Composite

Composite User Guide

Composite 2011

Trademarks

Disclaimer

Published by: Autodesk, Inc.
111 McInnis Parkway
San Rafael, CA 94903, USA

Title: Autodesk Composite 2011 User Guide

Document Version: 2

Date: March 11, 2010

Contents

Preface

1

Topics in this chapter:

About Composite

Composite is Autodesk's interactive solution for film compositing. It brings compositing and visual effects tools to high-end 2D artists working at HD, 2K, 4K or above, in film, television, and design. It features a full High Dynamic Range (HDR) pipeline, next-generation 3D compositing, high performance interaction at extremely high resolutions, and advanced rendering. It primarily features a database-less workflow for data management, one Composite seat, one scripting license, and unlimited rendering capabilities.

The Composite creative toolset includes keying, color correction, tracking, camera mapping, paint, spline-based warping, motion blur, and depth of field. Furthermore, it can be updated with extensions that add functionality to the product by means of new tools and core fixes.

About This Guide

The Composite User Guide is made up of two major sections: Getting Started and Reference.

Getting Started

This section describes the basic information you needs to know in order to get up and running with Composite quickly. Topics ranging from key interface features and accessing the Help provide you with everything you need to start compositing now! It is strongly recommended that you read this section before exploring the Reference portion of the User Guide.

Reference

This section provides in-depth information about all the tools and key features associated with Composite. Some features discussed in the Getting Started section are described in further detail here. The reference section includes advanced topics and is most useful for users who have a good understanding of Composite basics.

Getting Help

There are two ways to access Help for Composite, depending on whether you want more information on a particular subject or whether you need assistance with a specific problem

Viewing Online Help

Autodesk provides documentation in an HTML help system displayed in a Web browser. The Help is automatically installed with the software.

To launch the Help, do one of the following:

1. From the Composite menu, choose *Help* > *Composite Help*.
2. From the Windows taskbar, choose *Start* > *programs* > *Autodesk* > *Autodesk Composite 2011* > *Composite Documentation* > *Online Help*.

The Help appears in the browser window.

Contacting Support

For further assistance, please contact Technical Support at the following address: *http://www.autodesk.com/subscriptionlogin*.

Part 1: Getting Started

Getting Started includes these chapters:

About Getting Started

2

Topics in this chapter:

About The Chapters

The Getting Started section describes fundamental Composite concepts and explains some of the tool's basic functionality, which should allow you to begin using Composite as quickly as possible. Here is a brief description of the chapters included in this section.

Interface Fundamentals

This chapter provides the first detailed descriptions about some essential Composite interface features. Composite interface mechanics like the Gate UI and the Pick List, as well as critical functions such as zooming and panning, and working with nodes are explained. You'll find more in-depth information about some of the features discussed here, like The Player, in the Reference section.

Start Compositing Now! A Simple Workflow Overview

This chapter provides you with a step-by-step example of how to composite using Composite. The goal is to demonstrate how easily you can begin to use the tool, even before you explore some of the more advanced topics covered in the Reference section.

Interface Fundamentals

3

Topics in this chapter:

Basic Interface Functions

An intuitive set of editing functions are available to manipulate elements, such as compositions, folders, and so on. These include basic functions for cutting, copying, pasting, and deleting, as well as more advanced functions.

Mouse, Pen, Keyboard

You can interact with the user interface using a mouse or a pen and graphics tablet, in concert with the keyboard. If you have a mouse wheel, you can use it to scroll through menus.

NOTE To generate a right-click with a pen, press the **`Context Menu`** key in concert with the pen. If a key board includes a **`Context Menu`** key, it is typically located three keys to the right of the **`Spacebar`**.

Cutting, Copying and Pasting

Standard cut, copy, and paste commands let you manipulate elements, such as composition and folders.

To cut or copy, then paste an element:

1. Right-click the element and select Cut (**`Ctrl + X`** for Windows, Linux or **`Cmd + X`** for Mac OS) or Copy (**`Ctrl + C`** for Windows, Linux or **`Cmd + C`** for Mac OS).
2. Navigate to the location where you want to paste the element. Then right-click the location and select Paste or press (**`Ctrl + V`** for Windows, Linux or **`Cmd + V`** for Mac OS).

Entering Text

Tools such as the Slate and Notes tools require that you type into text editor fields.

To accept changes in a text editor field, do one of the following:

- Press **`Enter`** in the case of a single-line editor.
- Press **`Ctrl + Enter`** in the case of a multi-line editor.
- Click outside the editor.

To revert to the previous text:

- Press **`Esc`**.

For more text editor/field hotkey combinations, see Text Editor and Text Fields on page 820.

Dragging and Dropping

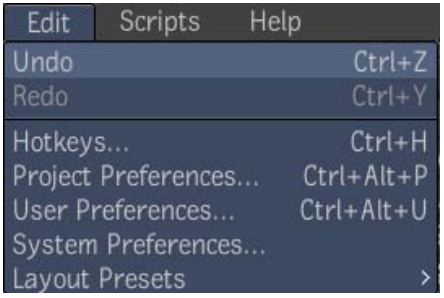
You can drag and drop the contents of most folders in the user interface. In the user preferences, you can choose whether you see the ghosted image of the composition or not. See Setting User Preferences on page 168.

Undoing and Redoing

You can undo most operations. You can also redo any operation you undo.

To undo or redo an operation, do one of the following:

- From the Edit menu, select Undo (or press **`Ctrl + Z`** for Windows, Linux or **`Cmd + Z`** for Mac OS). Or select Redo (or press **`Ctrl + Y`** for Windows, Linux or **`Cmd + Shift +Z`** for Mac OS).



- Or, in the taskbar, at the bottom of the UI, locate the Undo icon. The last action performed is displayed beside it. Then do one of two things:
  - To undo the last operation you performed, click the arrow next the Undo icon.

  - To undo any operation other than the last one you performed, or to redo an operation, click the text to the right of the Undo button, and

select the operation you want to undo or redo from the list. Note that the arrow that appears in the list indicates the most recent operation you performed. This operation and all operations above it are operations you can undo. All operations below the arrow are ones you can redo.

Things to Remember:

- You set the maximum number of items in the undo/redo list in the user preferences General tab.
- Browsers have a separate undo/redo list.
- Press **`Ctrl + Y`** for Windows and Linux, or **`Cmd + Shift + Z`** for Mac OS to redo an operation.

Deleting Elements and Compositions

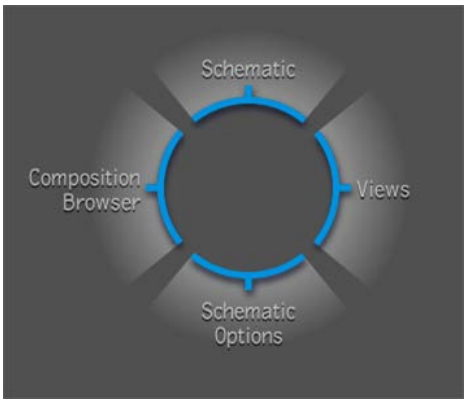
You can delete an element, such as a composition or folder.

To delete an element do one of the following:

- Select the element and press **`Delete`**.
- Right-click the element and select **`Delete`**.

The Gate UI

The Gate UI allows you to access any of the four hidden panels in the workspace. The Gate UI resembles a compass; it has four gates that each correspond to one of the hidden panels. The gates are distributed along the four cardinal directions.



The four gates are as follows:

Gate:	Displays:
a) North gate	The Schematic overlay. This is a semi-transparent, larger view of the Schematic. You can set the transparency of the background and nodes in the overlay, in the Creative tab of the User Preferences.
b) East gate	Pick List, Tools, or Views tab (depends on the last chosen option). You can drop views onto viewports, add tools to compositions, and access the Pick List -- see Tools, Views & Pick List tabs on page 30.
c) South gate	View options for the view on which the cursor is currently paused. For example, if you access the Gate UI when the cursor is over a Player, the south gate will display the Player options.
d) West gate	The Composition Browser.

To access the Gate UI:

1. Press the tilde (~) key or middle-click your mouse. The Gate UI appears.
2. Swipe north, south, east or west. The panel associated with that gate appears.
3. To exit the panel. move the cursor away from the panel.

NOTE If you press the tilde (~) key, or middle-click and swipe simultaneously, the Gate UI is not displayed. If you press the **Alt** key while swiping, the panel will stay open until you press **Esc**.

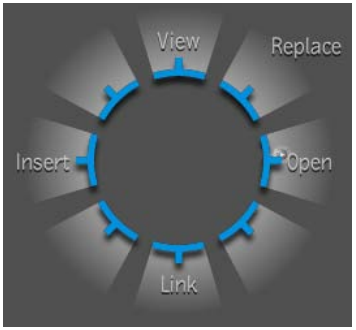
The Drop Gate

Similar to the Gate UI in appearance, the Drop Gate appears when you drop a composition or a tool into a Player, Layer Editor, or Schematic view. The available options for the Drop Gate depend on whether you are dropping a tool or composition, the view you are dropping it in, and also what node is currently selected in the dependency graph. For example, if you drop a composition on a Reaction node in the Schematic view, the Composition Drop Gate appears.

Here are the options associated with the Drop Gate when used in conjunction with a composition or a tool.

To use the Drop gate when dropping a composition, do the following:

1. Either drop a composition into the Player or Layer Editor. Or, drop a composition onto a tool in the Schematic view.



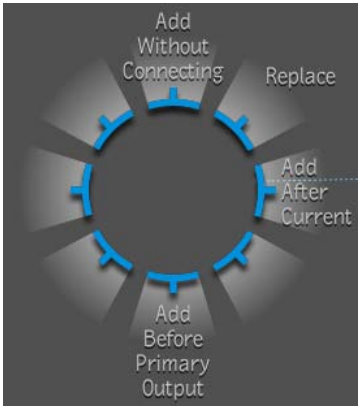
2. Then swipe through the appropriate option to select it, or click to cancel the drop.

NOTE If you drop the composition or tool, and swipe to select an option simultaneously, the option is executed but the Drop Gate is not displayed.

