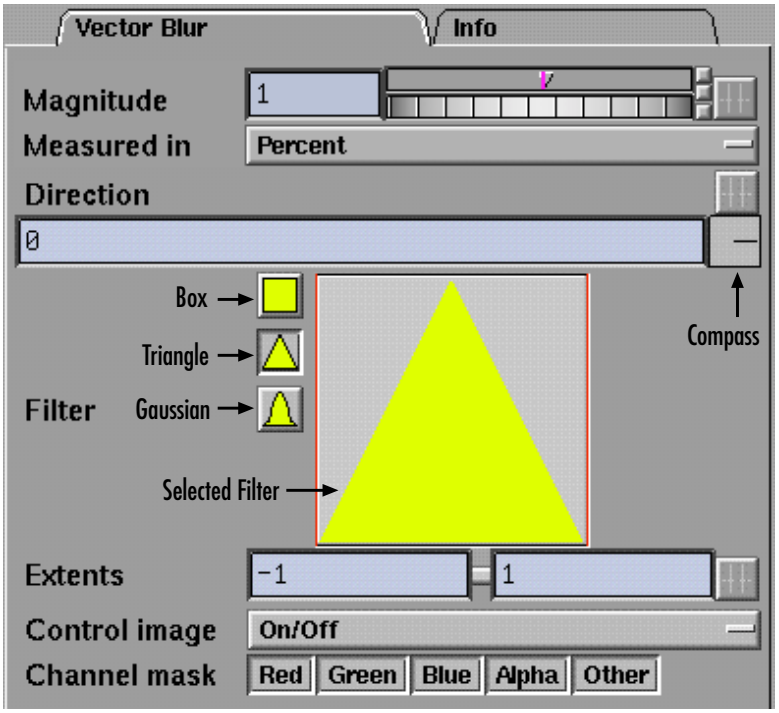
The Direction setting uses the control image to determine the direction and magnitude of the blur operation at each pixel in the primary image. The control image used for Direction must be a two-channel floating point input.

Tip:

Use the Bump or Surface Normal nodes to create such a control image.

The two channel values in the control image represent the *x* and *y* coordinates of a vector. Chalice uses this vector to calculate the direction and magnitude that will be used to blur the corresponding pixel in the primary image.

Vector Blur Parameter Tab



Magnitude

The Magnitude parameter enables you to specify the length of the vector used to define the blur. You set this parameter to specify the general level of blur, with higher values resulting in a greater blur effect. The greater the magnitude, the longer the blur will take to process.

For example, if the Magnitude is 6 and you are working in pixels, the total length of the vector would be 7 pixels (the pixel to be blurred occupies the center position in the row). Working in percentages (see the "Measured In" parameter description below), a Magnitude value of 6 would represent a vector measuring 6 percent of the total width of the image. There is no benefit to using negative values in this parameter; that is, a value of -6 is equivalent to a value of 6.

Measured In

The Work In parameter is used to specify whether the Magnitude value will represent number of pixels or a percentage of the total width of the input image.

Direction

The Direction 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 can set the Direction parameter by clicking the directional line on the compass icon and dragging it counterclockwise to rotate it to the desired angle. There is no benefit to rotating the compass icon in excess of 360°. For example, a setting of 380° will generate the same result as a setting of 20° (all other things being equal).

You can also set the Direction parameter manually by keying in a numerical value in the range of 0–360 degrees. The default value is 0, which equates to a horizontal left-to-right direction.

Filter

The Filter parameter enables you to select one of three function curves, each of which applies a different weight to the distribution of the blur effect:

- **Box filter:** The box filter averages the values of each pixel in the kernel, so that each pixel contributes in equal measure to the blur effect.
- **Triangle filter:** The triangle filter performs a weighted average of the pixel values, with the central pixels contributing more to the result.
- **Gaussian filter:** The Gaussian filter uses a bell-shaped curve to determine the contribution of each pixel in the kernel to the blur.

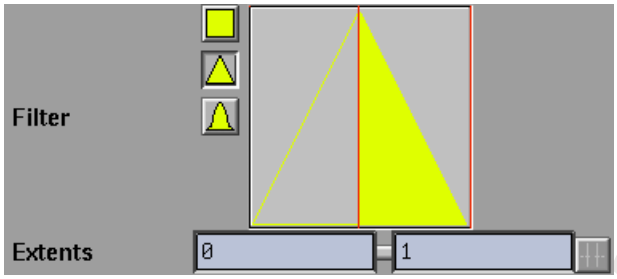
Extents

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 using the Extents parameter. To use the Extents parameter, you key a pair of values between -1 and 1 into the text entry fields:

- 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 might set the Extents parameters to 0 and 1.

Using a horizontal vector as an example, keying a value of 0 into the left text entry field and 1 into the right text entry field would eliminate the contribution of all pixels in the kernel that fall to the left side of the central pixel, as in the following illustration:



Extents Values in Relation to Direction 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.

If Direction value (in degrees) is...	Then Extents value of -1 equals...	Then Extents value of 1 equals...
0	**left** edge of vector	**right** edge of vector
90	**bottom** edge of vector	**top** edge of vector
180	**right** edge of vector	**left** edge of vector
270	**top** edge of vector	**bottom** edge of vector

Channel Mask

The Channel Mask parameter enables you to inhibit the modification of any channel by de-selecting the Red, Green, Blue, Alpha, or Other channel icons.

Control Image

This menu enables you to specify whether the On/Off or Direction method will be used for the control image, as explained in the earlier section on Using a Control Image with Vector Blur.

XPRESSO NODE

The Xpresso node enables you to program custom effects. It allows you to write mathematical expressions that define each pixel in the output image as a function of the input imagery being delivered to the Xpresso node.

The Xpresso node can have a single or multiple inputs. Each time you add an input to the Xpresso node, another input connector is automatically created. All inputs to an Xpresso node must be consistent in:

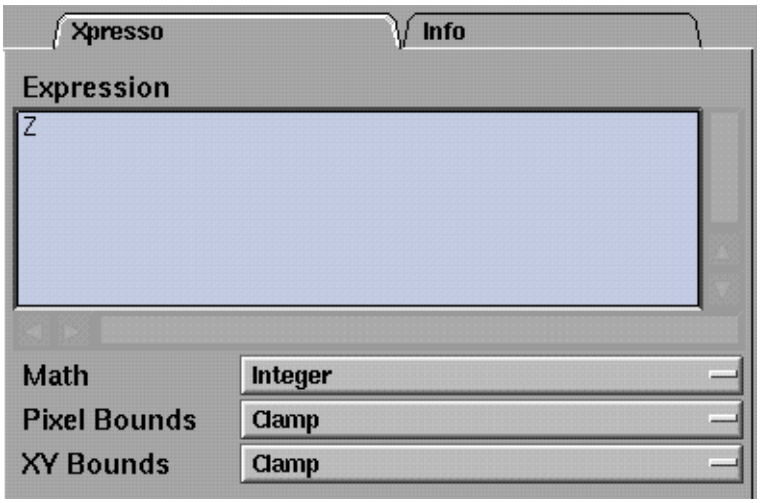
- bit depth (8-bit; 16-bit; or floating point)
- resolution (size in pixel dimensions)
- number of channels

The input to an Xpresso node can have any number of channels, as long as the source image includes all channels named in the expression used.

When you use the Xpresso node to calculate an output image, all pixels and channels of the input imagery are available to you for inclusion in the mathematical expressions you devise.

Xpresso Parameter Tab

The Xpresso node parameter tab contains an Expression field in which you can enter a mathematical expression and several parameter menus.



Expression Field

This is the field in which you enter a mathematical expression. For long, complex expressions, you may prefer to use a text editor instead. To do so, click the cursor in the Expression field and press the Alt-e key combination. The contents of the field will be displayed in the window of the default text editor for your system (i.e., Jot or Notepad).

Math Menu

For an input image consisting of integer data (i.e., an 8-bit or 16-bit source image) the Math menu enables you to specify whether the expression is calculated in integer or converted to floating point. (Floating point pixels are always computed as float.)

When you specify "Floating Point," the Xpresso node

- converts the integer data to floating point data (i.e., from the range of 0–255 for 8-bit imagery or 0–65535 for 16-bit imagery, to 0–1),
- performs the calculations you specified in the Expression field in floating point, and
- converts the result back to integer data for output.

Pixel Bounds Menu

The Pixel Bounds menu enables you to specify whether the Xpresso node should use Clamping or Modulo range delimiting methodologies when computing pixel values.

- **Clamp** can be used with integer and floating point input data. It "clamps" or delimits values at the maximum or minimum of a specified range. For example, for 8-bit color imagery, which has a range of 0–255, values will be clamped at 0 when the result is zero or less, and at 255 when the result is 255 or greater (floating point data will clamp at 0 and 1).
- **Modulo** can be used with integer input data only. It rolls over values when they reach the maximum of a specified range. For example, for 8-bit color imagery with its range of 0–255, a result of 256 will roll over to 0, 257 will be equal to 1, 258 will be equal to 2, and so on until 512 = 0, 513 = 1, etc.

XY Bounds Menu

The XY Bounds menu enables you to specify whether the Xpresso node should use Clamping or Modulo range delimiting methodologies when computing pixel coordinates. (See definitions of Clamp and Modulo, above.)

Y Increases

This menu enables you to specify the orientation of Xpresso. The default, "Y increases from bottom up," specifies that the 0,0 *x,y* coordinate is the lower left corner of the image. This is consistent with the orientation of all other Chalice monitors.

The other option, "Y increases from top down," puts the 0,0 point in the upper left corner. This option is provided for compatibility with previous versions of Chalice. When Chalice converts a 1.6 grail file containing an Xpresso node to version 2.0, this menu will be reset to "Y increases from top down," to match the previous behavior of Xpresso.

How the Xpresso Node Works

The Xpresso node functions by enabling you to specify the value of each pixel in each channel in the output image.

The Xpresso node processing begins at the first pixel of the output image plane 0 and continues until all pixels in the channel have been addressed and assigned a value. Then the Xpresso node addresses the subsequent channel of the output image, and so on until each pixel in each channel has been assigned a value.

You define this process of assigning values to pixels by keying a mathematical expression into the Expression data entry field in the Xpresso parameter tab.

Be sure to review the Guidelines for Using Xpresso and the examples that follow for more information.

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
R	the maximum radius from the center of the image
r	the current radius
A	360 degrees
a	the current angle (as specified from the right horizontal with positive counterclockwise, expressed in degrees)
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).
C	the number of channels
c	the current channel

Global Variables

The following globals are available to the Xpresso node:

$F	the current frame number
$T	the current time, in seconds
$MYGLOBAL	user-defined global

The $MYGLOBAL variable represents a global you create in the Chalice Globals window, which is described in Setting Global Variables in chapter 6.

Chalice Expression Language Functions

The following expression language functions are available in the Xpresso node (See appendix B, “Chalice Expression Language,” for more information about how to use these functions):

abs	acos	and	asin
atan	atan2	between	chf
cht	clamp	cos	deg
even	hypot	if	log
max	min	odd	pow
pulse	rad	rand	sign
sin	sqrt	tan	wrap

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. In this list, the order of precedence is from last to first.

conditional	exp ? exp : exp
logical or & and	|| &&
or	|
and	&
xor	^
equalities	==, !=, >, <, >=, <=
shifting	>> and << (this does not work on floats)
arithmetic	+, -, *, /, %, ** (e.g., base expression**exponent expression)
transcendental	hypot(exp,exp) atan(exp,exp), or atan(x,y) = atan(x/y) sqrt(exp) cos(exp), sin(exp) abs(exp) log(exp) natural rand() no argument, in integer it produces 0 to Z, in float it produces 0 to 1
unary	!, ~, -

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

Assume that there is one input to the Xpresso node. It occupies the number 1 buffer, which we will call [1]. The output image is specified by the zero buffer, or [0].

To set the red, green, and blue (RGB) channels so that the output image is equal to the input image, we could state [0] = [1] for each channel.

However, certain rules governing implied behavior that apply when addressing pixels enable us to simplify the resultant expression. Since setting the output buffer is one such implied behavior, another way to express this would be simply [1].

The following are a few guidelines (see also the examples that follow this list):

- You can reference an entire image by specifying only the image number (e.g., [1], which represents the first input image).
- You can reference a specific pixel at address x,y on channel c in image 1 by specifying [1,x,y,c].
- The expression [1,x,y] enables you to define the channel implicitly.

- The brackets do not need to contain a number; they may instead contain an expression.
- You can use more than one expression to define the output buffer. Expressions must be separated by a semicolon (;).
- The full form for addressing pixels is: [image number,x,y,c] or {image number,r,a,c}.
  - "Image number" refers to either the output buffer [0] or one of the input buffers ([1] to [n]).
  - The first instance, in brackets [], references the x and y address of the pixel, followed by the channel number.
  - The second instance, in braces {}, replaces the Cartesian coordinates x and y with the polar coordinates for current radius (r) and current angle (a).

Example 1

Let's say that we want all the red channel output data to be 10 units higher than the red channel input data, and we wish to make no change to the blue and green channel data. The expressions required to define this process would be:

```
[1]; [0,x,y,0] = [1,x,y,0] + 10
```

In other words, set the output image buffer to equal the input image; then check the red channel of the current input pixel and make the red channel of the current output pixel equal the input value plus 10.

Example 2

Let's say that we want to make the output contain black everywhere except where x=10, at which point the pixels should be yellow. The required expression would be:

```
[0] = 0; [0,x,y,0] = ((x==10) ? Z:0); [0,x,y,1] = ((x==10) ? Z:0)
```

In other words, set the entire output buffer to 0 (black); then check to see if the current x is equal to 10, and if so, then set the red channel to Z; and repeat this step for the green channel.

Example 3

In this example, the Xpresso node takes two inputs. Assume we want to make the top half of the output image display the input 1 image, and the bottom half display the input 2 image. Use this expression:

```
(y > (Y/2)) ? [2] : [1]
```

In other words, check to see if the current y (height) is greater than half of Y (maximum height), and if so, set current pixel to input 2; if not, set it to input 1.

Example 4

In this example, the Xpresso node takes two inputs. Assume we want to perform an operation similar to an "inside" composite and make the output image black (transparent) everywhere except where input 2 has some alpha channel (c=3) coverage, at which points we want to display the input 1 image:

```
[0] = [2,x,y,3] != 0 ? [1] : 0
```

In other words, check to see if the current alpha of input 2 is nonzero, and if so, set to input 1; if not, set to 0 (black).