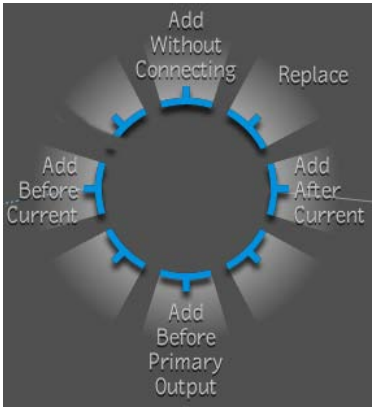
To use the Drop gate when dropping a tool do one of the following:

1. Drop a tool into a Player, Composition Browser (with no composition or tool selected), or Layer Editor (with no composition or tool selected).

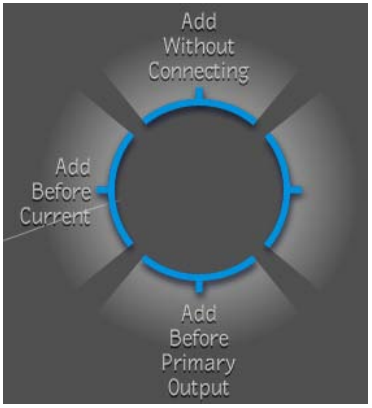
2. Or, drop a tool onto a selected composition in the Schematic view or the Composition Browser. The Tool Drop Gate appears.



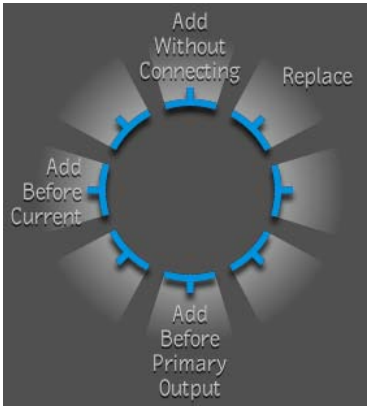
3. Or, drop a tool onto a selected tool in the Schematic view or the Composition Browser.



4 Or, drop a tool onto a primary output node in the Schematic view or a selected primary output node in the Composition Browser.



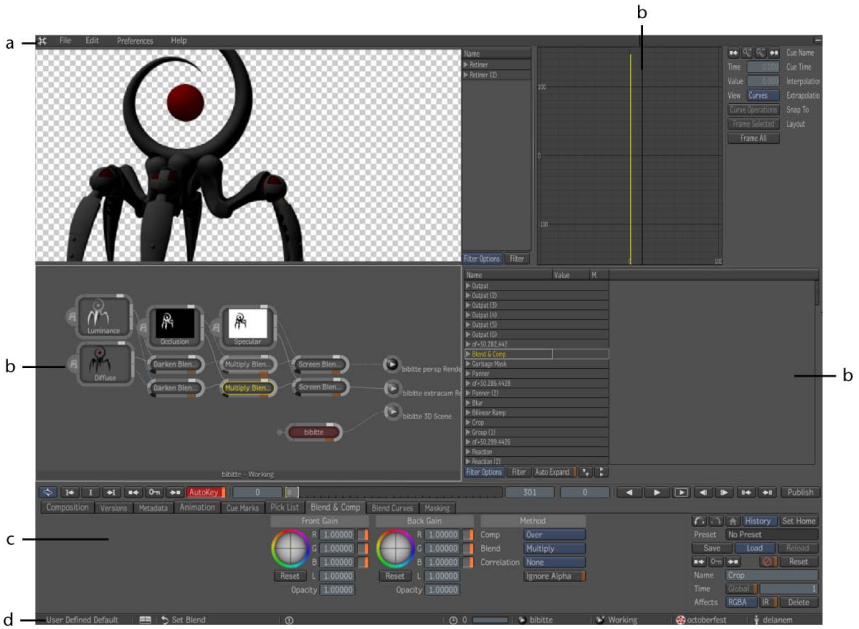
5 Drop a tool onto a secondary output node in the Schematic view or a selected secondary output node in the Composition Browser. The Tool Drop Gate appears.



6 After you have performed one of the preceding actions, swipe through the appropriate option to select it, or click to cancel the drop.

The Composite Interface

The Composite interface may include a variety of views and elements, all of which are designed to ensure a seamless and intuitive workflow. Each preset contains views typically used to perform particular tasks.



Views and elements include:

a **Menu Bar** This is always visible by default and cannot be modified. See The Menu Bar on page 19.

b **Viewports** There can be one or more viewports in the Composite interface, depending on the currently selected preset or on a user-defined layout. You can populate the viewports with views such as the Player, the Schematic, and Browser views. See Viewports and Views on page 20.

c **Tool UI** Displays information about the current tool or composition, or the current view when accessed from the Gate UI. You can edit information, or set options and attributes in this view. It is composed of four main sections: Animation controls, Player controls, Node-Independent tabs and Node-Specific tabs. See The Tool UI on page 23.

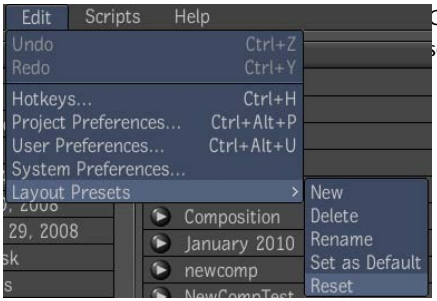
d **Taskbar** This is always visible by default and cannot be modified. You can, however, show/hide the taskbar by pressing **`Ctrl + T`**(for Windows and Linux) or **`Cmd T`**(for Mac OS). See Taskbar on page 29.

The Menu Bar

The menu bar appears at the top of the Composite window. It contains the following menus: File, Edit Preferences, and Help. There is also a minimize button at the far right.

To show or hide the menu bar: press **`Ctrl + M`**.

Use:	To:
	Open new or existing projects, close or save projects, import projects, archive or render projects, create new versions, perform pre-compositing, import projects, or exit projects.
	Undo, redo actions, access the Hotkeys, Project and User preferences and the Layout presets.

Use:	To:
	Create, delete, rename, reset layout presets. Or set a layout preset as default.
	Allows you to load or reload scripts.
	Access Composite help or documentation, Python scripting info, Learning Movies or Tutorials, the Support Centre, Release Notes, The AREA, the Hotkeys doc, Scripting, Token Substitutions and Initialization Variables, and About Composite.
	Minimize the Composite window

Viewports and Views

The Composite UI is composed of one or more viewports and populated with one or more views.

Viewport

A viewport is simply a container in which you can drop a view into. You can modify a viewport's layout using the Layout icon in the taskbar. You can change a viewport's size or drag it to another location by placing your cursor over a viewport border. See Taskbar on page 29.

Views

You can access all available views from the Views tab, using the Gate UI and then dragging a view into a viewport.

NOTE You cannot undo a drag view operation. If you frequently add or remove specific views, you can add them to the Pick List for even faster access.

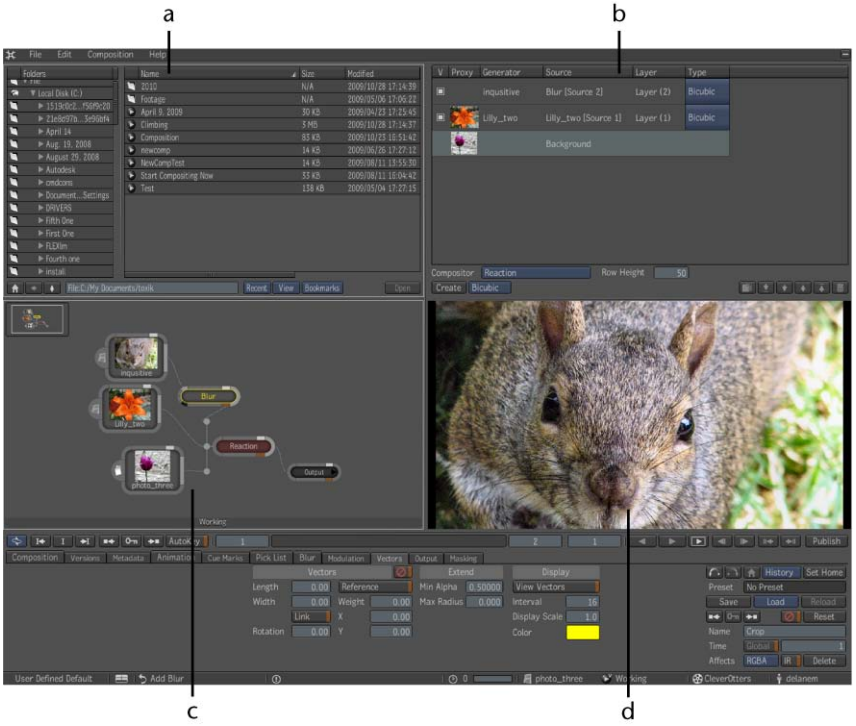
A view can be:

1. A view where you work on or view your composition;
2. A browser where you work with files.

The following views are available:

View	Description
Animation	Allows you to edit animations.
Composition	Allows you to view and to work with all the tools and parameters in a composition in a browser format. See The Composition Browser on page 48.
File Browser	Allows you to create compositions by importing media and provides a view of the folders that your installation references.
Layer Editor	Allows you to add, edit and reorder layers associated with one or more Reaction compositing nodes.
Pick List	Allows you to store frequently used tools, views and compositions. See Pick List on page 141.
Player	Allows you play a composition. The Player options determine what appears in the Player. See Player on page 153.
Schematic	Allows you to build a dependency graph for a composition. See Working with Schematic on page 50.

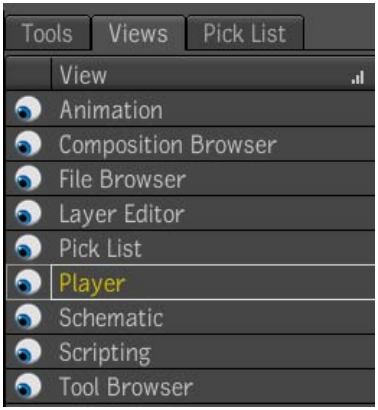
View	Description
Scripting	
Tools	Allows you to add a tools browser to any viewport.