Example 5: Pattern Generators

Here are several patterns you can generate easily in the Xpresso node for testing and other purposes. The input image for these patterns is required only to set the size of the output.

Tip:

It is a good idea to create and use a color node, for example, rather than an image that is connected to a large network, so that the all the upstream nodes do not need to be processed.

Grid Pattern

This example creates a square grid on tens (you can change the value to vary the size of the grid):

```
x % 10 == 0 || y % 10 == 0 ? Z : 0
```

Checkerboard

You must use the Integer setting in the Math menu for this example, which creates a checkerboard (you can change the values to make the squares larger or smaller):

```
(x/20) % 2 - (y/20) % 2 ? Z : 0
```

Zone Plate

This example will create a moire-type pattern. For this pattern you must use the Float setting in the Math menu.

```
sin ((360 * x * y) / X) * 0.5 + 0.5
```

Example 6: Alpha Max

This expression combines the alpha channels of two inputs, using the higher value of the two for the output.

```
[1]; [0,x,y,3] = [1,x,y,3] > [2,x,y,3] ? [1,x,y,3] : [2,x,y,3]
```

To get the "alpha min" of two inputs, replace the greater than (>) operator with the less than (<) operator in the above expression.

Example 7: Flip X

This expression will flip an image along the X axis (to create a mirror image):

```
[0,x,y] = [1, X - (x + 1), y]
```

To flip an image along the Y axis (to turn it upside down), use this variation on the above expression:

```
[0,x,y] = [1, x, Y - (y + 1)]
```

Tip:

Use the "Save As" command in the Presets menu in the Xpresso node panel to save the current expression under a unique name, which will then appear in the Presets menu of every Xpresso node.

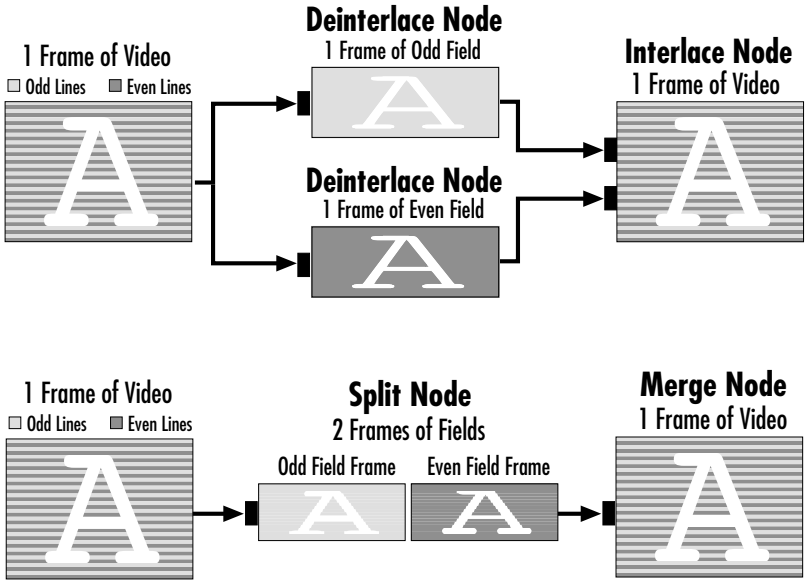
You can create a library of presets that you can select in any Xpresso node, without retyping the expression each time.

20 VIDEO NODES

The Video nodes enable you to create and modify interlaced video frames and fields and to transfer film to and from video frame rates. The **Video Safe** node modifies the colorspace of the input to conform to broadcast standards.

The 3:2 nodes 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 Interlace, Deinterlace, Merge, and Split nodes are field operators, in which frames are separated into fields or merged together into complete frames:



The **Deinterlace** node extracts either the odd or even field from each frame of the input. The **Interlace** node re-interlaces fields and requires two inputs, an odd field sequence and its corresponding even field sequence. For both of these nodes, the output sequence is the same length as the input.

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 these split frames and merges the fields back together. This means that the output of the Merge node is half the length of the input.

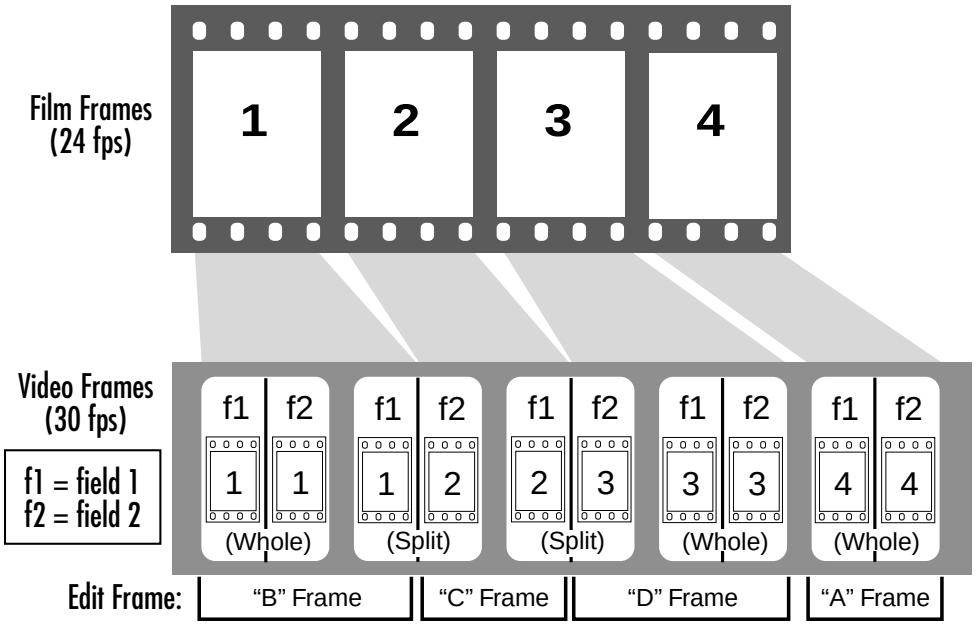
3:2 Pulldown Node ..... p. 20.2
3:2 Pushup Node ..... p. 20.5
Deinterlace Node ..... p. 20.7
Interlace Node ..... p. 20.9
Merge Node ..... p. 20.11
Split Node ..... p. 20.12
Video Safe Node ..... p. 20.14

3:2 PULLDOWN NODE

The 3:2 Pulldown node enables you to convert imagery from the frame rate of film (24 fps) to the frame rate of video (30 fps). By video, we are actually referring to the 525/60 scan rate associated with NTSC video. (The PAL video standard, on the other hand, is associated with the 625/50 scan rate. Typically, the transfer from 24 fps to 50 fps is accomplished by a simple doubling.)

To convert video to the frame rate of film, use the complementary 3:2 Pushup Node.

The 3:2 Pulldown node accepts one input and functions by creating five fields from every two input frames by replicating one field from every other frame. The following figure illustrates the functionality of the node, using a typical 3:2 pattern.



Selecting a Phase Shift

There are two basic pulldown patterns used to pad the frames to 60 fields per second: 3:2 and 2:3. However, there are a total of five different permutations, or phase shifts, necessary to match the footage being transferred to other video footage, depending on how it has been created and edited.

Depending on the system used for a pulldown, these variations may be referred to using different terminology. 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	1 + 2:3	1 + 3:2	2 + 2:3

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 24 fps, the key is to use the matching phase shift pattern for both transfers.

Tip:

Click the Info ("i") button on the 3:2 Pulldown node to find out which input frames were used as the source of the odd and even field lines that compose the current frame of the output.

Selecting a Field Order

Field Order refers to whether the odd field lines of each frame are assigned to field 1 and the even field lines to field 2, or vice versa. (Note that this parameter does not refer to selecting field dominance, that is, to selecting F1 or F2 as the field you always edit on.)

Executing a 3:2 Pulldown Sequence

By the nature of the conversion operation, additional frames are created. Therefore, to render a sequence that has passed through a 3:2 pulldown node, you must specify Start and End values in the Exec node panel that correspond to the actual number of frames resulting from the pulldown operation—not the frame range of the original input.

For example, a six-second, 144-frame film sequence that is processed through the 3:2 Pulldown node becomes 180 frames long. To execute the entire sequence, you would key a value of 1 into the Start parameter field of the Exec node and a value of 180 into the End field.

Tip:

The Frame Range parameter on the Info tab of each node displays the length of the sequence as output from the node. (To actually calculate the number of frames in the output of a pulldown, you would multiply the number of input frames by 5, and divide the result by 4.)

To process only a portion of the total input sequence, it is important to add a Sequence node between the input node and the 3:2 Pulldown node in order to specify the frames to be used.

For example, to convert only the action occurring during the last four seconds of a six-second film sequence, key a Start value of 49 and an End value of 144 into the Sequence node entry panel. The Sequence node will then deliver the image data in frame 49 of the source imagery to the 3:2 Pulldown node, but the frame number is remapped to 1.

The Sequence node and the Exec node are described in chapter 13, "Tool Nodes."

Pulldown Parameter Tab

Phase Shift

This parameter enables you to select the type of conversion the 3:2 Pulldown node should use. Select A, B, C, D, or E from the Phase Shift popup menu to specify a repeating field pattern. The most commonly used patterns are A and B:

Select "A" from the Phase Shift menu to use a "2:3" pattern…

- where the first video frame is made up of two fields based on the first film frame, the second video frame is made up of two fields based on the second film frame, and the third video frame is made up of one field based on the second film frame and one field based on the third film frame, and so on in a repeating 2:3 pattern.

Select "B" from the Phase Shift menu to use a "3:2" pattern…

- where the first video frame is made up of two fields based on the first film frame, and the second video frame is made up of one field based on the first film frame and one field based on the second film frame, and so on in a repeating 3:2 pattern.

Field Order

This parameter specifies whether the odd fields (lines 1, 3, 5, etc.) should be used to create field 1, with the interlacing even fields (lines 2, 4, 6 etc.) comprising field 2, or vice versa. It refers to spatial precedence and one choice should be used consistently throughout a facility.

Even, Odd

The even fields are used to create the first field and the odd fields are used to create the second field.

Odd, Even

The odd fields are used to create the first field and the even fields are used to create the second field.

3:2 PUSHUP NODE

The 3:2 Pushup node enables you to convert imagery from the frame rate of video (30 fps) to the frame rate of film (24 fps). It performs an inverse of a 3:2 pulldown by creating two film frames from five video fields by disposing of one of the fields.

There are five different phase shifts to choose from, each of which corresponds to a pulldown phase shift. For more information, refer to the 3:2 Pulldown Node description.

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, it is essential to use the same Phase Shift and Field Order parameter settings that were used in the 3:2 Pulldown node.

Executing a 3:2 Pushup Sequence

By the nature of the conversion operation, the output will contain fewer frames than the input sequence. Therefore, to render a sequence that has passed through a 3:2 Pushup node, you must specify Start and End values in the Exec node panel that correspond to the number of frames resulting from the pushup operation—not the frame range of the original input.

For example, a six-second, 180-frame video sequence that is processed through the 3:2 Pushup node becomes 144 frames long. To execute the entire six-second sequence, you would key a value of 1 into the Start parameter field of the Exec node and a value of 144 into the End field.

Check the Frame Range display in the Info tab of the 3:2 Pushup node to find out the length of the sequence as output from the node. (To actually calculate the number of frames in the output, you would multiply the number of input frames by 4, and divide the result by 5.)

Pushup Parameter Tab

Phase Shift

The Phase Shift parameter enables you to choose a pattern by which the 3:2 Pushup node will perform the conversion.

Field Order

This parameter specifies whether video field 1 should be used to create the odd fields (lines 1, 3, 5, etc.) in the film frame, and field 2 the interlacing even fields (lines 2, 4, 6, etc.), or vice-versa.

Even, Odd

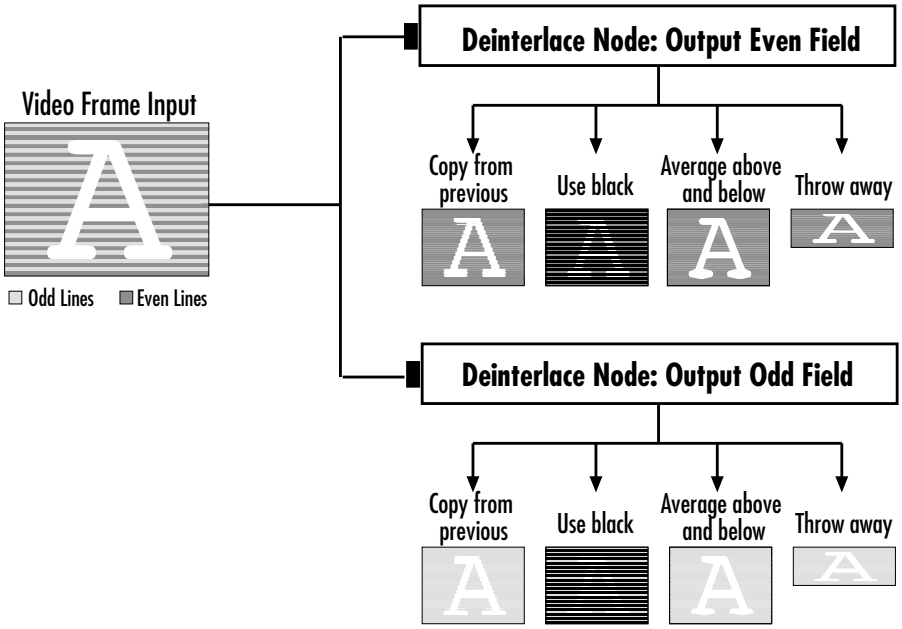
Field 1 is used to create the even field lines of the film frame, and field 2 is used to create the odd field lines.

Odd, Even

Field 1 is used to create the odd field lines of the film frame, and field 2 is used to create the even field lines.

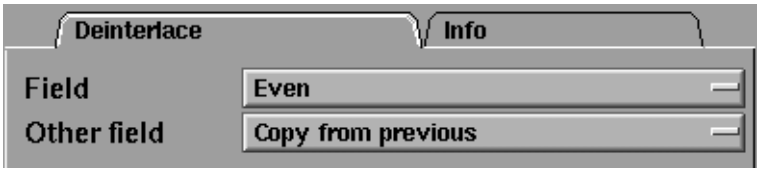
DEINTERLACE NODE

The Deinterlace node enables you to deinterlace video frames into frames of either odd or even fields, as you specify. You can also specify whether to fill in the missing field lines or to throw them away, as the following diagram illustrates:



For a comparison of the Interlace and Deinterlace nodes to the Split and Merge nodes, refer to the diagram at the beginning of this chapter.