Four Views: a) File browser b) Layer Editor c) Schematic d) Player

To add a view to a viewport:

1. Press the tilde key or middle-click your mouse to display the Gate UI, then swipe through the east gate to display the Tools, Views and Pick List tabs.
2. Select the Views tab.

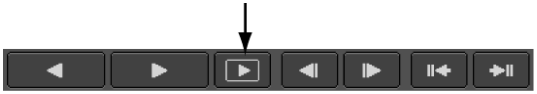


3 Select a view and drag it into a viewport.

To display a view in a fullscreen view:

1 Pause over a view you want to display as a fullscreen view and press the **F** key.

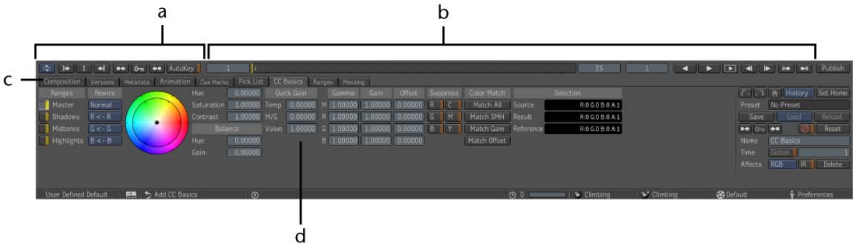
2 In the Player controls, click the fullscreen button to play the composition.



The view displays as a fullscreen view. Press **F** to exit the fullscreen view.

The Tool UI

The Tool UI is located at the bottom of the user interface, just above the task bar. It allows you to view information about the composition which is currently open, as well as view and adjust settings for any of the tool nodes in that composition.



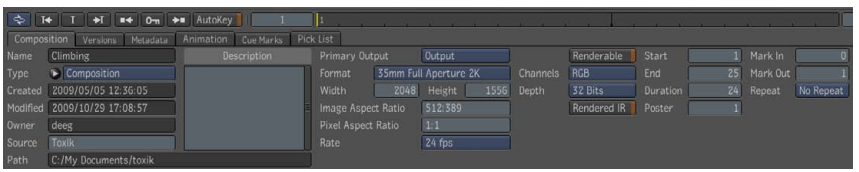
The Tool UI has four main areas:

Item:	Description:
a) Animation controls	Controls for setting and working with cue marks and keyframes. See Keyframing Workflows on page 674.
b) Player controls	Controls for playing media associated with a composition, in any of the views. See Using the Player Controls on page 36.
c) Details or node-independent tabs	A set of tabs that contain information about the composition and controls that are relevant for any of the nodes in the composition.
d) Tool UI or node-specific tabs	A tab that contains information and controls specific to the currently selected node in the composition.

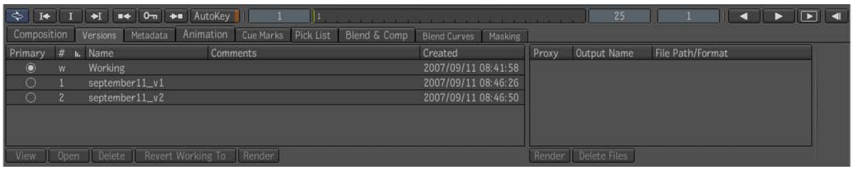
Node-Independent Tabs

There are several node-independent tabs, including (from left to right in the user interface): Composition, Versions, Metadata, Animation, Cue Marks and Pick List. Here is a short description of each tab:

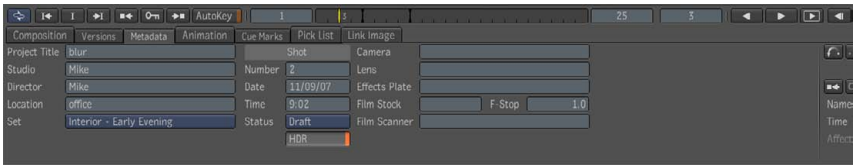
Composition: Displays information about the composition you currently have open.



Versions: Displays a list of all current versions, including the working or current version, associated with the composition you currently have open.

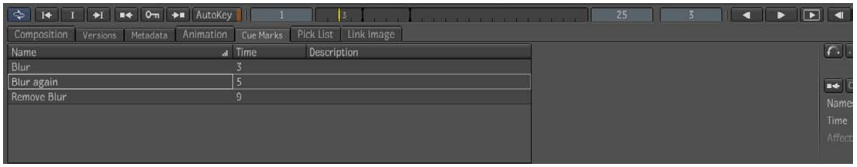


Metadata: Displays the description of the composition you currently have open, along with any custom attributes you have added. You can use this tab to edit the data associated with an individual field.

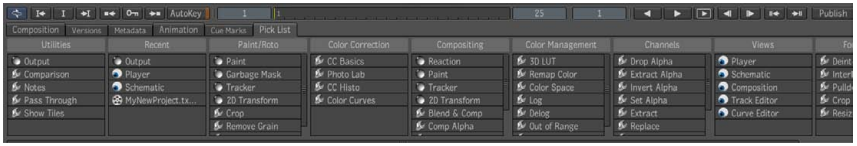


Animation: Displays controls for animating individual variables in the composition. See The Animation Tab on page 661.

Cue Marks: Displays the list of cue marks set for the composition you currently have open. See Working with Cue Marks on page 668.



Pick List: Provides a place for storing frequently used tools, view and compositions. See Working with the Pick List on page 142.

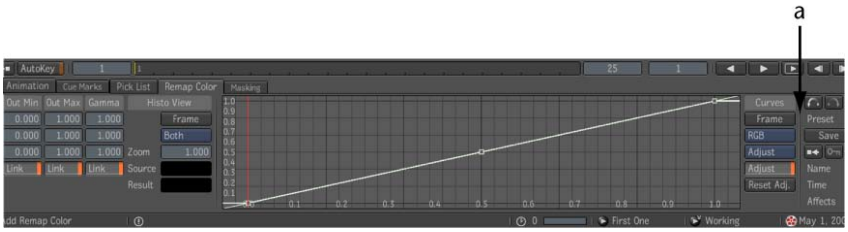


Node-Specific Tabs

One or more node-specific tabs appear when you select a node in a composition. For example, if you select the CC Histo tool, you'll see that it has a main tab, as well as a Ranges tab. You will typically select a node from the Schematic view, from the Composition Browser or from the Layer Editor context menu.

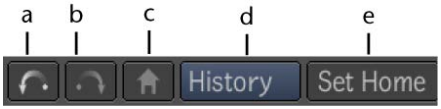
Each composition node has a tab associated with it. In the case of imported media, the composition node for the composition you currently have open has an Image Import tab that contains the same information as in the Import browser when you imported the media to create the composition -- see Importing Media on page 115. The composition node for a linked composition has a Link Image tab that contains a list of the rendered output associated with that composition -- see Linking Compositions on page 234.

A vertical bar in the tab indicates more options may exist underneath the area to the right of the bar. Drag right on the bar to expose the options.



The contents of most tool node tabs are organized into left and right areas. The left hand area contains Player-related controls and direct manipulation controls unrelated to the Player (for example, in the Diamond Keyer, the color prism), and fields and widgets you can use to set or enter values. The area on the right is the Tool Options area; these options are common to all tools.

The top row is used for selection history and navigation, and contains the following controls:



(a) Navigate to previously selected node (b) Navigate to next selected node (c) Go to home node (d) History (e) Set Home

a **Navigate to previous node** Navigate nodes before current selection.

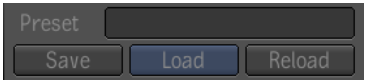
b **Navigate to next node** Navigate to nodes selected after current selection.

c **Go to home node** Navigate to node set as Home node.

d **History** View entire node list and select a node.

e **Set Home** Set a home node from the history list. To set another node as a home node, select a node name and click **`Set Home`**.

The next set of controls is for using Tool presets. See Tool Presets on page 146.



(a) Preset (b) Save (c) Load (d) Reload

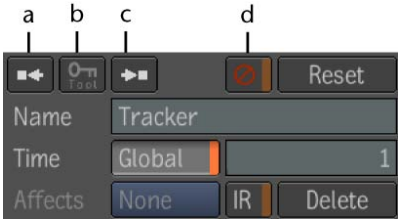
a **Preset** Displays the name of the tool preset currently loaded in a selected tool node.

b **Save** After customizing a tool, you can save it as a tool preset.

c **Load** Displays a list of tool presets.

d **Reload** Reloads the currently selected tool node with the same tool preset. This lets you revert back to the parameters contained in the tool preset.

The lower portion of the Tool Options area contains the following controls:



(a) Back One Keyframe (b) Set Keyframe (c) Forward One Key (d) Mute (e) Reset (f) Name (g) Time (h) Affects (i) IR (j) Delete

a **Back one keyframe** Move back one keyframe in the currently selected animation curve for this too node. For more information on setting the animation keys, see Keyframing Workflows on page 674.

b **Set keyframe** Set Keyframe: set a keyframe in the animation editor for this tool node. For more information on setting the animation keys, see Keyframing Workflows on page 674.

c **Forward one key** Move froward one keyframe in the currently selected animation curve for this tool node. or more information on setting animation keys, see Keyframing Workflows on page 674.

d **Mute** Mute the currently selected tool node. This is a convenient way to toggle the effect of that node on or off. When you mute a tool node, the node has no effect on the input it receives; it simply passes that input to the next tool node in the dependency graph. If you mute a tool node that accepts multiple inputs, the node passes its primary input to the next too node.

NOTE Note that muting a tool node may not always be equivalent to removing it from the dependency graph. That is, if the input and output formats of a tool node are different, removing the tool node may cause format conflicts in nodes further along the dependency graph, but muting the tool would not cause format conflicts. For example, if the format of the output is different from that of the primary input, the muted node adjusts the format, possibly in a tool specific way. For example, a muted Diamond Keyer node outputs a solid white matte, a muted Noise node outputs a black image. You cannot mute a supertool.