Deinterlace Parameter Tab



Field

This parameter specifies whether you wish to extract 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. Use the popup menu to choose Odd or Even fields.

Other Field

This parameter enables you to specify how to treat the vacant spaces that remain once the Deinterlace node removes either the odd or even scanlines. Use the popup 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.

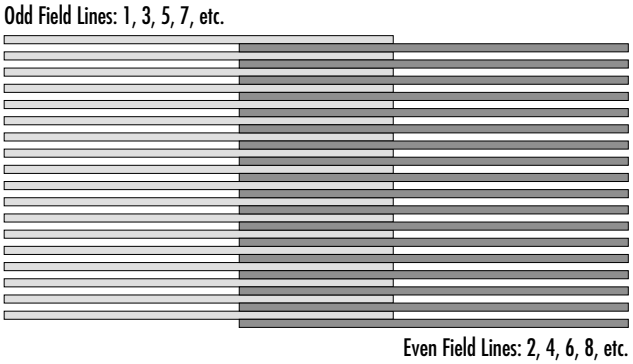
Throw Away

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, an image that was originally 720 x 486 lines will become 720 x 243 lines.)

INTERLACE NODE

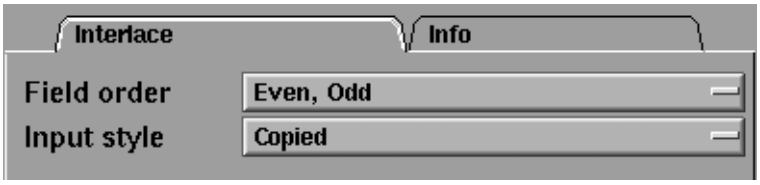
The Interlace node recombines video-type frames that have been split into separate sequences of odd and even field frames, which means that the Interlace node requires two inputs.

You can use the Interlace node to recombine frame fields that you de-interlaced using the Deinterlace Node.



Interlace Parameter Tab

The Interlace parameter tab features two parameters: Field Order and Input Style. Your choice of options should be based on the method used previously to de-interlace the frames using the Deinterlace node.



Field Order

This parameter allows you 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.).

Even, Odd

Select this option when the *top input image* should be used to create the *even fields* and the bottom input image for the odd fields.

Odd, Even

Select this option when the *top input image* should be used to create the *odd fields* and the bottom input for the even fields.

Input Style

This parameter specifies the input style with which the fields will be interlaced. These parameter options are complementary to those available when using the "Other Field" parameter to de-interlace the frames in the Deinterlace node.

Copied

Re-interlaces video fields that were previously 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 previously de-interlaced using the "Throw away" option.

MERGE NODE

The Merge node enables you to recombine video-type frames that have been split into a 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.

Merge Parameter Tab

The Merge parameter tab features two parameters: Field Order and Input Style. Your choice of options 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: odd, then even, or even, then odd.

Input Style

This parameter specifies the input style with which the fields will be merged. The options to this parameter are complementary to options used to split the frames in the "Other Field" parameter of the Split node.

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% in the Y dimension.

Copied

This input style is appropriate for all fields that were previously split using an option other than "Throw Away."

The "Copied" option merges fields that were previously split apart using the "Use Black," "Copy from Previous," or "Average Above and Below" options in the Split node.

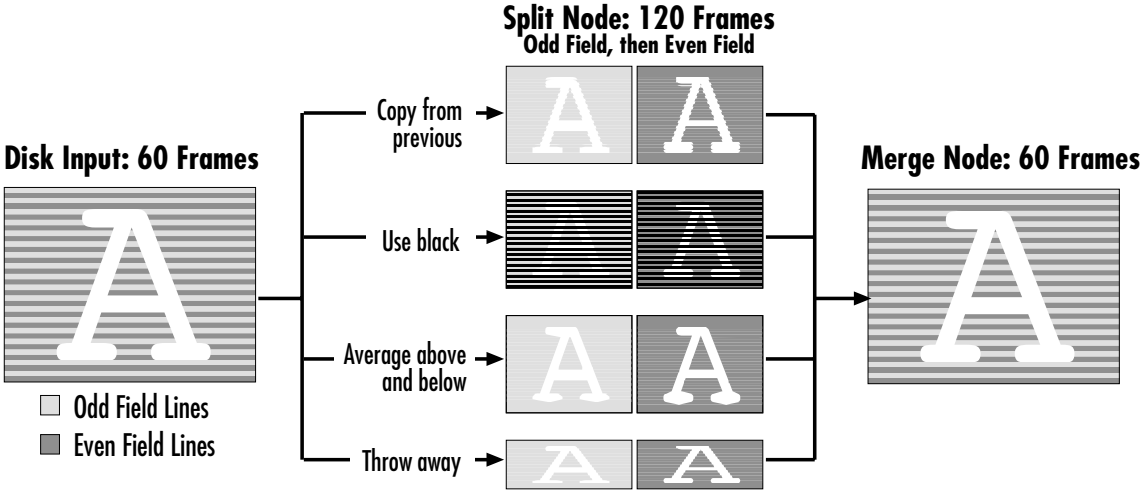
SPLIT NODE

The Split node enables you to separate a series of video frames into alternating "field frames." For example, a video clip of one second duration comprises 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.

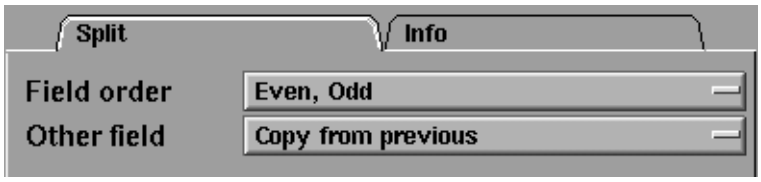
Use the complementary Merge Node to recombine the field frames.

Note:

To deinterlace frames and keep only the odd or even field information in each frame, without creating additional frames, use the Deinterlace Node instead of Split.



Split Parameter Tab



Field Order

This parameter specifies whether the alternating field frames generated by the Split node should begin with the odd field (lines 1, 3, 5, etc.) or even field (lines 2, 4, 6, etc.).

Even, Odd

Begin with an even field, and use the odd field in the following frame.

Odd, Even

Begin with an odd field, and use the even field in the following frame.

Other Field

This parameter enables you to specify how to treat the vacant spaces that remain once the Split node separates the odd or even lines into separate field frames. Use the popup 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.)

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.

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.

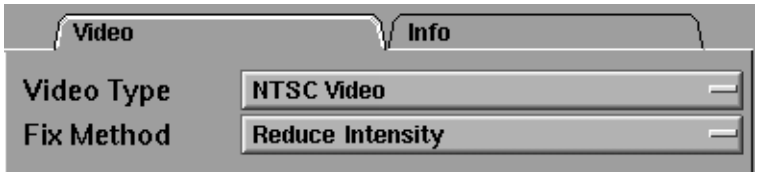
	NTSC Video	PAL Video
Analog Color Space Referenced	YIQ	YUV
Gamma Correction Value Applied	2.2	2.8
Max Chrominance Amplitude	50 IRE *	50 IRE
Max Composite Signal Amplitude	110 IRE *	110 IRE

* 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 color space 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 color space used by the selected standard, then that color is corrected by reducing either the intensity or saturation, as specified by the user.

Video Parameter Tab



Video Type

Use the popup menu in the Video Type parameter 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 video uses the YIQ color space.
- **PAL Video**: The television signal standard used for broadcast in most of Europe and in many other countries. PAL video uses the YUV color space.

Fix Method

The Fix Method parameter features a popup menu that enables you to specify 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.

21 PLUG-IN NODES

Plug-ins give you the opportunity to program your own nodes for Chalice, which can then be used in a Chalice network like any other node. Many of the features available in Chalice nodes can also be used in plug-in nodes. For example, you can process single or multiple inputs, create a custom UI, and animate parameters using the STED.

Plug-ins are DSO linkable modules that reside (by default) in the "Chalice2.0/plugIns" directory. You can set a path to additional directories with the PLUGIN_DSO_PATH environment variable.

Documentation for Plug-in Developers

Complete instructions (in html format) for plug-in developers are installed as part of the Chalice Online Help System. You can access this material from within Chalice by selecting the "Writing Plug-ins" item under the Chalice Help menu.

Source Code

The complete source code files for plug-in examples are available in chalice2.0/support/src/plugIn.

On the Web

The most up-to-date versions of the html documentation and source examples are also available in the Support section of the Silicon Grail website at http://www.sgrail.com.

CHAL

COMMAND LINE PROCESSING WITH CHAL

Chal is the command line renderer for Chalice. To execute a grail file or files from a UNIX shell or from an NT Command Prompt, without invoking the Chalice interface, use the following command at the prompt:

```
chal file.grail
```

…where "chal" informs the system that you want to run a grail script and "file.grail" is the name of the grail file that contains the process you want to run.

This command will run every process attached to the Exec node in the specified file. If you are not in the directory in which this file is located, you must either

- change directories before invoking the chal command or
- include the absolute pathname of the grail file in the command.

If the grail file contains more than one Exec node, chal will notify you of the Exec nodes it found in the file and ask you which Exec you want to run. Use the [-e node] modifier (see below) to specify which Exec node to process.

Chal Options

Chal offers a number of optional commands to modify a script by using this format:

```
chal [options] file.grail
```

For example, the following command, which uses the "-f" modifier, would process the first 50 frames (by specifying start, end, and increment values of 1, 50, and 1) of the first input to the Exec node in a grail file named "file.grail":

```
chal -f 0 1 50 1 file.grail
```

Note:

The first input to the Exec node is specified by the 0 after the "-f." Exec inputs are numbered sequentially starting from 0 for the top input. However, you can also use the name of the Exec input with this modifier, as described in the Options List.

Options List

The list of bracketed, optional commands below is complete as of publication of this document. The most up-to-date list is always available by entering the following command at the system prompt:

```
chal2.0 [IRIX]

chal2.0.lnk [NT]
```

-d …rescans the hard disk or other storage device to update image files for all Disk Input nodes in the grail file.

-e [node] …specifies a particular Exec node you wish to run. You must use this option when your grail file contains more than one Exec node, to specify

which Exec chal should process (e.g., type "-e exec1", where "exec1" is the unique name of the Exec node).

-f …specifies an Exec input and start, stop, and increment. Use this option to specify a particular input to an Exec node when the Exec node has more than one input. The input can be specified by its name, if you know it, or by number. For an Exec node with two inputs, "0" would signify the first, or top, entry in the node panel and "1" the second entry. You can also specify frame start, stop, and increment with -f.

For example, "-f 0 1 130 1" would process frames 1 to 130 of the first input to an Exec node, while "-f smoke_layer 1 10 1" would process frames 1 to 10 of an Exec input named "smoke_layer."

-i …specifies the interleave option, which will process frame 1 of the first input to the Exec node, then frame 1 of the second input, and so on, until all frames of all Exec inputs are processed.

-j …specifies sequential processing, in which every frame of the first input connected to the Exec node is processed before processing begins on the subsequent input.

-k …instructs chal to continue processing when it encounters an error. It will skip the current frame and attempt to process the next one. Note that for preview renders (-P option, below) you can use the -M option instead, which fills missing frames with the color you specify (see -M, below).

-p [numProcs] …runs the script on more than one processor (by default, chal will run on only one CPU). For example, to specify that chal use four processors while running the script (assuming your workstation has at least four CPUs), type "-p 4".

-q …specifies "quiet mode," which prevents feedback from appearing on screen while the script is running.

-r …creates a temporary "restart" file, in which chal will save information for each successfully completed frame until the entire chal job is done. If chal is aborted (by a render queue system, e.g.) during a "-r" render, you can use the exact same chal command that started the original job to restart the job where it left off. If the command varies at all, however, chal will start the job from the beginning.

When using the restart option, be sure that two (or more) chal renders do not write to the same restart file.

-s [numSlices] …specifies the number of slices to use ("-s 4" would specify 4 slices).

-t …displays the average rendering time per frame for each Exec input processed.

-v …validates the grail script instead of rendering.

-D [global def] …defines global variables, and overrides the definition of the global in the grail file with the value specified on the command line. For example, "-D JOB usr/people/richard" would define the $JOB global for the grail file or

files currently being processed by chal to be "usr/people/richard" regardless of the definition specified in the grail file itself.

-F [filename] …creates a list of files opened for reading filename.

-M [rgbVals] …instructs chal to replace any missing input frames with a frame of a solid color and continue processing, You specify the color to use by entering an RGB value after the -M option. This option works only in conjunction with the preview render option (see below). So, to specify a low-quality, half-resolution preview render and fill any missing frames with red, you would type "-P l 50 -M 1 0 0."

-P [level size] …specifies that chal render at a specific quality level and resolution. The quality level is set by typing the letter l for low, m for medium, or h for high. The resolution is set as a percentage of the full size of the image. To specify a preview execute at low quality and half of full resolution, for example, you would type "-P l 50."

-T [tag] …defines the process tag.

-V The "chal -V" command displays the version number of chal that you are currently using,

Chal Caching Options

By default, chal will use all available RAM and assume that disk space is unlimited (however, chal will only use as much disk space as is actually needed to complete the job). The following optional commands modify the default RAM and disk cache settings used by chal to process the grail file:

-C r [ramsize] …sets the RAM cache to use for the chal script. For "ramsize," use a number representing the total size of the RAM cache in bytes, or use a number followed by one of these four options: k, m, G, or p.

- k …specifies kilobytes (multiply value by 1024)
- m …specifies megabytes (multiply value by 1024 * 1024)
- G …specifies gigabytes (multiply value by 1024 * 1024 * 1024)
- p …specifies percent (value is the percentage of RAM to use)

For example, "-C r 10p" would specify that chal use 10 percent of the total RAM to run the script.

-C d [disksize] …sets the disk cache size to use for the chal script. For "disksize," use a number representing the total size of the disk cache in bytes, or use a number followed by one of these three options: k, m, or G (see above).

B CHALICE EXPRESSION LANGUAGE

Using Expressions in Chalice

Chalice enables you to use expressions, rather than a fixed numerical value, to modify the output of many nodes. You can apply functions to modify parameter values, animate values over time, and access channel data from other nodes.