---

e **Reset** Resets all parameters of this tool node to their default values.

f **Name** Examine or edit the name of the currently selected tool node. To edit the name, click in the field, enter your edits, and then press Enter.

g **Time** Displays the global and local time in a node affected by a Time Offset tool. Enables Global to display the global time in the field to the right of the Global button. Disables Global to display the local time of the node in that field.

h **Affects** Specifies the channels (RGB, RGBA, or A) the currently selected tool uses. The options available depend on the currently selected tool. In some tools, this options may not be available.

i **IR** Allows the currently selected tool node to create an intermediate result whenever you play the composition. You can also enable or disable intermediate results in the Schematic, by clicking the IR tab in the bottom-right corner of the node icon; this tab is orange when you enable IR. See Caching on page 226.

j **Delete** Delete the currently selected tool node.

Taskbar

The taskbar anchors the user interface and gives you immediate and consistent access to frequently-used core features and commands, such as presets, undo, warning messages, as well as the current composition, version, project, and user. The taskbar is located at the bottom of the desktop.

To show or hide the task bar, press **`Ctrl + T`** on Windows and Linux, or **`Cmd + T`**on Mac OS.

The following describes each element in the taskbar, from left to right.

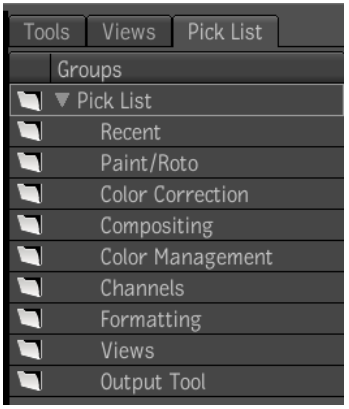
Click:	To:
	Display the Preset list. You can choose from preconfigured presets or create your own.
	Select a different layout for the current preset. Click the icon and select a layout.
	Undo or redo an operation. Click the arrow icon to undo the last operation in the list, or click the text beside the arrow to display the list of operations performed. The arrow that appears in the list indicates your current position in the undo list; all operations above it are operations you can undo, and all operations below are ones you can redo. In the User Preferences window (General tab), you can specify the number of undo levels.
	View messages. When messages are issued, the icon color reflects the severity: white, yellow, or red. Click the message area to display a list of messages received to date. In the User Preferences window (General tab), you can specify the type of messages that are displayed and the maximum number of items in the User Preferences.
	Start the render queue monitor or view the progress of a render. Click the icon to start the render queue monitor. Click the number and select a background rendering task

Click:	To:
	from the list that appears; the number indicates how many render tasks in the list are currently in progress. The progress bar to the right of the render task name in the list shows the task's progress: a blue bar is displayed as the task progresses and is completed successfully. A gray bar is displayed if the render task failed or was cancelled (the text "failed" is also displayed if a failure occurs). To remove a failed or completed render task from the list, select it, and then respond to the prompt for removal. To cancel a task that is in progress, select it, and then respond to the prompt for cancellation. To delete a task in progress, first cancel it, and then delete it.
	Displays the list of compositions that are currently open. The name of the current composition is displayed beside the icon. If the user opens a composition version, the version name appears to the right of the composition name. The gray icon represents a read-write version, while a yellow icon represents a read-only version. To switch to a composition (or version), select it from the list.
	Open the Project Preferences window. The current project is displayed beside the icon.
	Open the User Preferences window. The user name is displayed beside the icon.

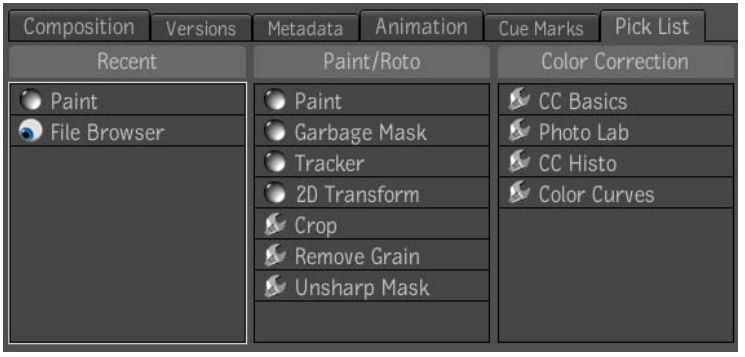
Tools, Views & Pick List tabs

Tools and Views tabs display all the available tools you can add to compositions, as well as the views you can display. Tools and views are stored in their respective folders. When you click a tool category folder, the tools are displayed in the lower portion of the panel.

The Pick list tab provides you with easy access to frequently used tools, view and other elements. Each user has a Pick List. Its contents are independent of any project and are consistent across sessions. The Pick List exists as a tab in two locations: a) through the east gate of the Gate UI and b) the tool UI.



a) Pick List tab



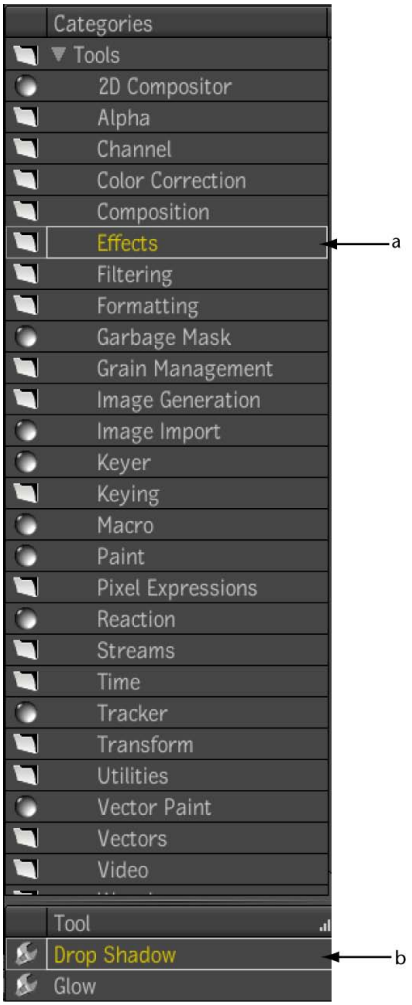
b) Pick List in Tool UI

To display the Tools, Views and Pick List tabs:

1. Middle-click or press the tilde (~) key to display the Gate UI.
2. Swipe through the east gate and select one of the tabs.

To drag an item from the Tools or Views tab:

➤ In the Tools or Views tab, select an item from the lower portion of the panel, and drag it into a view. If you drag a tool into the Schematic view, you can drop it onto any connection to insert it in the composition.

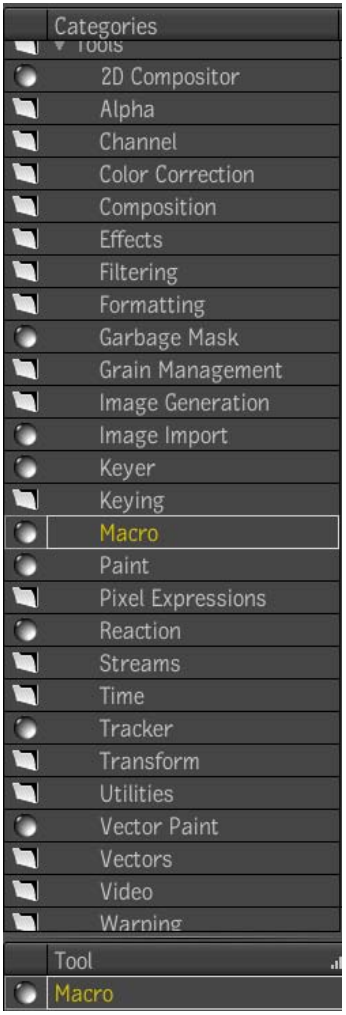


a) Tool Category b) Tools

For more information on views, please see Viewports and Views on page 20.

For more information on the Pick List, please see Pick List on page 141 in the Getting Familiar with Your Workspace chapter.

In addition to regular tools, there is a specific category of tools known as Super tools. A super tool is a tool that is composed of a set of tools. Super tools include Macro, Reaction, Garbage Mask, 2D Compositor, Image Import, Keyer, Tracker, and Paint. Super tools are indicated by a sphere icon.



For more information on each of these super tools, please see the following:

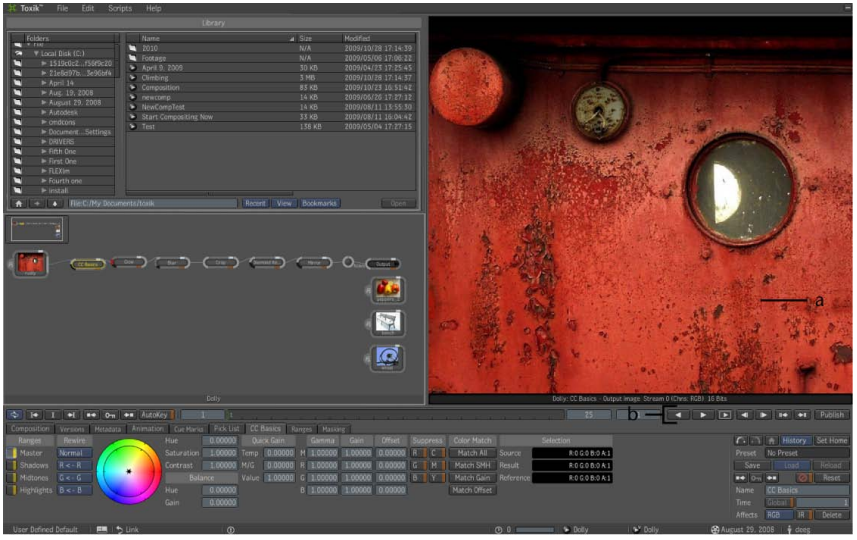
- The Macro Super Tool on page 699

- Reaction Compositing and Effects on page 240
- Garbage Mask Tool Tabs on page 538
- 2D Compositor on page 339
- Using the Image Import Tool on page 125
- Keyer Super Tool on page 522
- About Tracking and Stabilizing on page 754
- About Raster Paint on page 567
- Vector Paint on page 595