You can use expressions in any node that features an Expressions checkbox in its node panel. When "Expressions" is checked, you can key expressions into the data entry fields of the node parameters.

Expressions are a very powerful way to customize and extend the capabilities of the Chalice system, but even novice users can take advantage of expressions by following the examples given in this manual.

For example, the Track node generates tracking data that can be accessed throughout the Chalice system whenever you need to reference selected pixel coordinates across a range of frames. You can use expressions to reference tracking data in a Corner Pin node to set the position of each corner. The basic steps for performing a four-corner pin using track data, including the exact expressions you need to use, are enumerated in the description of the Corner Pin Node in chapter 17, "Movement Nodes."

Another instance in which you could reference tracking data is in the Pixel Meter node. To obtain the average RGBA values for the area tracked in a Track node, you would key an expression into the Pixel Meter node that accesses the track data to drive the pixel metering function. Then the resulting pixel meter data, too, could be referenced in other nodes by using the appropriate expression. For more information, see the Pixel Meter Node description in chapter 13, "Tool Nodes."

Note:

Chalice also includes an Xpresso Node, in which you write expressions using Xpresso node syntax. The information you need to write Xpresso expressions is included in the node description in chapter 19, "Effects Nodes."

Writing Expressions

As stated above, this manual includes complete expressions that you can use for several common operations. However, you can adapt these expressions or write other expressions by following the guidelines given in this appendix.

You can use numbers, variables, arithmetic operators, and functions to write Chalice expressions. Essentially, an expression tells Chalice how to modify an input or inputs to produce an output at that node in the network. 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.

Numbers

You can use the integers and floating point numbers in Chalice expressions:

- Integers are the whole numbers: -1, 1, 2, 3, etc.
- Floating point numbers are the decimal fractions: -1.1, 0.5, 1.25, etc.

Note:

Floating point numbers can also be expressed in exponential notation. Exponential notation refers to 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, you would use the exponent operator or the "pow" function.)

Variables

A variable is a name consisting of one or more uppercase letters, and each variable name starts with a dollar sign ($) to identify it as a variable.

You can use variables in Chalice expressions in place of specific numbers or other values. A variable is defined by the Chalice system and can represent a value that changes. For example, the $F variable represents the number of the current frame in a sequence.

You can use the following list of variables in Chalice expressions:

$F ..... current frame number

$T ..... current time, in seconds

$W ..... current image width

$H ..... current image height

$E ..... value of *e* (2.7182818284590452354)

$PI ..... value of *pi* (3.1415926535897932384)

$JOB ..... Job directory (this variable is user definable)

$OP ..... name of current node (used in Disk Output and Text nodes only)

$F and $T

$F is perhaps the most commonly used variable. It is typically used in numeric expressions to specify frames for modification, but it can also be used in the Disk Output node to embed frame numbers into filenames (see the description of the Disk Output Node in chapter 12, "I/O Nodes," for more information).

Instead of $F, you may occasionally need to use the $T variable, which is suitable for subframe (floating point) operations. Using a frame rate of 30 fps as an example, at frame 60 the $T variable would represent 2 seconds.

$JOB

This variable is used in the Disk Input and Disk Output nodes to help define the directory path of source imagery and output imagery.

The default location of the Job directory is the directory from which you started Chalice. Unlike the other variables listed here, you can change the definition of $JOB. For more information, see "Setting Global Variables" in chapter 6.

$OP

This variable is used only by the Disk Output and Text nodes. It represents the name of the current node and is used as part of the filepath specification (in Disk Output) or for diagnostic display purposes in the Text node.

Refer to the relevant node descriptions for more information.

Arithmetic Operators

Following is a list of the arithmetic operators available for writing Chalice expressions and some examples that illustrate their use.

The operators are listed in order of precedence; that is, in the order by which they are evaluated in the expression, from first to last.

Operations in parentheses are evaluated first; logical operations are evaluated last. All operators with the same priority are evaluated from left to right.

( ) ..... Operations in parentheses

- ..... Negation (i.e. 3 + -3, which evaluates to 0)

* / ^ % ..... Multiplication, division, exponent and modulus

+ - ..... Addition and subtraction

Logical operators:

> ..... greater than or equal to

< ..... less than or equal to

!= ..... not equal to

== ..... equal to

|| ..... or

&& ..... and

! ..... not; negation

Note:

In general, you should avoid comparing floating point numbers for equality—do not use the conditional operator "==" with floats. Round-off errors can cause very small (10e-8) inaccuracies in floating point calculations.

For example, in the following expression…

Expression: ..... Result:

3 + 4 * 5 23

…the "multiply," 4 * 5, is evaluated first; then the result, 20, is added to 3.

To multiply the result of 3 + 4, or 7, by 5—that is, to evaluate the add operation first—you would enclose the addition operation in parentheses, as follows:

Expression: ..... Result:

(3 + 4) * 5 35

Other Examples

Expression: ..... Result:

2 * 3 ^ 2 + 4 * 6 / 2 48

That is, 36 plus 12:
2 * **3** = 6, and 6 **^ 2** = 36
4 * **6** = 24, and 24 **/ 2** = 12
36 + 12 = 48

Expression: ..... Result:

((3 + 4) * 5 - 5) / 6 5

That is, 30 divided by 6:
3 + **4** = 7, and 7 * **5** = 35
35 - **5** = 30, and 30 **/ 6** = 5

Expression: ..... Result:

13 % 5 + 13 / 5 ..... 5.6

That is, 3 plus 2.6:
13 % 5 = 3 , and **13 / 5** = 2.6
3 + 2.6 = 5.6

Note that "13 % 5" is read as "13 modulo 5"; 3 is the remainder of dividing 13 by 5.

Math Functions

You can use the following math functions in Chalice. Input can be a numerical value or an expression that, when evaluated, will result in a numerical value. Unless otherwise specified, a value can be in integer or floating point form.

Trigonometric Functions:

(All angles are in degrees unless otherwise noted.)

acos(val) ..... Trigonometric arccosine of val.
e.g. acos(0) = 90

asin(val) ..... Trigonometric arcsine of val.
e.g. asin(0.866025) = 60

atan(val) ..... Trigonometric arctangent of val.
e.g. atan(1.73205) = 60

atan2(float y, float x) ..... Compute the arctangent of y/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 the case where x is zero correctly, returning 90 or -90.
e.g. atan2(1, 0) = 90
atan2(0, 1) =0
atan2(0, -1) = 180

asin(val) ..... Trigonometric arcsine of val.
e.g. asin(.866025) = 60

cos(val) ..... Trigonometric cosine of val.
e.g. cos(60) =.5

cosh(val) ..... Hyperbolic cosine of val.

rad(val) ..... Convert val to radians (val is in degrees).
e.g. rad(180) = 3.1415926

sin(val) ..... Trigonometric sine of val.
e.g. sin(60) = 0.866025

sinh(val) ..... Hyperbolic sine of val.

sqrt(val) ..... Square root of val.
e.g. sqrt(144) = 12

tan(val) ..... Trigonometric tangent of val.
e.g. tan(60) = 1.73205

tanh(val) ..... Hyperbolic tangent of val.

deg(val) ..... Convert val to degrees (val is in radians).
e.g. deg($PI) = 180

Logarithmic/Exponential functions:

exp(val) ..... Logarithmic exponentiation function (eval).
e.g. exp(2) = 7.38906

log(val) ..... Natural logarithm of val.
e.g. log(2.718281828) = 1

log10(val) ..... Logarithm base 10 of val.
e.g. log10(10)= 1

pow(float base, float exponent) ..... This computes the base to the power given.
e.g. pow(2, 3) = 8

Value Type Functions:

These functions allow you to work with positive/negative, even/odd, and integer/floating point values.

abs(val) ..... Absolute value of val.
e.g. abs(-2.6) = 2.6

clamp(val, min, max) ..... Clamps the value between min and max. Used to prevent val from going outside the specified range.

even(val) ..... Returns 1 if val is even, 0 if val is odd

odd(val) ..... Returns 0 if val is odd, 1 if val is even.

floor(val) ..... Returns the largest integer smaller than val.
e.g. floor(2.78135) = 2

int(val) ..... Integer value of val by truncating.
e.g. int(2.6) = 2
int(-2.6) = -2

min(val1, val2, …) ..... Returns the smallest value listed.

max(val1, val2, …) ..... Returns the largest value received.

pulse(val, min, max) ..... Creates an on/off pulse. If the value is less than min or greater than max, pulse returns 0, otherwise it returns 1.

sign(val) ..... Returns -1 if the val is negative and +1 if val is positive.

smooth(val, min, max) ..... Returns a smooth interpolation between 0 and 1. If val is less than min it returns 0, if val is greater than max it returns 1.
e.g. smooth($F, 12, 55)
This generates an ease-type curve between values 0 and 1 starting at frame 12, and ending at frame 55.

trunc(val) ..... Truncates all digits to the right of the decimal
e.g. trunc(4.5678) = 4

wrap(val, min, max) ..... Resulting value always falls between min and max. It creates a sawtooth wave for continuously increasing and decreasing values of the value.

Dataflow Control Functions:

These functions allow you to control the order in which commands are executed in Chalice.

if(exp, true_val, false_val) ..... Returns true_val if exp is true, and false_val if exp is false.
e.g. if($F > 12, $F, 12)
This will send back the value 12 unless $F is greater than 12, in which case it will return the current frame number.

block(exp1; exp2; …) ..... The block function instructs Chalice to execute the series of expressions sequentially at each pixel. Chalice normally executes a command at all pixels in the screen before moving to the next expression

Interpolation Functions

The following interpolation functions generate smooth values between 1 and 0:

bezier() ..... Interpolates the in and out point by fitting a bezier curve using the slopes and accelerations.

cubic() ..... Interpolates the in and out point using the slopes. This results in a cubic polynomial.

constant() ..... Constant value of the in point.

ease() ..... Values ease in smoothly starting at 1 and ease out smoothly at 0.

easein() ..... Ease in smoothly at the start only.

easeout() ..... Ease out at the end only.

easep(val) ..... An ease-in/ease-out curve which is weighted by the power. This weighting has the effect of shifting the inflection point.

easinp(val) ..... Same as easep() but it is an ease-in curve.

easeoutp(val) ..... Same as easep() but it is an ease-out curve.

linear() ..... Straight line interpolation from 1 to 0.

noise(x,y,z) ..... Generates random value given a position in space. Nearby positions will have similar values.

quintic() ..... Interpolates the in and out point using slopes and accelerations. This results in a quintic (degree 5) polynomial.

rand() ..... Generates random values between 0 and max at random locations in an image.

smooth(float val, float min, float max) .. The return value is a smooth interpolation between 0 and 1. When the value is less than the minimum, the return value is 0. If the value is greater than the maximum, the return value is 1.
e.g. smooth($F, 12, 55)
This will generate an ease-type curve between values 0 and 1, starting at frame 12 and ending at frame 55.

turb(x,y,z, depth) ..... Like the noise function, but allows for fractalization of the noise. Thus, at a depth of three, the noise will be fractalized three times.

Data Channel Access Functions

You can use these functions to access parameter data from another node. Some function types, such as the ch (channel) and prm (parameter) functions, can be used generally; others, such as the meters function, the track functions, and the lprm functions, enable you to reference parameters specific to a particular node type.

Items in brackets indicate available choices; for example, in the track function, "[xv|yv]" indicates that you can use "xv" or "yv." Also, be aware that if you use the ch functions, you must enclose the node/channel name in quotes, as shown in the examples. (This is not necessary for the other data channel access functions.)

Channel (ch) Functions

The ch functions have been included in Chalice since the first version of the software was released. They have since been augmented by the parameter functions, which offer more flexibility in data access. The ch functions are still a fast and easy way to access node data from nodes that do not offer multiple datasets.

The ch, chf, and cht functions allow you to reference parameter data from any other node:

ch("node_name/channel_name")

chf("node_name/channel_name", frame#)

cht("node_name/channel_name", time)

For example, you might want to use the "brightness" value from a node called "brightness1" in other Brightness nodes. Using the ch function you would enter the following to access the data:

ch("brightness1/brightness")

Similarly, the chf and cht functions are used to access animated parameter data or data from specific frames:

chf("brightness1/brightness", $F)

To access a specific value from a parameter with more than one associated value field (*x* and *y* coordinates, e.g.), use the exact parameter field names as listed in the STED. Using the Translate parameter in the Transform node as an example, the following would access the *x* value:

ch("transform1/translatex")

Tip:

To find out the exact parameter field name to use, refer to the name used in the STED window list for the parameter.

Parameter (prm) Functions

The prm functions allow you to reference parameter data from any node in the network, including parameters that contain more than one entry (unlike the ch functions).

prm(node_name, parm_name, value_index)

prmf(node_name, parm_name, value_index, frame#)

prmt(node_name, parm_name, value_index, time)

The **parm_name** refers to the name of the specific node parameter to be referenced, e.g., "brightness." (Chalice recognizes the same parameter name that is used in STED for that parameter.)

The **value_index** refers to a specific type of value within that parameter, as when you need to specify the x value in a parameter consisting of a pair of *x,y* coordinate values. For parameters that have only a single value to reference, 1 is always used as the value index. (Multiple-field parameters are explained later in this description.)

Notice that the prm functions use a slightly different syntax than the ch functions. The names of the node and parameter being accessed are not enclosed in quotation marks. Additionally, they are separated by a comma instead of a slash. Using the same brightness parameter example from the ch function description above, to access this data using the prm function you would enter the following:

prm(brightness1, brightness, 1)

In the above example, "brightness1" is the node name and "brightness" is the name of the parameter that holds the brightness value. Note that if the node name includes spaces, you must surround the node name with quotation marks.

CORRECT: prm("very bright", brightness, 1)
WRONG: prm(very bright, brightness, 1)

Data channel access functions offer several advantages. First, if you need to tweak values later, any change you make to the referenced Brightness node settings will be reflected in all other nodes that access the data.

Second, the prmf and prmt functions allow you to access parameter values that have been animated, or to access a parameter value at a specific frame. The following example will access the brightness parameter value for the current frame ($F):

prmf(brightness1, brightness, 1, $F)

To access the brightness value of frame 6 only, replace "$F" with "6":

prmf(brightness1, brightness, 1, 6)

Third, you can adjust the brightness level in the other Brightness nodes by modifying the prm function and still maintain the shape of the curve over time, perhaps to give different layers of a composite slightly higher or lower brightness levels (this example lowers the brightness by 20 percent):

prmf(brightness1, brightness, 1, $F) * 0.8

The prmf and prmt functions also allow you to create a frame offset for the accessed data. The next example creates a six-frame delay by specifying the current frame using the $F variable and then adding 6:

prmf(brightness1, brightness, 1, $F + 6)

Note:

It is recommended that you use either the prmf or prmt function consistently in a grail file, but not both. Due to the nature of floating point math, two expressions that are the same, except that one uses prmt and one uses prmf, may produce different results. (The same holds true for the chf and cht functions.)

The brightness parameter, which has been used in the examples so far, only holds one value. However, to access a Sample Color parameter that holds separate values for R, G, and B, sequential value index numbers are used to indicate the value field to access. In this case, "sample, 1" would be used to reference the Red channel data, "sample, 2" to reference the Green channel, and "sample, 3" for the Blue.

In the Transform node, the Scale, Translate, and Pivot parameters each contain value fields for *x* and *y* coordinates. To reference *x* and *y* values for a parameter, use 1 as the value index for the *x* value and 2 as the value index for the *y* value. The following obtains the *y* value from the Translate parameter in a node named "transform1":

prm(transform1, translate, 2)

Note:

For multiple-field parameters that hold a pair of coordinate values, the STED will list the parameter name as (using the above Transform parameter example) "translatex" or "translatey." To reference these or any similar parameters in prm functions, use the name as it appears in the STED, but without the "x" or "y" appended. To indicate which coordinate value to reference, use the appropriate value index number instead.

To specify the frame for which to obtain the value, you would use prmf. In this case, the prmf function would return the *y* translate value at the current frame:

prmf(transform1, translate, 2, $F)

List Parameter (lprm) Functions

The lprm functions allow you to reference parameter values from nodes that create a list of entries with the same parameter options, such as Mcomp, which lists its inputs. These functions allow you to specify which entry in the list Chalice should reference:

lprm(node_name, list_name, list_index, parm_name, value_index)

lprmf(node_name, list_name, list_index, parm_name, value_index, frame#)

lprmt(node_name, list_name, list_index, parm_name, value_index, time)

The **list_name** refers to the type of list entry, such as "inputs" for the Mcomp, Zcomp, Sequence or Exec nodes; "points" for the Track node; or "pixelMeters" for the Pixel Meter node. (The table below shows the specific names to use for each type of list.)

The **list_index** refers to the index number of the list entry you have selected. For example, the fourth input to an Mcomp node would have a list index of 4. The list index has a range of 1–n, where *n* represents the total number of entries in the list.

The **parm_name** refers to a specific parameter of the list entry to access, such as "start" for a Sequence node input or "pos" (position) for a pixel meter in a Pixel Meter node. For example, an expression that obtains the transparency value from the second input to a node titled "mcomp1" would be written as follows:

```
lprm(mcomp1, inputs, 2, transp, 1)
```

In some cases, the parameter may include more than one value. For example, the "pos" parameter for each "pixelMeters" entry contains both *x* and *y* values. You use the **value_index** to specify which value to use. For the "pos" parameter, you would specify "pos, 1" for the *x* coordinate or "pos, 2" for the *y* coordinate.

The index values for each parameter type are given in the table below:

Node Type	List Name	Parameter Name	Value Index		
Exec	inputs	execstart	1		
		execend	1		
		execinc	1		
		slices	1		
Mcomp/Zcomp	inputs	transp	1		
Pixel Meter	pixelMeters	size	1 (*x* value)	2 (*y* value)	
		pos	1 (*x* value)	2 (*y* value)	
Sequence	inputs	start	1		
		end	1		
		inc	1		
		hold	1 (start)	2 (middle)	3 (end)
Track	points	pos	1 (*x* value)	2 (*y* value)	

Meters Function

The meters function allows you to reference a specific channel of pixel meter data from a Pixel Meter node. Pixel meter data consists of averaged RGBA values for a specified area in an image. (See also the Pixel Meter Node description in chapter 13, "Tool Nodes.")

```
meters(meters_node_name, pixel_meter#, channel#, frame#)
```

The following example would reference the red channel data (0) at the current frame ($F) from the first pixel meter (1) of a node named "meters1":

```
meters(meters1, 1, 0, $F)
```

Track Functions

The Track functions—track, tracka, and tracksr—enable you to access pixel coordinates from track point data, or to compute the difference in angle or vector described by track point coordinates.

The **track** function allows you to reference pixel coordinate data from a Track node in the current expression.

track(track_node_name, [xv|yv], point#, frame#)

To use the track function, specify the name of the Track node, the X-axis (xv) or Y-axis (yv) values, the track point from which to take the values, and the frame number. The following example references the X-axis values (xv) for the current frame ($F) of the first track point (1) in a node named "track1":

track(track1, xv, 1, $F)

The **tracka** function returns the difference between two frames of the angle formed by the track data.

tracka(track_node_name, point1#, point2#, ref_frame#, cur_frame#)

The Track node being referenced must have at least two track points to obtain the vector used in the comparison. You specify which two track points to use to obtain the vector as well as the reference frame and the current frame to be compared.

The following example computes the difference in angle of the vectors formed by track points 2 and 3 between frame 1 and the current frame ($F) of a node named "track1":

tracka(track1, 2, 3, 1, $F)

The **tracksr** function computes the difference in scale factors of track data between two frames, which you specify.

tracksr(track_node_name, [scale|xscale|yscale], ref_frame#, cur_frame#)

- If you use "scale," the tracksr function measures the distance between the two track points at each of the two frames you specify in the expression and computes the difference.
- "Xscale" uses the distance between track points along the horizontal axis.
- "Yscale" uses the distance between track points along the vertical axis.

The Track node being referenced must have two track points to obtain the distance used in the comparison. If the node has more than two track points, the tracksr function will use the first two points.

The following example, which assumes a Track node named "track1," would calculate the distance between the first two track points at frame 1 and again at frame 25 and return the difference between the two distance values:

tracksr(track1, scale, 1, 25)

C SOFTCLIP CONVERSION

Appendix C provides the equations used to perform Cineon Softclip conversions in the Bit Reduce and Bit Expand nodes (as well as in the DPX/Fido tab sections of the Disk Input and Disk Output nodes).

Note:

The Cineon Softclip conversion algorithm works with 10-bit log data, even if the input to the Bit Reduce or Bit Expand node is linear. However, you do not need to convert to 10-bit log before assigning values to the conversion parameters.

If the input image data is linear, you should enter corresponding linear values in the Softclip conversion parameters. Chalice will convert these values internally to 10-bit log according the specifications of the "Advanced - Cineon Settings" parameter tab.

Bit Reduce Node

The following equation shows the precise conversion process for values above Reference Black:

A breakpoint is computed for Softclip:

Breakpoint = RefWhite – Softclip.

If an input value is below this breakpoint, the output will not be influenced by the Softclip setting. These values will be calculated as follows:

OUT = 10 ^ ((IN – RefWhite) * 0.002/Film Gamma) ^ (Display Gamma / 1.7) * (Gain – Offset)

Gain = (2 ^ N – 1) / (1 – 10 ^ ((Reference Black – RefWhite) * 0.002 / 0.6) ^ (Display Gamma / 1.7)

Offset = Gain – (2 ^ N – 1)

If an input value is above the breakpoint, the output must be adjusted according to the parameters specified in Softclip:

OUT = (IN – Breakpoint) ^ (Softclip / 100) * Kneegain + Kneeoffset

Kneeoffset = 10 ^ ((Breakoint – RefWhite) * 0.002 / Film Gamma) ^ (Display Gamma / 1.7)) * Gain – offset

Kneegain = (2^N – Kneeoffset) / ((5 * Softclip) ^ (Softclip / 100))

Variables:

OUT = Converted output value.

IN = Raw Cineon Input value.

RefWhite: The 10-bit log value of the 16-bit linear value entered in the Reference White parameter. This value is calculated according to the settings of the "Advanced - Cineon

Settings" parameters. If defaults are used, 4095 in Reference White will be converted to 685. The latter value will define RefWhite in the equation.

Film Gamma: Read directly from the Film Gamma parameter.

Display Gamma: Read directly from the Display Gamma parameter.

N = The Format type of the converted image. For example, a conversion to 8-bit linear would result in a value of 2^8 – 1 = 255.

Bit Expand Node

The conversion assigns output values by computing the inverse log of the input Cineon data, then scaling and adjusting according to the reference white, reference black, film gamma, display gamma, and output data format, as the following equation shows:

OUT = RefWhite + log10[((IN + Offset) / Gain) ^ (1.7 / Display Gamma)] / (0.002 / Film Gamma)

Gain = (2 ^ N – 1) / (1 – 10 ^ ((Reference Black – RefWhite) * 0.002 / 0.6) ^ (Display Gamma / 1.7)

Offset = Gain – (2 ^ N – 1)

Variables:

OUT = Converted output value.

IN = Raw Cineon Input value.

RefWhite: The 10-bit log value of the 16-bit linear value entered in the Reference White parameter. This value is calculated according to the settings of the "Advanced - Cineon Settings" parameters. If defaults are used, 4095 in Reference White will be converted to 685. The latter value will be used in the equation.

Film Gamma: Read directly from the Film Gamma parameter.

Display Gamma: Read directly from the Display Gamma parameter.

N = The Format type of the converted image. For example, a conversion to 8-bit linear would result in a value of 2^8 – 1 = 255.

INDEX

Symbols

$E B.2
$F variable
  in expressions B.2
  in image sequence filenames 12.13
  when not to use 12.24
$H variable B.2
$JOB variable 12.23
$OP variable 12.23
$PI B.2
$T variable B.2
$W variable B.2
==, floating point numbers and B.3
2:3 pattern 20.4
3:2 pattern 20.4
3:2 pulldown 20.2
3:2 pushup 20.5
5245 (50 ASA) 18.38
5248 (100 ASA) 18.38
525/60 scan rate 20.2
5279 (500 ASA) 18.38
5298 (500 ASA) 18.38
625/50 scan rate 20.2

A

A field, in STED 9.6
A/B wipe in monitors 8.5
Abort Button 5.5
abs(val) B.5
acceleration point, in STED 9.4
acos(val) B.4
Admin menu 10.10
Adobe Photoshop files 12.4
Adobe Type 1 fonts 13.21
AdvantEdge (Ultimatte composite) node 15.15
alpha channel 15.2
  composite nodes and 15.1
  displaying in monitor 8.8
  generating
    in Channel Swap node 13.2
    in Disk Input node 12.18
    in Rotospline node 16.12
    in Ultimatte PFG node 16.36
  inverting
    in Chromakey node 16.2
    in Rotospline node 16.12
  premultiplying RGB values by 14.23
  transparency factor 15.5
  unpremultiplying RGB values by 14.23
alpha channels
  combining, max or min
    in MinMax node 15.14
    in Xpresso node 19.20
  swapping 13.2
alpha coverage 15.2
alpha max 15.14
alpha min 15.14
Alt key, using to
  access text editor window 13.4
  drag-select multiple curve points 18.9
  enter carriage returns in text fields 6.6
  invoke text editor 6.6
  open a monitor 8.5
anamorphic display
  in flipbooks 8.17
  in monitors 8.11
animating images, in Transform 17.12
anti-aliasing filter
  in Corner Pin node 17.4
  in Rotospline node 16.12
  in Text node 13.22
append notifier 5.19
Arrow keys, using to
  transform imagery 17.13
  transform Rotospline shapes 16.9
  transform text in Text node 13.19
  translate Rotospline points 16.8
asin(val) B.4
Aspect Ratios 11.7
atan(val) B.4
atan2(float y, float x) B.4
Atop node 15.9
audio, in flipbooks 8.16
Auto Cache Input option
  in Time Convolve node 19.10
  in Time Stretch node 13.24

Auto Update Node Icon preference 11.6
Autosave Preferences panel 11.10

B

backing, bluescreen
  correcting flaws in 16.18
  reconstructing 16.26
  suppressing to black 16.36
batch processing A.1
BCG layer, in Color Correct node 18.7
Bessel filter 17.15
Bezier function in STED 9.7
bezier() B.6
Bind Buffer option 8.5
bit depth
  conversion, optimizing
    in Bit Expand node 14.2
    in Bit Reduce node 14.7
  selecting in Color node 12.11
Bit Expand node 14.2
  reversing Bit Reduce operation in 14.2
Bit Reduce node 14.7
  conversion methods available 14.7
Black & Rolloff conversion method 14.11
Black Gloss parameter 16.39
Blackman window type 17.15
blacks, matching in Ultimatte CC node 16.45
block(exp1; exp2; …) B.6
Blue Logic 16.38
"bluescreen blue" values 16.2
bluescreen compositing 16.13
bluescreens, reconstructing a reference clip for 16.26
Blur node 18.24
blurring
  across frames 19.8
  in Convolve node 18.26
  in Scale node 17.11
  in Vector Blur node 19.11
border pixels
  in Blur node 18.25
  in Convolve node 18.29
  in Edge node 18.35
  in Rank node 18.43
  in Unsharp Mask node 18.49
  in Vector Blur node 19.11
bottom input, composite nodes 15.1
Bounce button
  in Disk Input sequencer 12.17
  in flipbook 8.15
  in Sequence node 13.14
box (constant) filter 17.15
branches 5.12
brightness
  adjusting in Brightness node 18.14
  adjusting in Color Correct node 18.7
  formula used 18.14
  offset 18.14
  varying, in Convolve node 18.26
Brightness node 18.14
Buffer Auto Increment option 10.9
Buffer Persistence option 10.9
buffers, image 8.4
Bump node 19.2
bumpmap
  creating
    in Bump node 19.2
    in Surface Normal node 19.3
  using
    in Emboss node 19.4
    in Vector Warp node 19.6