The Player

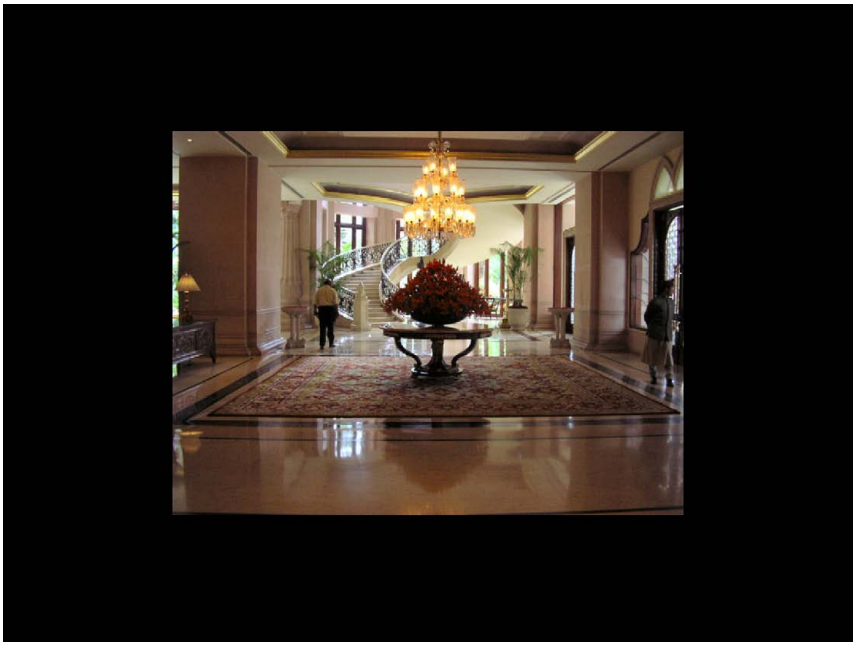
The Player is used to play or preview a composition, or preview media. There are three different versions of the Player: the Player View, the Mini-Player and the Fullscreen Player.

- The Player View: Allows you to play compositions and view the changes you make to compositions as you work. You can drag a Player View from the Views tab and drop it into a viewport.



- The Mini-Player: Allows you to preview compositions located in the Footage folder or on your Desktop. See The Mini-Player on page 211.

- The Fullscreen Player: This version of the player is useful when you want a more immersive environment. You can use it to play compositions, work on compositions, and preview media. See The Fullscreen Player on page 213.



Using the Player Controls

The Player controls are shared by all Player views. The Start Frame, End Frame, and Current Frame numbers in the Player controls reflect those of the currently selected Player.

The Player controls allow you to:

- Play a composition
- Set the repeat mode for the playback
- Scrub through the composition
- Adjust the start or end frames of the composition
- Set a cue mark
- Move to an existing cue mark
- Delete a cue mark
- Set a keyframe manually
- Move to an existing keyframe
- Turn Autokey on or off

To play a composition:

Click:	To:
	Play the composition backward.
	Play the composition forward.
	Enter the fullscreen Player. Press the **F** key to return to the Player view.
	Step backward one frame.
	Step forward one frame.
	Move to mark in.

Click:	To:
	Move to mark out.

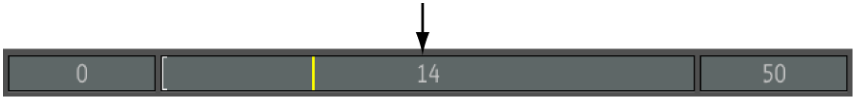
To set the repeat mode:

Click the Repeat Mode on the left of the Player controls, and select a repeat mode from the menu:

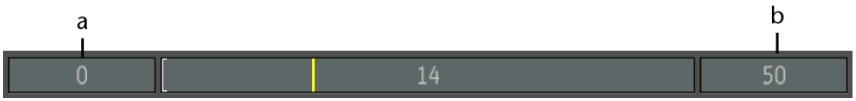
Select:	To set the repeat mode to:
	No repeat. Playback ends when the Player reaches the last frame of the composition.
	Loop continuously.
	Ping-pong. Plays the composition forward to the last frame then backward to the first frame, in a continuous cycle.
	No repeat, to restrict playback to the region between the mark in and mark out frames.
	Loop continuously, restrict playback to the region between the mark in and mark out frames.
	Ping-pong, restrict playback to the region between the mark in and mark out frames.

To scrub through a composition:

Click and drag in the Current Frame field.



To adjust the start or end frames of the composition:



a) Start Frame field b) End Frame field

Click and drag in the Start Frame field to adjust the start frame for the composition, and/or in the End Frame field to adjust the end frame for the composition. The Start and End fields in the Composition tab of the Tool UI update to reflect the changes you make here.

To set a cue mark:

Advance to the frame for which you want to set a cue mark, then click the Set Cue Mark button.

The cue mark appears in the Current Frame field.

To move to an existing cue mark:

Click a cue mark button on the left of the Player controls to move to an existing cue mark.

Click:	To:
	Move to the previous cue mark.
	Move to the next cue mark.

To delete a cue mark:

In the Tool UI, select the Cue Mark tab, locate the cue mark you want to delete, then right-click and select Delete.

To set a keyframe manually:

1 Mark the attributes for which you want to set a keyframe. See Marking Attributes for Keyframing on page 674.

2 Advance to the frame at which you want to set a keyframe.

3 Click the Set Keyframe button.

To move an existing keyframe:

Click a keyframe button on the left of the Player controls to move to an existing keyframe.

Click:	To:
	Move to the previous keyframe of the currently selected tool node.
	Move to the next keyframe of the currently selected tool node.

To turn Autokey on or off:

Click the Autokey button. See Setting Keyframes Automatically on page 680.

For more information on the player, please see The Player on page 189.

Zooming and Panning

You can zoom or pan in the Schematic view and in any of the three player views: Player View, the Mini-Player, and the Fullscreen Player.

To zoom do one of the following:

- In the view or player, press the Up Arrow to zoom in, or the Down Arrow to zoom out.
- Press **`Ctrl + Spacebar`** (on Windows, Linux) or **`Cmd + Spacebar`** (on Mac OS) and drag to the right to zoom in or left to zoom out.
- Press **`Home`** to reset the zoom factor to 1. Pressing **`Home`** again will set zoom to previous zoom factor. This is not available to all viewers.

To pan:

- In the view, press **Spacebar** and drag.

To navigate views:

- Press **Ctrl + Tab** to move to the next view. Press **Ctrl + Shift + Tab** to move to the previous view.

Things to remember:

- When you zoom out in a Player view, the resolution is automatically adjusted to the optimal one for that zoom level. This reduces the load on the computer.

Navigating and Browsing

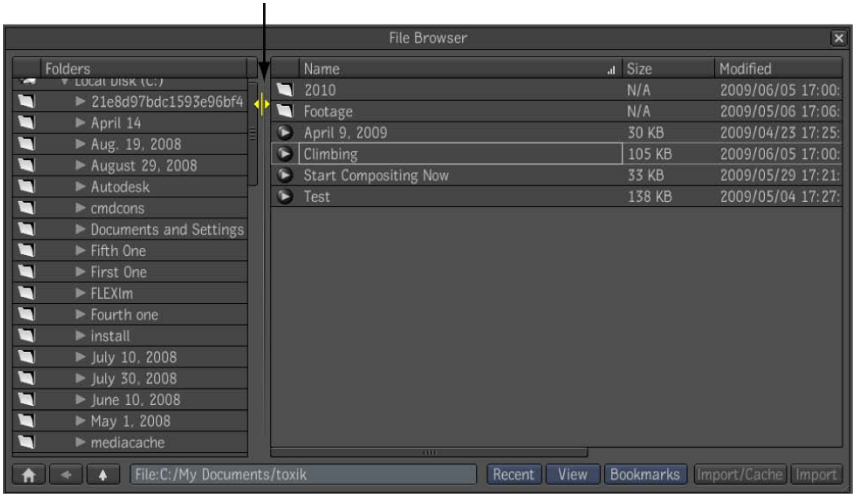
The Composite File browser enables you to locate, organize, and manage compositions, projects, etc.

The File browser is a central tool in the overall Composite workflow. It has the following general uses:

- Identifies Composite compositions or external media, such as files on a hard disk.
- Helps you locate, organize, and manage compositions already in a project.
- Provides access to compositions for Composite tools and for Composite utilities.

Customizing the File Browser

You can resize any component panel in the Composite browser by dragging its side. For example, you can change the division between the Folders and Contents area in the File browser by dragging the border between the two areas.



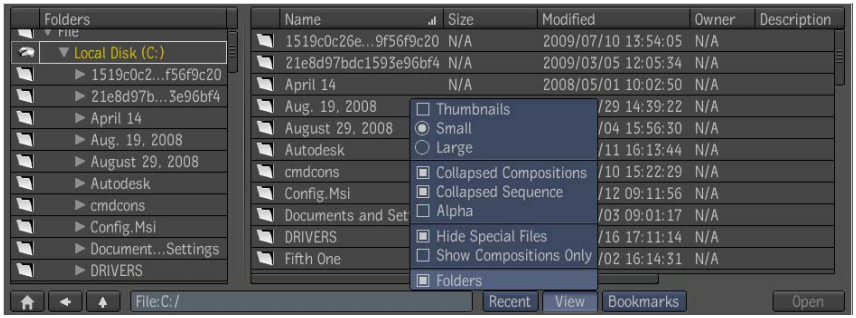
The File Browser

The File Browser provides a view of the folders that your installation references. It also allows you to create compositions by importing media.

The Folders area in the File browser shows the hierarchy of available folders in your project, as well as external drives and resources, in an expandable tree-like structure. By default, the Folders area appears on the left side of the browser, but you can close it to increase the size of the Contents area.

To show or hide the folders area:

- Click the View button and select Folder.



Browser Settings and Controls

The File Browser contains the following main areas and buttons:

Item	Description
Folders list	Displays the file system on your machine, including all mapped drives, as well as Wiretap. Using this area, you can navigate to the media you want to import.
Folder Contents area	Displays the contents of the folder currently selected in the Folders list.
Navigation buttons	Use these buttons to navigate through folders and files.

These are the navigation buttons:

NOTE You can also navigate in the browser using Hotkeys. See Browsers on page 823.

Use:	To:
Home:	Return to the top level of the project folder.
Back:	Move back through the folder list in the File Browser.
Up:	Move to the level of the parent folder.
Path Field:	You can enter paths in this field. Paths can be separated by either forward or backward slashes.
Recent:	View a list of recently viewed folders. Place the cursor over a visited folder and click its name to return to that folder.
View:	Change the way a browser displays information. See The View Button on page 43.

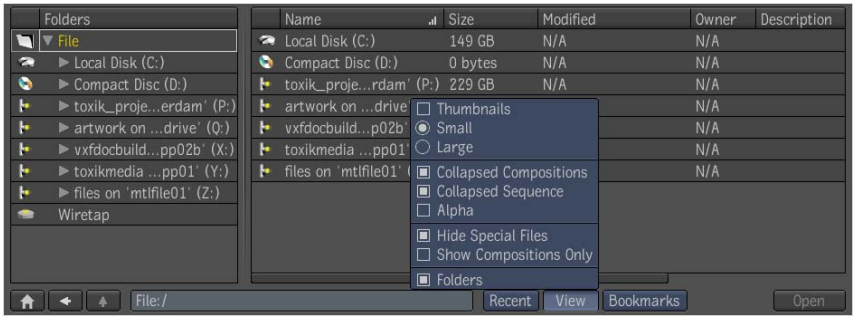
Use:	To:
Bookmarks:	Navigate to a directory. See The Bookmarks Button on page 44.
Open:	Opens the selected files and adds them to the current Composite composition. If the File Browser selection is a Composite composition, this button will be labeled Open, and will open the selected composition.

The View Button

A view is how the browser displays information in the Contents area. You can view information as a list or as thumbnails in the Contents area. The List view displays more information in the Contents area than other views and allows you to easily sort information by clicking one of the column headings. A thumbnail is an icon for a folder, source, or composition. There are thumbnails for folders, fonts, audio, and so on. The thumbnails for sources or compositions show a frame.

You can change thumbnails to show a source or composition's matte, if it contains one.

The folder hierarchy shows folder relationships: child folders cascade from the parent folder(s). Parent folders are displayed with triangle icons that cascade child folders below and to the right when clicked. The area below the folder shows either thumbnails or a list of the folder's contents depending on the mode you're in (list or thumbnail).



To view thumbnails or a list in a browser:

Click the View button and select Thumbnails to view thumbnails. Deselect Thumbnails to see a list.

To change the thumbnail size in the Contents area:

Click the View button and select Small or Large.

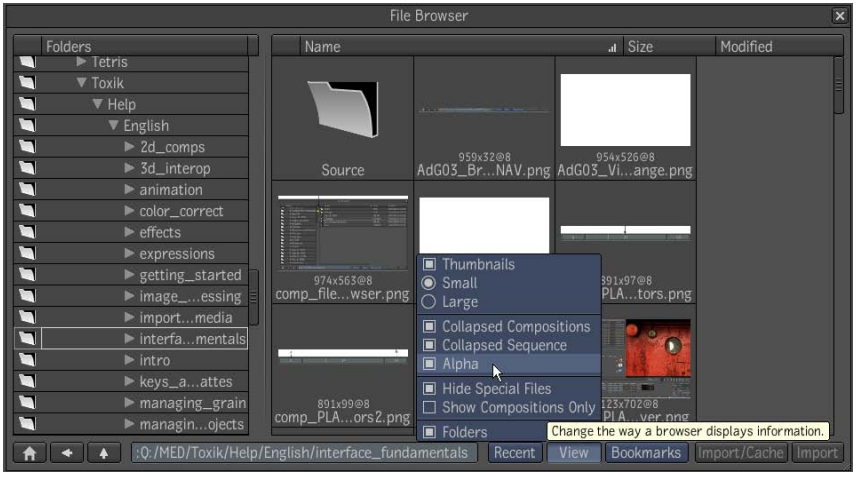
The sources or compositions in the Contents area are resized as follows:

Thumbnails	Size (in pixels)
Small	160 x 120
Large	249 x 180

To view mattes:

Click the View button and select Alpha.

Thumbnails for sources or compositions with mattes display their black and white matte. Thumbnails for sources or compositions without mattes remain in color.

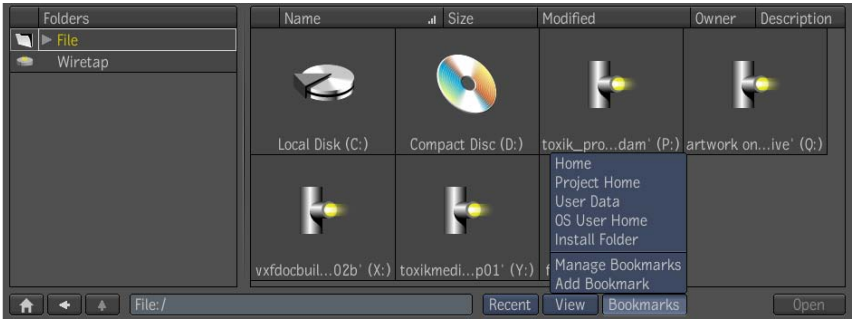


The Bookmarks Button

Bookmarks allow you to quickly navigate to a directory. Click the button and then click a name in the pop-up list. Clicking a bookmark's name positions the browser in the target directory just as if the path had been entered manually.

There are five predefined bookmarks, which cannot be removed or renamed. They are the following:

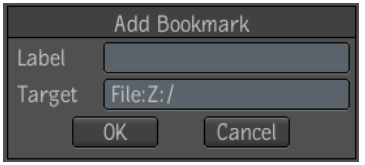
Name	Function	Value	Comment
Home	Shorthand to browser's home	Not editable	Depends on the "Set as Home" setting
Project Home	Bookmarks location of project root	Not editable	Depends on selected project
User Data	Bookmarks location of user directory	Not editable	Depends on user logged on
OS User Home	Home directory of user logged on	No editable	Depends on user logged on
Install Folder	Composite installation folder	Not editable	



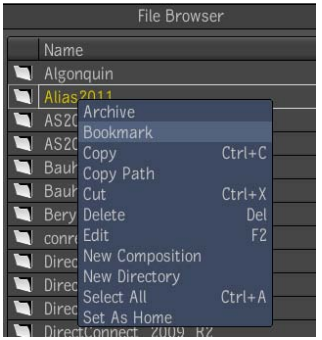
In addition to the predefined bookmarks, personal bookmarks can also be created. There are two methods to create personal bookmarks:

To create a personal bookmark:

From the bookmarks list, select Add Bookmark or press **`Ctrl + D`** (on Windows and Linux) or **`Cmd + D`** (on Mac OS). The Add Bookmark dialog appears.



Or right-click on a directory in the File browser and select “Bookmark,” to bookmark that directory.



All personal bookmarks are automatically assigned the name of the bookmarked directory, or the name of the bookmarked directory plus a "#" extension, where "#" is the first number, beginning with 1, which makes the name unique. Although multiple bookmarks can share the same target directory, duplicate naming of bookmarks is not permitted. The "#" extension ensures that no two bookmarks have exactly the same name. For example, if you name a bookmark "New Bookmark," and then try to give another bookmark the same name, it will be named "New Bookmark (1)."

Managing Bookmarks

Bookmarks are managed through the Bookmark Manager.

It is used for the following tasks:

- Adding new bookmarks

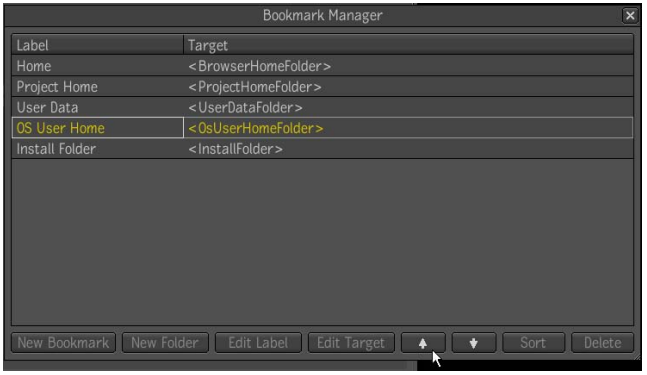
- Creating bookmark folders in order to categorize bookmarks
- Editing personal bookmark labels
- Editing personal bookmarks or the Media bookmark's target
- Customizing the bookmarks and bookmark folders order
- Moving personal bookmarks from one folder to another
- Removing bookmark folders and personal bookmarks

To manage bookmarks:

1. Use the Manage Bookmarks option. It is accessible through the bookmarks pop-up list.

2. Clicking Manage Bookmarks opens the Bookmark Manager.

3. Use the up or down arrows to rearrange the order of the bookmarks in the Bookmark Manager. Click the Sort button to arrange the bookmarks in alphabetical order.



Floating File Browser

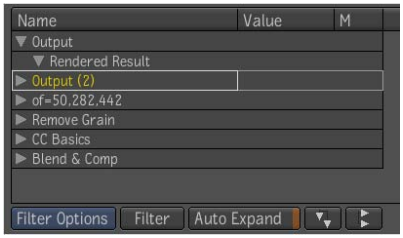
You can also display the File Browser as a floating browser, so that you can use it at any point during a session to access project folders.

To display the floating File Browser, do one of the following:

- Press **`Ctrl + O`** (for Windows and Linux) **`Cmd + O`** (on Mac OS) or to open the File Browser with the "Show Compositions Only" view option set.
- Press **`Ctrl + I`** (for Windows and Linux) or**`Cmd + I`** (for Mac OS) to open the File Browser without the "Show Compositions Only" view option, i.e. view all files.

The Composition Browser

The Composition Browser displays all the nodes for the current composition. Using this browser, you can view parameters and set keyframes for animation. It's also an easy way to quickly view parameter names when referencing them in expressions.



The Composition Browser has three columns. The Name column contains the name of the node or attribute. The Value column contains the value associated with an attribute. The M (marked) column indicates whether an attribute is marked for keyframing.

The Composition Browser is also available in the Animation tab of the tool UI.

Composition Browser Functionality

There are five buttons at the bottom of the Composition Browser: the Filters Option button, the Filter button, The Auto Expand button, The Expand Column button, and the Collapse Column button.