C

Cache Meter menu 5.4
Cache Settings panel 10.10
Cache Size parameter 6.5
Cache Usage, show current 5.4
caches (RAM and disk)
  flushing
    in Cache Meter menu 5.4
    in Cache Settings panel 10.11
  setting in chal A.3
  setting in Chalice 10.10
caching
  individual node data 6.6
  overview of 4.10
camera jitter, removing 17.18
camera moves, simulating in Transform node 17.12
Canny edge detection 18.29
Cartesian coordinates 19.19
Catmull-Rom/Overhauser Spline filter 17.15
CDI (Chalice Direct Image) format 12.4
ch functions B.7

chal A.1
optional commands A.1
system requirements 3.1
Chalice
installing 3.2
launching 5.2
Chalice Direct Image format 12.4
Chalice expression language B.1
functions used by Xpresso node 19.17
Chalice image representation theory 4.6
Chalice interface, layout of 5.3
Chalice Online Help System 10.12
CHALICE_CUSTOM_PATH 7.11
CHALICE_HELP_BROWSER 3.2
CHALICE_TMPDIR 3.6
chalicetune 3.6
channel access functions B.7
Channel Name list, in STED 9.5
Channel Swap layer, in Color Correct 18.10
Channel Swap node 13.2
channels, image. *See also* alpha channel
adding, removing, swapping
in Channel Swap node 13.2
in Color Correct node 18.10
in Disk Input node 12.18
editing 18.9
channels, parameter
animating 6.9
editing 9.1
characteristic curve 18.18
checkboxes in parameter tabs 6.4
Checkerboard 19.20
Choose File dialog box 12.12
chroma signal 20.13
Chromakey node 16.2
Cineon conversion documents referenced 12.20
Cineon Extract node 14.17
Cineon file formats 12.4
Cineon images
conversion of
default 12.19
Kodak documents referenced 4.8
linear to log 12.26
log to linear 12.19
overview of available options 4.7
Softclip method 14.12
using "insert into" option 12.20
using "remap to" option 12.19
exporting
DPX 12.26
Fido 12.26
importing
DPX 12.19
Fido 12.19
Cineon Insert node 14.19
Cineon Linear 12.3
converting back to 10-bit log 14.17
Cineon Softclip conversion method 14.12
algorithm for C.1
Cinesite Digital Film Center 12.20
clamp(val, min, max) B.5
clamping values
in Minus, Multiply, and Plus nodes 15.11
in Xpresso node 19.16
Classic Screen Correction, Ultimatte 16.18
clean plate 16.18
clipboard 10.4
Clone command 10.4
Color Control, Ultimatte 16.44
Color Correct node 18.4
color correction
comprehensive 18.4
cumulative operations 18.4
curve editing 18.9
for broadcast video 20.12
hue-based 18.19
color filter nodes 18.3
Color Logic, Ultimatte 16.38
Color node 12.9
Color Picker
in Bit Reduce node 14.14
in Color Correct node 18.6
color ramp 12.9
color representation in Chalice 4.6
color, specifying in Text node 13.22
color values, inverting
in Color Correct node 18.12
in Inverse node 18.21
color wheel
in Chromakey node 16.3
in Color Correct node 18.9
colors, reducing number of (posterize effect) 19.7
colorspace conversion
in Bit Expand node 14.2
in Bit Reduce node 14.7

colorspace conversion continued
in Video Safe node 20.12
linear to linear, optimizing 14.21
Comma key, using for frame navigation 5.18
command line renderer (chal) A.1
comment fields 13.4
Comment node 13.4
Comment parameter of Info tab 6.6
comments, displaying in popup panel 13.4
Compass icon, in Vector Blur node 19.13
composite nodes 15.1
composite video signal 20.12
compositing
bluescreen 16.13
disparate images 15.1
multilayer
customizing, in Xpresso node 19.15
in Mcomp 15.4
in Zcomp 15.6
premultiplied images 15.2
Ultimatte 15.3
terminology used in 15.1
using Ultimatte AdvantEdge 15.15
compression
Jpeg 12.28
setting quality of 12.32
LZW 12.32
MPEG 12.30
of image files 12.25
Packbits 12.32
TIFF 12.31
using prediction encoding 12.32
Zip 12.32
setting level of 12.32
connecting nodes 5.11
constant function in STED 9.6
constant() B.6
contacting Silicon Grail 1.3
Continuous Updates option 10.9
contrast
adjusting
by modifying low and high values 18.11
in Color Correct node 18.7
in Contrast node 18.16
formula used 18.16
matching levels in another image 18.12
pivot 18.16
Contrast node 18.16
Contrast Stretch layer, in Color Correct node 18.11
Control Action menu 18.1
Control Channel menu 18.2
control image
using with filter nodes 18.1
using with Vector Blur node 19.11
using with Vector Warp node 19.6
Control key, using to
open a monitor without cooking image 8.6
select multiple nodes by dragging 5.14
Control Palette buttons in STED 9.7
conversion, frame rate
film fps to video fps 20.2
video fps to film fps 20.5
conversion, image. *See also* Cineon images: conversion of
bit depth
in Bit Expand node 14.2
in Bit Reduce node 14.7
in Linear Remap node 14.21
linear to log, in Disk Output node 12.26
log to linear, in Disk Input node 12.19
conversion nodes 14.1
Convert node menu 14.1
convolution, explanation of 18.26
Convolve node 18.26
Cook button
location in monitor toolbar 8.7
use of 8.4
cooking image data 4.4
Copy command 10.4
coring operation 18.31
Corner Pin node 17.2
correction frame or clip 16.18
cos(val) B.4
cosh(val) B.4
creating new files 5.6
Crop node 17.6
cross-dissolve 15.13
Cubic filter 17.15
cubic function in STED 9.7
cubic() B.6
Cursor Mode option in monitors 8.8
Curve Editor
in Bit Reduce 14.15
in Color Correct node 18.9

custom networks 7.11
Custom Networks command, in Edit menu 10.5
Custom node menu
  adding and removing items 7.11
  location in interface 5.5
  sample networks included in 1.2
Cut command 10.4
Cycle button
  in Disk Input sequencer 12.17
  in flipbook 8.15
  in Sequence node 13.14

D

Dark Intelligence
  in Bit Expand node 14.4
  in Bit Reduce node 14.9
data entry fields
  how to use 6.8
  in STED 9.6
  shortcuts for changing values in 6.8
Data Format Preferences panel 11.8
data stream 4.2
dataflow
  of image data 4.2
  of process commands 4.2
  order of operations in 4.5
dataflow control functions B.6
de-interlacing video frames 20.6
debugging grail scripts 4.3
deg(val) B.5
Degrain node 18.31
Deinterlace node 20.6
Delete All command 10.5
Delete NODE(s) command 10.4
deleting
  node presets from menu 6.11
  nodes 5.14
desaturating an image
  in Color Correct node 18.10
  in Monochrome node 18.22
    formula used 18.22
Diff node 15.11
difference values, used by Ultimatte nodes 16.16
direct I/O 4.10
Directory field in file dialogs 5.6
disconnecting nodes 5.13
Disk Arrays, flipbooks and 11.5
disk cache 4.10
  setting in chal A.3
  setting in Chalice 10.10
Disk Cache parameter 10.11
Disk Input node 12.12
Disk Output node 12.23
disk space
  required by Chalice application files 3.1
  temporary 3.6
displacements, animated 19.6
display buffers. *See* image buffers
Display Unit buttons 14.13
Dissolve node 15.13
distortion effects 19.6
Dither Ramp option 12.11
double buffering
  layout option 10.7
  monitor display option 8.11
doubling frames 13.13
downstream nodes 4.2
DPX image files
  exporting 12.26
  importing 12.19
Draw Mode in Rotospline 16.8
dynamic filtering 17.14

E

ease function in STED 9.6
ease() B.6
easein function in STED 9.6
easein() B.6
easeout function in STED 9.7
easeout() B.6
easeoutp(val) B.6
easep(val) B.6
easinp(val) B.6
edge detection 18.34
  Canny 18.29
  Kirsch 18.34
  Laplacian 18.28
  Prewitt 18.29
  Roberts 18.34
  Sobel 18.34
Edge Mode selector in Rotospline 16.10
Edge node 18.34
edges
  detecting 18.34
    using Convolve node 18.27

edges continued
dilating (growing) 18.36
eroding (shrinking) 18.36
matte, gray/black suppression of 15.15
modifying, in Rotospline node 16.7
smoothing, in Ultimatte AdvantEdge node 15.16
Edit buttons (+, -, =) in STED 9.5
Edit Global Variables panel 10.5
Edit menu 10.4
EDITOR environment 10.12
effects nodes 19.1
email
Chalice-users 1.3
Silicon Grail 1.3
Emboss node 19.4
Enable ROI menu option 8.10
end frame, resetting 5.19
Enter key, using to actuate data field entries 6.4
environment variables
CHALICE_CUSTOM_PATH 7.11
CHALICE_HELP_BROWSER 3.2
CHALICE_TMPDIR 3.6
EDITOR 10.12
HOME 3.4
PLUGIN_DSO_PATH 21.1
Erode/Dilate node 18.36
error value (track correlation) 13.30
Escape key, removing connector lines with 5.13
even field 20.8
even(val) B.5
Exec node 13.5
inputs to, specifying in chal A.2
setting number of slices in 13.7
specifying in chal A.1
Exec notifier panel 13.5
executing a network 13.5
exp(val) B.5
exponential notation, in expressions B.2
exporting
connections from Group nodes 7.8
imagery 12.23
movie formats 12.29
on NT 12.24
nodes 5.8
parameter data 9.9
expressions
displaying results of, in Text node 13.18
guidelines for writing B.1
referencing pixel meter data in 13.10
referencing track data in
to control corner pins 17.2
to drive pixel meter 13.11
using, in Switch node 13.17
Xpresso node
guidelines for writing 19.15
order of evaluation in 19.18
Expressions checkbox in node panel 6.2
Extents parameter 19.13
Extract From conversion method 12.27
eye dropper tool. *See* Color Picker

F

F9 key, using to re-center image in monitor 8.8
Factory Defaults (for node presets) 6.11
Fast Realtime Playback (FRP) 12.27
fconv (frame conversion utility) 2.1
fcp (frame copy utility) 2.1
Fido image files
exporting 12.26
importing 12.19
field dominance 20.3
"field frames" 20.10
field order 20.3
Field Order parameter 20.4
fields, video
extracting odd or even 20.6
interlacing odd and even 20.8
file extensions
.ch 9.9
.grail 5.2
list of, for image formats 12.2
File Info, for input images 12.14
File menu 10.2
File Preview option, in Choose File dialog 12.12
file utilities (fconv and fcp) 2.1
files
creating 5.6
image. *See* image files
merging 5.8
opening 5.6
most recently used 10.3

files continued
saving 5.7
using Autosave 11.10
upgrading to new version of Chalice 5.7
film gamma 12.21
film grain
falloff 18.39
matching 18.37
in bright areas 18.37
in dark areas 18.37
presets for film stocks 18.37
randomizing source frames of 12.17
removing 18.31
from bluescreen backing 16.22
simulating 18.37
size of 18.39
film luminance, as computed in Chalice 13.3
film stocks, matching grain patterns to 18.37
film transfer operations 20.1
Filter Area matte display
in Ultimatte GK node 16.23
in Ultimatte RSC node 16.30
filter nodes 18.1
area filters 18.23
color filters 18.3
filters
anti-aliasing 17.4
blur
in Convolve node 18.26
in Scale node 17.11
in Time Convolve node 19.8
in Vector Blur node 19.13
vector-based 19.11
convolution
explanation of 18.26
in Blur node 18.24
in Convolve node 18.26
in Time Convolve node 19.8
in Unsharp Mask node 18.48
in Vector Blur node 19.11
dynamic 17.14
Lanczos
in Corner Pin node 17.5
in Match Move and Stabilize nodes 17.23
in Transform node 17.15
Mitchell
in Corner Pin node 17.5
in Match Move and Stabilize nodes 17.23
in Scale node 17.10
in Transform node 17.15
super sampling 17.4
two-pass linear 17.4
used for transformations 17.15
used to smooth track data 17.19
window types 17.15
flipbook interface, configuring 11.4
Flipbook Preferences panel 11.4
Flipbook Setup window 8.14
flipbooks 8.14
creating 8.14
playback controls for 8.15
playback of, from disk 11.4
playing audio in 8.16
saving 8.16
flipping images in Rotate Flip node 17.8
floor(val) B.5
Flush All Caches command
in Cache Meter menu 5.4
in Cache Settings panel 10.11
Follow Cooks option 10.9
fonts, PostScript 13.21
formats, image 12.2
four-corner pin 17.2
fps (frames per second) 8.16
fractional multiplication in composite nodes 15.2
Frame Controller
location in interface 5.4
overview of 5.18
using to set key frames 6.10
Frame Icon 5.16
image appears black in 12.4
Frame Range Intelligence
in Bit Expand 14.3
in Bit Reduce 14.9
frame range, resetting
in Disk Input node 12.16
in Frame Controller 5.19
in Sequence node 13.12
frame rates, film and video 20.2
frames
convolving across a range of 19.8
doubling 13.13

frames continued
holding (repeating) 13.13
in Disk Input node 12.17
looping 13.14
in Disk Input node 12.17
missing, using placeholders for in chal renders A.3
navigating 5.18
randomizing, in Disk Input node 12.17
remapping 13.14
rendering in chal A.2
renumbering
in Disk Input node 12.17
in Sequence node 13.12
using fconv utility 2.1
reversing order of 12.17
sequencing 13.12
preparatory to tracking 13.26
skipping 13.13
video (de-interlaced) 20.8
Frames-Per-Second (fps) parameter 8.16
frequency analysis
in Degrain node 18.31
in Sharpen node 18.45
frequency domain
high-frequency changes in 18.31
low-pass filtering in 18.24
FRP files
exporting imagery as 12.27
optimizing output of 12.28
saving flipbooks as 8.16
Full Layout option 10.7
Full Path parameter 12.14
Function menu 9.6
in Curves Editor of Color Correct node 18.10
functions
ch, chf, cht B.7
lprm, lprmf, lprmt B.9
meters B.10
modifying, in STED 9.7
prm, prmf, prmt B.8
track, tracka, tracksr B.11
used in Chalice expressions B.4
data channel access B.7
dataflow control B.6
interpolation B.6
trigonometric B.4
value type B.5

G

Gaffer's tool 19.4
gamma
adjusting
in Color Correct node 18.7
in Gamma node 18.18
formula used 18.18
matching in Ultimatte CC node 16.45
gamma curve 18.18
gamma function in STED 9.7
Gamma monitor display parameter 8.8
Gamma node 18.18
gaps in source image sequences 12.17
garbage mattes 16.5
Gate parameters, Ultimatte 16.42
Gaussian filter 17.15
GIMP 2.1
installing 3.2
GK Alpha matte display 16.30
global variables 6.12
used in Disk Output node 12.23
defining in chal A.2
using in Xpresso node 19.17
gradient, creating in Color node 12.9
grail file 5.1
grail script 4.3
grain. *See* film grain
Grain Killer, Ultimatte 16.22
Grain node 18.37
Green Logic 16.38
greenscreen 16.13
grid pattern 19.20
Group command 10.6
Group menu 10.6
grouping nodes 7.7

H

Hamming window type 17.15
Hanning window type 17.15
HD resolution (YUV files) 12.3
Help menu 10.12
Help, node-specific 6.2
Help Tags, toggling on and off 10.12
High (monitor display) parameter 8.8
high-frequency image content 18.31
highlights, adjusting in Color Correct node 18.7
Histogram utility monitor 8.13

History, wipes using 8.6
HOME 3.4
Horizontal Adapt, in STED 9.9
host code 3.3
hot keys
  auto increment of image buffers 10.9
  buffer persistence 10.9
  clone selected node 10.4
  cook image into buffer without opening monitor 8.5
  create line breaks in text fields 13.20
  create new file 10.2
  create online problem report 10.12
  delete all nodes in workspace 10.5
  delete selected node(s) 10.4
  delete connector line attached to cursor 5.13
  draw straight connector lines in layout 10.7
  edit (cut, copy, paste) 10.4
  export selected nodes to new file 10.2
  follow progress of cooks in layout 10.9
  frame navigation, in monitors 8.3
  full layout view 10.7
  group selected nodes 10.6
  hide icons of selected nodes 10.7
  merge file 10.2
  monitors to front 10.7
  new layout in workspace 10.8
  online help 10.12
  open a monitor without cooking 8.6
  open file 10.2
  open STED window 10.5
  open text editor window 6.6
  pull out of Group node 10.6
  push into Group node 10.6
  quit Chalice 10.3
  re-center image in monitor 8.8
  revert to autosaved file 10.2
  save file 10.2
  save file under new name 10.2
  select all nodes 10.4
  select multiple nodes 5.14
  show icons of selected nodes 10.7
  transform (translate, rotate, scale)
    image, in Transform work monitor 17.13
    shape, in Rotospline work monitor 16.9
    text, in Text node work monitor 13.19
  undo delete 10.5
  undo monitor zoom 8.8
  ungroup selected Group node 10.6
  update all node icon images 10.7
  zoom workspace layout 10.7
HSL layer, in Color Correct node 18.8
hue
  adjusting
    in Color Correct node 18.8
    in Hue Adjust node 18.19
  HSV and HSL colorspaces compared 18.19
Hue Adjust node 18.19

I

I/O, direct 4.10
I/O nodes 12.1
Icon Display Shift value 11.2
if(exp, true_val, false_val) B.6
image buffers 8.4
  button panel for 5.4
image conversion
  comparison of methods for 14.7
  linear to linear
    optimizing 14.21
    using Curves 14.13
image data
  Cineon 10-bit log 4.7
  linear 4.7
  nonlinear 4.7
  representation of, in Chalice 4.6
image file formats
  movie 12.29
  overview of differences among 4.6
  supported by Chalice 12.2
image files
  getting information about 12.14
  naming 12.23
  updating, from disk 12.14
image resolution, setting data preferences for 11.9
image sequence
  changing length of
    in 3:2 Pulldown node 20.3
    in Sequence node 13.13
    in Time Stretch node 13.24
  gaps in 12.17
image slicing 4.11

image stabilization 17.17
images
  blurring
    using Blur node 18.24
    using Vector Blur node 19.11
  Cineon. *See* Cineon images
  compositing 15.1
  desaturating
    in Color Correct node 18.10
    in Monochrome node 18.22
  exporting 12.23
    formatting options 12.24
    FRP movie file 12.27
    on IRIX 12.24
    on NT 12.24
  flipping 17.8
    in Xpresso node 19.20
  importing 12.12
    Maya RGBAZ files 12.21
    YUV files 12.22
    ZPIC files 12.22
  matching
    color levels 16.44
    movement 17.17
  mirroring, in Rotate Flip node 17.8
  premultiplication status of 15.5
  premultiplied
    compositing 15.2
    restoring non-premultiplied values to 14.23
  premultiplying 14.23
  previewing, in Choose File dialog 12.12
  rendering (temps and final shots) 12.23
  rotating
    in Rotate Flip node 17.8
    in Transform node 17.14
  scaling
    in Scale node 17.9
    in Transform node 17.14
  stabilizing 17.17
  tracking 13.25
  translating 17.14
  wiping between 8.5
  writing to disk (rendering) 12.23
importing
  image files 12.12
  parameter data 9.10
Info button 5.17
Info tab 6.4
Input Connector(s) 5.16
input methods (split or combined)
  to Mcomp node 15.4
  to Zcomp node 15.6
inputs, order of, in compositing 15.1
Insert options for Cineon conversion
  Insert into 16-bit 12.20
  Insert into Float 12.20
Inside node 15.10
Inspect monitor panel 8.10
installing
  Chalice 3.2
    custom 3.1
  GIMP 3.2
  Online Help system 3.2
int(val) B.5
Intelligence
  Frame Range 14.9
  Light and Dark
    comparison of algorithms used 14.10
    in Bit Expand node 14.4
    in Bit Reduce node 14.9
  Ultimatte 16.17
"Intelligence: Fix Matte" buttons 16.40
Interlace Mode parameter, in Ultimatte AdvantEdge 15.16
Interlace node 20.8
interleave option, in chal A.2
interleaved processing 13.6
interpolation functions B.6
Inverse node 18.21
Invert Values layer, in Color Correct node 18.12
inverting color values, formula used 18.21
IRIX system requirements 3.1

J

jitter, camera 17.18
Job field, in Chalice file dialogs 5.7
Jot 6.6
Jpeg/Jfif compression 12.28
Jump button in file dialogs 5.6

K

Kaiser A
  in Match Move and Stabilize nodes 17.23
  in Scale node 17.11
  in Transform node 17.15

Kaiser window type 17.15
key (software), obtaining
  on IRIX 3.3
  on NT 3.4
Key Framer device
  location in node panel 6.2
  using to set key frames 6.10
key frames
  creating and editing
    in Bit Reduce work monitor 14.15
    in Rotospline editor panel 16.11
    in STED 9.4
  creating, in node panel 6.10
keyboard equivalents for Chalice commands. *See* hot keys
Kirsch edge detection 18.34
Kodak Cineon 10-bit log imagery 12.3
Kodak Motion Picture & Television Imaging 12.20

L

ladder bar (data entry device) 6.8
Lanczos filters 17.15
Laplacian edge detection 18.28
launching the Chalice application 5.2
layers, disabling 15.5
Layout menu 10.7
lens flare 16.41
Letter Space parameter, in Text node 13.21
Level Indicator
  location in interface 5.4
  using with Group nodes 7.8
levels, matching
  in Contrast Stretch layer 18.12
  in Ultimatte CC node 16.45
license, floating 3.2
license manager
  IRIX 3.3
  NT 3.4
Light Direction parameter 19.4
Light Intelligence
  in Bit Expand node 14.4
  in Bit Reduce node 14.9
lights per stop 18.13
Lin. 90% White 12.21
Line Space parameter, in Text node 13.22
linear data 4.7
linear function in STED 9.6
Linear Remap node 14.21
linear() B.6
Link Straight option 10.7
Linux system requirements 3.1
list parameter functions B.9
loading node presets 6.11
Local Node Menu
  location of button on node 5.15
  opening a monitor from 8.2
  using to change node display 5.17
locked-off shot 16.18
Log 90% White parameter 12.20
log(val) B.5
log10(val) B.5
logical operators B.3
lookup tables (LUTs) for monitor display 8.9
Low (monitor display) parameter 8.8
Low Path parameter 12.16
low-pass filtering 18.24
lprm functions B.9
luminance
  formulae used to compute 13.3
  using to track 13.25
luminance matte 16.4
LUTs for monitor display 8.9
LZW compression 12.32

M

magenta, effects of flare suppression on 16.41
Magnify utility monitor 8.11
Main Menu Bar 10.1
  location in interface 5.3
Make Directories option 12.25
makelic
  ./makelic (IRIX) 3.3
  makelic.exe (NT) 3.4
mask
  creating in Rotospline 16.5
  using an optional input image as 18.1
Match Distance Scale option 17.21
Match Move node 17.17
Match Rotation option 17.21
Match Size Scale option 17.21
math functions B.4
Math menu, in Xpresso node 19.15
matte density formula used by Ultimatte 16.16

matte edges
correcting, in an Ultimatte composite 15.15
modifying, in Erode/Dilate node 18.36
matte nodes 16.1
matte shapes, creating in Rotospline node 16.5
mattes. *See also* alpha channel
combining 15.14
complementary 15.10
creating
in Chromakey node 16.2
in Rotospline node 16.5
in Ultimatte nodes 16.15
garbage 16.5
inverting, in Ultimatte PFG node 16.38
temporary 16.17
Ultimatte, displaying 16.16
max(val1, val2, …) B.5
maximin operation (remove bright noise) 18.41
Maya image files, importing 12.21
Mcomp node 15.4
median rank operation
in Degrain node 18.31
in Rank node 18.43
Medium Path parameter 12.15
memory, consuming, in Protect Cache 6.6
memory segments, shared
explanation of 4.11
increasing number of, on IRIX 3.6
memory usage in Chalice 4.10
Merge command 10.2
Merge node 20.9
merging
a file into currently open file 5.8
video fields 20.9
metacharacters, using in search strings 7.6
meters function B.10
midtones, adjusting in Color Correct node 18.7
min(val1, val2, …) B.5
minimax operation (remove dark noise) 18.41
MinMax node 15.14
Minus node 15.11
mirror image, creating
in Rotate Flip node 17.8
in Xpresso node 19.20
mirroring border pixels in convolutions 18.29
missing frames, replacing with color, in renders A.3
Mitchell B and C parameters
in Corner Pin node 17.5
in Match Move and Stabilize nodes 17.23
in Scale node 17.10
in Transform node 17.15
modulo 19.16
Monitor Auto Update 8.3
changing default state 11.3
monitor interface, configuring 11.2
Monitor Mode, changing 11.3
Monitor Popup Menu 8.10
Monitor Preferences panel 11.2
monitor resolutions, redefining 12.7
Monitor Toolbar 8.7
monitors 8.2
opening 8.2
without cooking image 8.6
resizing 8.8
selecting mode for 8.3
slicing in, resetting default 11.3
updating 8.3
updating region in 8.6
utility 8.11
work 8.1
zooming in and out 8.8
Monochrome layer, in Color Correct node 18.10
Monochrome node 18.22
Most Recently Used (MRU) Files list 10.3
motion blur
for transformations 17.15
in Match Move and Stabilize nodes 17.23
using the Convolve node 18.27
using the Time Convolve node 19.8
using the Vector Blur node 19.11
movement nodes 17.1
Movers node menu 17.1
movie file
exporting images as 12.29
FRP 12.27
Play command 10.2
movie formats 12.4
Movie Save… dialog (in flipbook) 8.16

MPEG compression 12.30
MRU Files list 10.3
multilayer compositing 15.4
Multiple Monitor mode 8.3
Multiply node 15.11
multiprocessing, in chal A.2

N

name field (in node panel) 6.2
naming grail files 5.7
negative image, creating 18.21
neighborhood operator 18.41
networks
  branching 5.12
  how to build 5.1
New command 10.2
90% white card parameter 12.27
node caching, overview of 4.11
Node Connector menu 5.10
node layout, overview of 5.15
node menus
  Composite 15.1
  Convert 14.1
  Effects 19.1
  Filters 18.1
  I/O (Input/Output) 12.1
  location in interface 5.3
  Matte 16.1
  Movers 17.1
  Plugin 21.1
  Tools 13.1
  Video 20.1
node panel
  location in interface 5.3
  overview of 6.2
  parameter tabs in 6.3
node parameters, setting 6.7. *See also* parameter values
Node Preferences panel 11.6
node presets 6.11
nodes. *See also* node menus
  changing display of 5.17
  connecting 5.11
    exporting connections from Group nodes 7.8
  creating 5.10
    from Node Connector Menu 5.10
  deleting 5.14
  diagnostic (tool nodes) 13.1
  disconnecting 5.13
  downstream 4.2
  exporting 5.8
  finding 7.5, 10.5
  frame-number modifiers 4.2
  grouping 7.7
  parts of 5.15
  previewing effect of 5.18
  replacing 5.14
  selecting multiple 5.13
  setting preferences for 11.6
  unique names for 10.4
noise. *See also* film grain
  generating 18.40
  randomizing source frames of 12.17
  removing
    bright 18.41
    dark 18.41
    from bluescreen backing 16.22
    in Degrain node 18.31
    in non-moving background images 19.8
    in Rank node 18.41
Noise node 18.40
noise(x,y,z) B.6
nonlinear data 4.7
Notepad 6.6
NT system requirements 3.1
NTSC luminance, as computed in Chalice 13.3
NTSC video
  modifying color data for 20.12
  pedestal 20.12
  scan rate associated with 20.2

O

odd field 20.8
odd(val) B.5
Offset Frame Numbers parameter 12.17
online help 10.12
Open command 10.2
opening files 5.6
operating systems compatible with Chalice 3.1
operations, order of, in dataflow 4.5
operators (math)
  used in Chalice expressions B.3
  used in Xpresso node 19.18

orphans, filtering
in Ultimatte GK node 16.24
in Ultimatte RSC node 16.35
Output Connector 5.16
Outside node 15.10
Over node 15.9
overwriting files 12.25