The Filters Options Button

The Filters Options lets you select one or more filters. The filters are divided into two groups: Tools and Channels.

Multiple filters are additive (except for the All filter option, which unselects other filters and shows all nodes). For example, clicking Mute and Context

Point would display muted nodes as well as nodes that have a context point set on them. Therefor adding tool filters adds tool nodes to your display.

Click:	To:
Tools	
Selected	**Display tools (in the result) that are currently selected.**
Similar	**Filter tools that are similar to the selected tools.**
Muted	**Display tools (in the result) that are muted.**
Context Point	**Filter tools that have context points on them.**
All	**Display all nodes in the composition.**
Channels	
Animated	**Display nodes (in the result) that contain channels that are animated.**
Expression	**Filter inputs that are connected to expressions.**
Marked	Display nodes (in the result) that contain channels marked for keyframing.

The Filter Button

Click this button to apply the filters you have selected.

The Auto Expand button

Click this button to expand the parent folders of the selection and center on the first item, when the global selection changes.

The Expand Columns button

Click this button to expand all the columns in the browser.

The Collapse Columns button

Click this button to collapse all the columns in the browser.

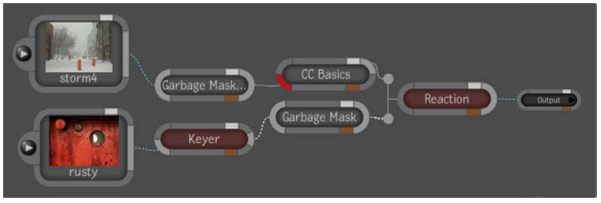
Working with Schematic

The Schematic is the view in which you build the composition – a set of connected nodes – for a composition. When you create a new composition, the only node in the tree is the output node; by default, every composition has only one primary output node.

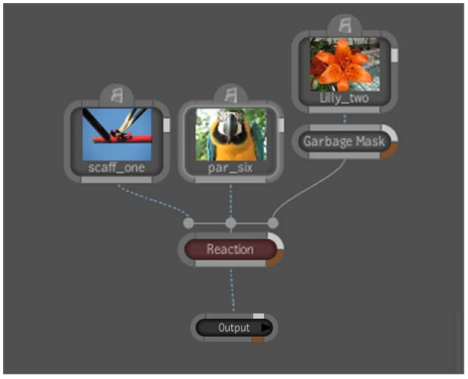
NOTE Unless otherwise specified, the term “node” and “tool node” are interchangeable throughout this guide.

When you create a composition by importing media, the composition contains one input node (which points to the media you imported) and one output node. You then build the dependency graph by adding tool nodes. For more detailed information on the different types of nodes, see Node Overview on page 52.

You can set the direction in which processing proceeds in the User Preferences Creative tab. For 2D workflows, you can build the dependency graph left-to-right or top-down direction. For 3D workflfows, you can build the dependency graph in a left-to-right or bottom-top direction -- see Setting User Preferences on page 168.



A dependency graph with left-to-right flow.



A dependency graph with a top-down flow.

NOTE Nodes do not necessarily have to be connected. For example, you can create branches that you connect or disconnect to experiment with different scenarios.

When working in the Schematic view, you can:

- Show or hide the Schematic Navigator.
- Clean up the view (clean up all or a portion of the composition, automatically zoom to fit the dependency graph in the view).
- Change the name of a node.
- Work with connections (connect one node to another, insert a node between two nodes, disconnect one node from another, reposition a node within the composition, change the appearance of connections, have connections highlight when you pass the cursor over them).
- Work with groups and super tools (create a group, edit a group or enter a Super tool, exit a group or a Super tool, ungroup a group, add inputs and outputs to group nodes).
- Work with branches (collapse or expand a branch).
- Mute or unmute a node.
- Turn intermediate results on or off for a node.
- Set or clear a context point on a node. See Setting Context Points on page 191.
- • Reset the node to its default values.
- Cut, copy, paste, or delete nodes. Navigate nodes.

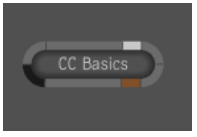
NOTE When using super tools or working inside a grouped tool, the name of the tool or group is displayed a the bottom of the Schematic, so you always know where you are in relation to the dependency graph.

Node Overview

There are several different types of node features, as well as different types of node connections.

Node Types

There are a number of different types of nodes, each represented by a different icon.

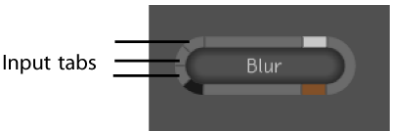
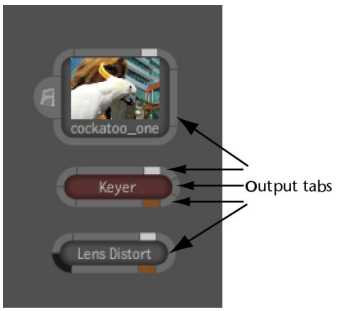
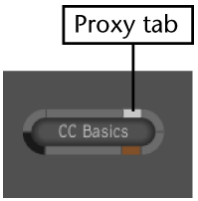
Node type	Description
	An input node for a composition created during a normal media import. It is identified by a small sheet icon on the left of the proxy. When you select an imported media input node, the tool UI displays the Import Image tab. The input node displays a proxy of the media it references.
	An input node for footage, created using “Import as Footage”. It is identified by a small film icon on the left of the proxy. The Link Image tab opens when a footage input node is selected in the Schematic view.
	An input node for a linked composition. It is identified by a small composition icon on the left of the proxy. When you select a linked composition input node, the tool UI displays the Link Image tab. The input node displays a proxy of the rendered output it references.
	A normal tool node.
	A super tool node. Each super tool can have its own icon. The Reaction super tool icon is shown here. A super tool node behaves as a group node in that you can enter the node to work with the tools it contains.

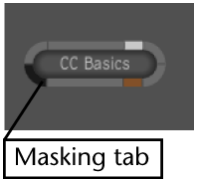
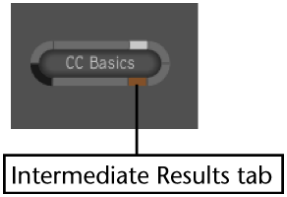
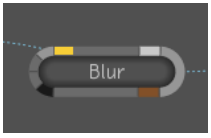
Node type	Description
	A group node that you create by selecting two or more nodes and grouping them together. You can right-click a group and select Edit Group to work only with the nodes in that group.
	A visual group node is similar to a normal group node, however you are able to see its contents without entering it. You can also change its background color.
	The primary output node for a composition. A dependency graph always has exactly one primary output node. You cannot delete this node, but you can add secondary output nodes to the composition. See Multiple Output Nodes on page 56. The output node represents the result of the composition. When you select this node, the tool UI displays the Output tab. The output node does not display a proxy of the result.
	A secondary output node for a composition. Note that secondary output nodes vary slightly in appearance from the primary input node, it is a lighter gray.

Node Anatomy

Each node has a set of tabs around its outer edge. The name of each node appears either underneath it or inside it.

Tab:	Description:
Input	You connect an input to this tab. If a node accepts multiple inputs, the tab is divided into the number of inputs the node accepts. ■ A node that accepts a single input:

Tab:	Description:
	 ■ A node that accepts nine inputs: 
Output	Contains the output of the node. You create connections between nodes by connecting the output tab of one node to the input tab of another. The Output node contains the result of the dependency graph. Note that you can connect an output to more than one input. 
Proxy	Click this tab, or the **T** hotkey, to show or hide the proxy for the node. 

Tab:	Description:
Masking	There is an input for masking. See Pixel Masking on page 560. 
Intermediate Result	Click this tab to enable intermediate results for the node. See Caching on page 226. 
Marked for keyframing icon	A yellow icon appears nest to the image input tab when the tool has been marked for keyframing. See Marking Attributes for Keyframing on page 674. 

Output Nodes

There are several characteristics of Output nodes that are useful for you to know.

Multiple Output Nodes

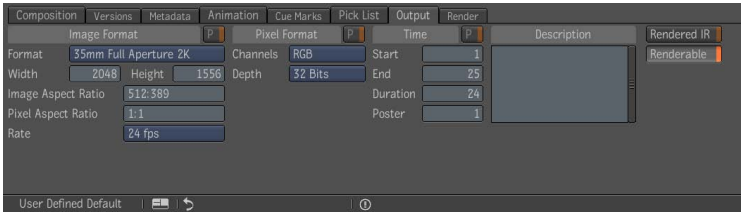
A composition can support multiple output nodes which provide simultaneous renders from different points in the dependency graph. This characteristic of

the composition lets you link to a composition at different points in the dependency graph.

An output node cannot be muted, however, intermediate results can be enabled on output nodes and output nodes can be grouped at any level. There's no restriction on the presence of an output node at the highest group level (the composition level).

Each output node has its own format (resolution, pixel aspect ratio, rate, channels and depth) and it crops its input image according to its format.

An output node has the following editable values. You can reset the values back to their defaults in the Tool Options area (Reset button):



Value:	Description:
Name	Initialized to Output (#).
Description	Of arbitrary length, empty by default.
Image Format	Width, height, pixel aspect ratio and rate. Primary output is initialized from the composition project settings, additional outputs from the primary output.
Pixel Format	Channels and depth. Primary output is initialized from the composition project settings, additional outputs from the primary output.
Start	Start frame index. Primary output is initialized from the composition project settings, additional outputs from the primary output.
End	End frame index, exclusive. Primary output is initialized from the composition project settings, additional outputs from the primary output.
Duration	End – Start.
Poster	Frame used for Proxies, defaults to start value.

Value:	Description:
As Primary Output Image Format	Whether the width, height, pixel ratio and rate values follow the primary output values. Always off on primary output. On by default on additional outputs.
As Primary Output Pixel Format	Whether the channels and depth values follow the primary output values. Always off on primary output. On by default on additional outputs.
As Primary Output Time	Whether the start, end and poster values follow the primary output values. Always off on primary output. On by default on additional outputs.
Rendered on/off	Whether the output is rendered upon a render command, on by default.
Render mode	Render mode used upon a render command. Set by the composition render mode project setting by default.