P

Packbits compression 12.32
PAL luminance, as computed in Chalice 13.3
PAL video
modifying color data for 20.12
scan rate associated with 20.2
pans, simulating in Transform node 17.12
parameter data, importing and exporting 9.9
parameter functions B.8
parameter tabs 6.3
parameter values
animated, effect of changing end frame 5.19
fractional v. pixel coordinate 12.7
parameters
animating 6.9
enabling expressions in 6.9
Paste command 10.4
pattern generators 19.20
peak point 16.16
Period key, using for frame navigation in monitors 5.18
phase shifts, video transfer 20.2
Phase A 20.4
Phase B 20.4
photographic stops 18.12
Photoshop files 12.4
Pick All Ancestors/Descendents commands 5.13
"Pick" buttons, in Contrast Stretch 18.11
Pivot parameter 17.14
Pixel Bounds menu, in Xpresso node 19.16
pixel meter data, accessing in other nodes 13.10
Pixel Meter node 13.9
pixel values, metering 13.9
in Bit Reduce work monitor 14.15
Play FRP Movie Files command 10.2
playback controls, flipbook 8.16
plug-in nodes 21.1
developer documentation for 10.12
PLUGIN_DSO_PATH 21.1
Plus node 15.12
Point mode, in Rotospline 16.8
polar coordinates 19.19
popup menus in parameter tabs 6.3
position data, generating 13.25
Posterize node 19.7
PostScript fonts 13.21
pow(float base, float exponent) B.5
prediction encoding 12.32
Preferences menu 10.9
preferences, setting
autosave 11.10
data format 11.8
flipbook 11.4
monitor 11.2
node 11.6
proxy 11.7
prefiltering
images to be tracked 13.26
in Sharpen node 18.47
MPEG 12.31
Premultiplication status
changing default setting 11.9
of image layers in Mcomp 15.5
premultiplied image 15.2
Ultimatte 15.3
Premultiply node 14.23
Preserve Input Sort Order parameter 15.7
Presets menu 6.3
preview execute, specifying in chal A.3
Prewitt edge detection 18.29
print-through 16.16
Printer Lights layer, in Color Correct node 18.13
prm functions B.8
problem reporting, from within Chalice interface 10.12
proceduralism 4.1
Processed Foreground, Ultimatte 16.36
processing a grail network 13.5
Protect Cache parameter 6.6
proxy files
activating 12.5
creating 12.15
defined 12.5
disabling 12.6

proxy files continued
matching default monitor resolutions to 12.7
tips for working with 12.7
updating, using Rescan Disk option 12.7
using fractional parameter values with 12.7
Proxy Input menu
location in interface 5.4
using to set global proxy level 12.6
Proxy Preferences panel 11.7
Proxy tab, in Disk Input node 12.15
Pull Out command 10.6
pulldown operation 20.2
pulse(val, min, max) B.5
Push In command 10.6

Q

Quadratic filter 17.15
quiet mode, in chal A.2
quintic function in STED 9.7
quintic() B.6
Quit Chalice command 10.3
quitting Chalice 5.9

R

rad(val) B.4
Radar Map 5.4
Radial (node) menus 11.6
RAM
required by Chalice 3.1
show current usage of 5.4
RAM cache 4.10
setting in chal A.3
setting in Chalice 10.10
ramp, color 12.9
rand() B.6
randomizing frames of input imagery 12.17
Range Slider, in flipbook 8.16
Rank node 18.41
raw data 9.8
realtime playback, of Chalice FRP files 12.27
Red Logic 16.38
reference black level for NTSC video 20.12
Reference Resolutions 11.7
region of interest (ROI), in monitor display 8.6
Remap options for Cineon conversion
Remap from 16-bit linear 12.27
Remap to 16-bit linear 12.19
Remap to float linear 12.19
remapping frames 13.14
rendering 13.5
batch A.1
cancelling 13.5
replacing nodes 5.14
Report Problems command 10.12
rescan options
Rescan All Disk Input Nodes 10.11
Rescan Disk, in Disk Input node 12.14
Resequence Source Frames checkbox 12.16
Resolution popup menu 12.11
restoring deleted nodes 10.5
Revert command 10.2
RGB Density parameter 16.39
RGB Veil parameter 16.40
ripples 19.6
Roberts edge detection 18.34
ROI (region of interest) 8.6
Rosco Labs 16.13
Rotate Flip node 17.8
Rotate Pivot parameter 17.21
Rotate Ref(erence) parameter 17.21
rotating images, in Transform node 17.12
rotation, using track data to match or undo 17.19
Roto Screen Correction, Ultimatte 16.26
Rotospline node 16.5

S

S field, in STED 9.6
sample color, selecting, in Ultimatte 16.19
sampling parameters
in Match Move and Stabilize 17.24
in Transform node 17.16
Save As command 10.2
Save command 10.2
saving
files 5.7
node presets 6.11
SC Alpha matte display 16.29
Scale node 17.9
Scale Ref A and B parameters 17.21
scaling, using track data to match or undo 17.19
scan rates, video 20.2

Scope utility monitor (vectorscope) 8.12
  using with Hue Adjust node 18.20
screen area, in Ultimatte 16.18
screen layout 5.3
Screen Reconstruction, in Ultimatte 16.29
Scroll Bars 5.5
Search Nodes command 10.5
Search Nodes panel, how to use 7.5
seed parameter
  in Grain node 18.38
  in Noise node 18.40
segments, in STED
  editing 9.3
  tying and untying 9.5
selecting multiple nodes 5.13
self-documenting 4.1
sequence filtering, in the Choose File dialog 12.13
Sequence node 13.12
  role in Chalice system 4.3
Sequence tab, in Disk Input node 12.16
sequential processing 13.6
  chal option for A.2
Setup panel (for flipbooks) 8.14
setup.exe 3.4
SG_LM 3.4
shadows, adjusting in Color Correct node 18.7
Shape Limits menu 17.3
Shape mode, in Rotospline 16.9
shared memory
  allocating 4.11
  increasing, on IRIX 3.6
sharing concept 4.9
Sharpen node 18.45
Shift key, using to select multiple nodes 5.14
shortcuts. *See also* hot keys
  adding comments 7.2
  cloning nodes 7.3
  data entry 7.4
    resolutions 7.4
  for accessing node info 7.2
  for changing node display in workspace 7.3
  for finding specific nodes 7.5
  for node creation 7.3
  for node selection 5.13
  grouping nodes 7.7
  navigation
    in monitors 7.4
    in the workspace 7.2
    zoom mode 7.3
  reusing node networks 7.11
  setting preferences to match your work habits 7.2
Show Icon Image preference 11.6
Show My License Info 10.10
Show Network License Info 10.10
Shutter Phase parameter
  in Match Move and Stabilize nodes 17.23
  in Transform node 17.16
Shutter Speed parameter
  in Match Move and Stabilize nodes 17.23
  in Transform node 17.16
sign(val) B.5
Silicon Grail
  contact information 1.3
  website 1.3
sin(val) B.4
sinc filter 17.15
Single Monitor mode 8.3
sinh(val) B.4
Size (tracking) parameter 13.30
skipping frames 13.13
slicing 4.11
  chal option for A.2
  during renders 13.7
  monitor images 4.11
    number of slices to use 11.3
slope handle, in STED 9.4
Slopes menu 14.16
smooth(float val, float min, float max) B.7
smooth(val, min, max) B.5
smoothing
  bumpmap edges 19.2
  edges, in Ultimatte AE node 15.16
  image data 17.19
  in Match Move and Stabilize nodes 17.22
  track data 17.19
Sobel edge detection 18.34
Softclip
  conversion method 14.12
    algorithm used in C.1
  parameter in Disk Input node 12.21

SoftImage 12.22
software key, obtaining
  on IRIX 3.3
  on NT 3.4
Sounds option 10.8
source code
  for plugin examples 21.1
  for UNIX filter examples 13.31
source imagery
  generating (color fields and ramps) 12.9
  loading into Chalice 12.12
  verifying, by rescanning disk 12.14
source nodes 12.1
  Color node 12.9
  Disk Input node 12.12
Space-Time Editor (STED) 9.1
  data entry fields 9.6
  functions 9.6
  graph 9.3
  key frames 9.4
  menus 9.9
  segments 9.3
  waveforms 9.1
  window, parts of 9.2
spatial analysis
  in Degrain node 18.31
  in Sharpen node 18.45
specular highlights, in Emboss node 19.5
Speed (tracking) parameter 13.30
splines
  converting, to and from linear segments 16.10
  drawing and editing
    in Rotospline node 16.8
    in Ultimatte RSC node 16.26
Split node 20.10
sqrt(val) B.5
squash notifier 5.19
Stabilize node 17.17
stabilizing imagery 17.17
stdin (standard input) 13.31
stdout (standard output) 13.31
STED. *See* Space-Time Editor (STED)
STED button 9.1
Stops layer, in Color Correct node 18.12
Stretch Factor parameter 13.24
stretch notifier 5.19
Strip Off Extra Channels 12.25
strobe effects 19.8
suffix filtering, in the Choose File dialog 12.13
super sampling filter 17.4
surface normal
  in Bump node 19.2
  in Surface Normal node 19.3
Surface Normal node 19.3
swapping channels
  in Channel Swap node 13.2
  in Color Correct node 18.10
  in Disk Input node 12.18
Switch node 13.17
System Feedback area 5.5
system requirements of Chalice 3.1
system variables
  used in Chalice expressions B.2
  used in Xpresso node 19.17

T

T field, in STED 9.6
tabs, parameter 6.3
tan(val) B.5
tanh(val) B.5
teamwork, using Chalice to promote 4.9
telecine-type transfers 20.2
temp comps 12.23
  executing 13.6
  quality of 13.7
  resolution of 13.7
  work monitor display of 17.12
temporary disk space requirements 3.6
temporary matte 16.17
Temps Output tab 12.25
Temps Path parameter 12.26
text editors, invoking 6.6
Text node 13.18
3:2 Pulldown node 20.2
3:2 Pushup node 20.5
Time Convolve node 19.8
time mark, in STED 9.4
Time Stretch node 13.24
Title Bar
  Chalice 5.3
  node 5.15
tool nodes 13.1
toolbar, monitor 8.7
top input, composite nodes 15.1

track data
generating 13.25
smoothing 17.19
using to
control position of corner pin 17.2
drive pixel meters 13.11
match movement 17.17
stabilize imagery 17.18
track function B.11
Track node 13.25
"Track Node" parameter 17.20
track points 13.29
disabling 13.30
tracka function B.11
tracking
auto 13.25
manual 13.27
minimum source processing 13.28
noisy imagery 13.26
non-sequential frames (with gaps) 13.26
parameters 13.28
path (drawing in monitor) 13.28
points 13.28
color of 13.28
repeating (re-tracking) 13.26
selected points and frames 13.26
setting frame range to track 13.29
tips for creating a good track 13.25
tracking, directional, for bluescreen reconstructi 16.32
tracksr function B.11
trailing effects 19.8
Transform node 17.12
transforming
imagery, in Transform node 17.12
using arrow keys 17.13
shapes, in Rotospline node 16.9
using arrow keys 16.9
text, in Text node 13.19
Transparency parameter in Mcomp layers 15.5
triangle (linear) filter 17.15
trigonometric functions B.4
trunc(val) B.6
truncate notifier 5.19
tuners, how to use 6.7
turb(x,y,z, depth) B.7
tutorials, Chalice 1.1
two-pass Lanczos filter 17.5
two-pass linear filter 17.4
two-pass Mitchell filter 17.5
tying segments, in STED 9.5
typefaces 13.21

U

UI Heap 5.4
Ultimatte AE (AdvantEdge) node 15.15
"Ultimatte Blue" paint 16.13
Ultimatte CC node 16.44
Ultimatte CSC node 16.18
Ultimatte GK node 16.22
Ultimatte Intelligence 16.17
Ultimatte nodes
how matte data is generated in 16.15
how to use in network 16.13
introduction to 16.13
Ultimatte PFG node 16.36
Ultimatte RSC node 16.26
Undo button (in node panel) 6.3
Undo Delete command 10.5
Undo Distance Scale option 17.21
Undo Rotation option 17.21
Undo Size Scale option 17.21
Ungroup command 10.6
UNIX filename conventions 5.7
UNIX filters 13.31
source code examples 13.31
UNIX node 13.31
Unpremultiply node 14.23
Unsharp Mask node 18.48
untying segments, in STED 9.5
Up button in file dialogs 5.6
Update Parameters if Input Changes 12.17
upgrading existing files 5.7
User Defaults (for node presets) 6.11
utility monitors 8.11

V

V field, in STED 9.6
validate, using Exec node to 13.6
value point, in STED 9.4
value type functions B.5
variables
environment. *See* environment variables
global. *See* global variables
system. *See* system variables
Vector Blur node 19.11

Vector Warp node 19.6
vectorscope display 8.12
Version command 10.12
Vertical Adapt, in STED 9.8
vertical flip
  in Disk Input node 12.14
  in Disk Output node 12.25
  in Rotate Flip node 17.8
  UNIX filter example 13.31
vertical scale of Bit Expand graph 14.5
video
  broadcast standards for 20.12
  converting
    from 30 fps to film (24 fps) 20.5
    to 30 fps from film (24 fps) 20.2
  extracting odd or even fields from 20.6
  interlacing fields of 20.8
  merging split fields of 20.9
  modifying color data for 20.12
  scan rates 20.2
  splitting into fields
    odd and even, alternating 20.10
    odd or even 20.6
video nodes 20.1
Video Safe node 20.12

W

wakes 19.6
Waveform utility monitor 8.12
waveforms (representing parameter data) 9.1
wildcard characters, using in search strings 7.6
wipes between images 8.5
Wireframe display option, in Rotospline node 16.12
Work Heap 5.4
work monitor
  in Bit Expand node 14.4
  in Bit Reduce node 14.9
  in Chromakey node 16.2
  in Color Correct node 18.5
  in Corner Pin node 17.3
  in Crop node 17.6
  in Match Move node 17.20
  in Pixel Meter node 13.9
  in Rotospline node 16.5
  in Stabilize node 17.20
  in Text node 13.19
  in Track node 13.28
  in Transform node 17.12
  in Ultimatte AE node 15.15
  in Ultimatte PFG node 16.37
  in Ultimatte RSC node 16.26
work monitors 8.1
  Auto Update, changing default state 11.3
  changing color of tools in 11.3
  in Ultimatte nodes, introduction to 16.17
Workspace 5.5
wrap(val, min, max) B.6
Writing Plug-ins command 10.12

X

Xpresso node 19.15
  guidelines for using 19.18
XY Bounds menu, in Xpresso node 19.16
XY coordinates, tracking 13.30

Y

YIQ conversion, in Video Safe node 20.12
YUV conversion
  in Video Safe node 20.12
  options for, when importing 12.22

Z

z-depth data
  order of 15.7
  using to composite 15.6
Zcomp node 15.6
Zip compression 12.32
Zip Level parameter 12.32
zone plate 19.20
zoom effect, simulating in Transform node 17.12
zooming
  monitor image 8.8
    using F9 key to undo 8.8
  workspace view 10.8
ZPIC image files 12.22