Primary Output

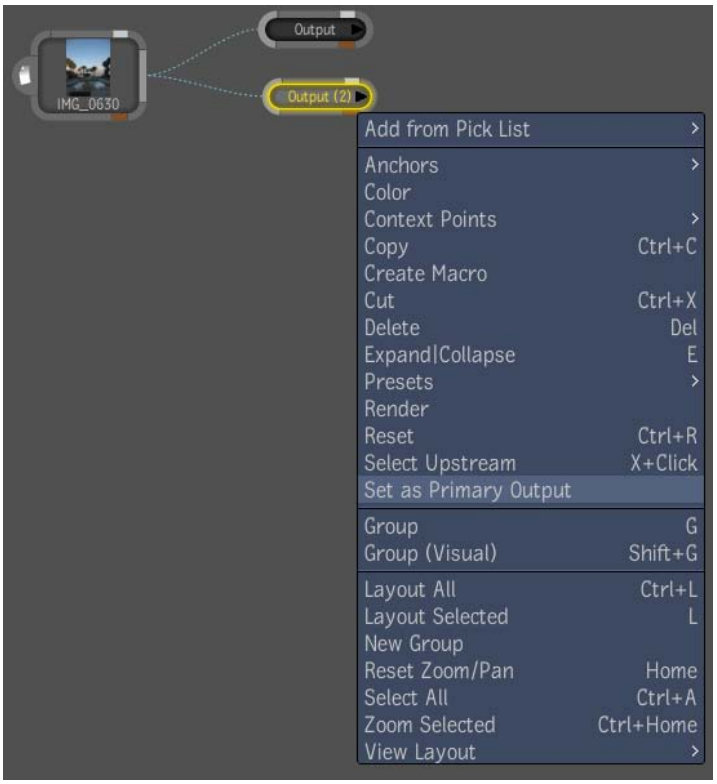
One of the outputs is tagged as being the primary output. Aside from identifying the principal output of a composition, the primary output is most useful in the following workflows:

- The composition start/end (in the timeline) and rate are dictated by the primary output start/end and rate respectively.
- Composition format displayed in the File Browser is driven by the format of the primary output.
- Composition proxies shown in the File Browser are generated from the primary output.
- Player in Composition display mode shows the image results of the primary output node.
- Comparison in the player against the composition refers to the primary output.
- When linking to a composition, even though all outputs of the linked composition are represented on the link node, only the primary output socket is automatically connected to the destination graph.

- When inserting a composition into another one, the input node of the primary output from the source graph is used as the connection point when the composition is dropped on an input socket in the destination graph.
- When dropping tools in the Schematic view, the south gate option Add Before Primary Output connects the new nodes before the primary output node.
- Additional outputs are initialized (or reset) with the values of the primary output. Other outputs can also follow various sets of values from the primary output (image format, pixel format, time).

The first output node that is automatically created with a new composition is the primary output by default. However, in a composition with multiple outputs, any output can be set as the primary output at any time. This can be achieved through a drop-down menu listing the output node names in the Composition tab.

You can also use the Set As Primary right-click option on an output node in the Schematic view.

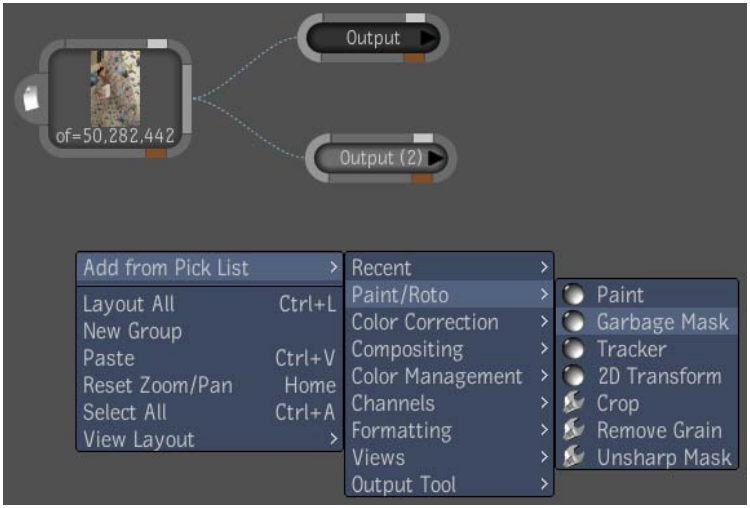


Adding Output Nodes

Creating a new composition automatically adds an output node to it. However, you can add more output nodes anywhere in the composition.

To add an output node to a composition, do one of the following:

- Drag the output tool from the Utilities folder in the Tools tab and drop it into the Schematic view.
- Select an output node in the Schematic view, right-click and select Copy, or press **`Ctrl + C`** (for Windows and Linux) or **`Cmd + C`** (for Mac OS). Then right-click again and select Paste or press **`Ctrl + V`** (for Windows and Linux) or **`Cmd + V`** (for Mac OS).
- If you have added the Output tool to a Pick List, right-click in the Schematic view and select the Output tool from the Pick List.



Deleting Output Nodes

The primary output node cannot be deleted, which ensures that a composition has at least one output. Graphical delete options on the primary output are absent and any attempt to delete it in another way (hotkey or scripting) results in an error message.

To delete a secondary output node, do one of the following:

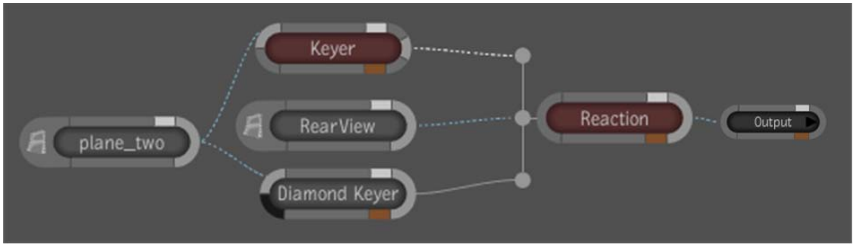
- In the Schematic view, highlight the secondary output(s) you want to delete, right-click and select Delete or press the Delete key.
- Highlight the secondary output node(s) you want to delete in the Composition Browser, right-click and select Delete.

Node Connections

The connection lines in a dependency graph visually indicate by color and style what you are outputting: RGBA, RGB or A. You can also choose how you want the lines to appear: curved, angled or straight.

Color	Line Style	Output
Gray	Solid line	RGBA
Light-blue	Dashed line	RGB

Color	Line Style	Output
White	Dashed line	A



To change the appearance of a connection line:

1. Middle-click or press the tilde key (~) key to display the drop gate, and swipe south to the Schematic Options.
2. Under Display, select an option form Links for the connection lines.

NOTE You can also set the links style in your user preferences. See Setting User Preferences on page 168.

Node Options

There are a number of ways to manipulate nodes in the Schematic view.

To clean up all or a portion of the dependently graph, do one of the following:

1. To clean up the complete tree, right-click and select Layout All, or press **Ctrl + L** (for Windows and Linux) or **Cmd + L** (for Mac OS).
2. To clean up a portion of the tree, select the nodes to clean up, then right-click, and select Layout Selected or press **L**.

To reset the zoom and pan to fit the dependency graph:

➤ Right-click and select Reset Zoom/Pan or press **Home**.

To change the name of a node:

1. Select the node to display its tabs in the tool UI.

2 In the Tool Options at the right, click in the Name field and edit the name.

3 Press **Enter** to accept your edits.

The node in the Schematic view updates to reflect the new name.

To display the details of a node without opening a node:

1 Press the **D** key and pass the cursor over a node. **Shift + D** will turn on all the node details without having to pass over the node. Press **Shift + D** again to turn off the node details.

2 A tooltip displays the details of the selected node. If the node was renamed, the node's original name appears in the tooltip.

To display the thumbnail of a node, select one or more nodes and do one of the following:

1 Right-click the node and select Thumbnail.

2 Click **T**.

3 Click the tab on the node.

To connect two nodes, do one of the following:

1 Click the output of a node and drag to the input of another node (or click and drag from the input area of one node to the output area of another). A gray connection line appears as you drag. Release to create the connection.

2 Press **Shift** and drag one of the nodes to the other, so that the output area of one node brushes, or "kisses", the input area of the other. A connection line appears when the two nodes kiss. Release to establish the connection. If you want to cancel the operation, release **Shift** as you continue to drag.

NOTE There are hotspots located along each edge of the viewer to be used to auto-pan the viewer when holding the cursor down over these areas for a predetermined time. This will be used when dragging or connecting nodes. The speed at which the panning is done can be controlled by using the **Ctrl** hotkey to speed up panning or the **Alt** hotkey to slow down panning.

To insert a node between two other nodes:

➤ Press **Alt** and drag the node you want to insert onto the connection line between the two nodes, then release. If you want to cancel the operation, release **Alt** as you continue to drag.

To disconnect nodes, do one of the following:

1. Press **Ctrl** and drag the cursor (scissors icon) through the connection.
2. Right-click the connection line between the nodes and select Disconnect.

To quickly disconnect and reconnect nodes:

➤ Hold down the **Ctrl** key, click one end of a connection link to break the link. Drop the link on a different node.

To reposition a node within a dependency graph, do one of the following:

1. Disconnect the node and then connect it in a new position.
2. Press **Alt** and drag the node to a new position in the tree, then release. The connection lines adjust to disconnect the node from its previous neighbors and connect it in the new position.

To replace a node's tool with another tool:

1. Display the Tools, Views & Pick List tab by middle-clicking or pressing the tilde (~) key. Drag and drop a tool onto the selected node. A drop gate appears.
2. Swipe through the Replace option. The selected node is replaced with a new tool.

To create a group consisting of multiple nodes:

Select the nodes you want to group, right-click one of the nodes and select Group or Group (Visual). You can also press **G** for Group or **Shift + G** for Group (Visual).

NOTE If more than one node is selected, they will be grouped. If there is only one node selected and it is not a group node, it is grouped, otherwise if it is a group node, it is ungrouped.

When you create a visual group node, all the nodes in this group display semi-transparent backgrounds that allow the nodes to adopt the color of the visual group itself. Nodes which are not part of the visual group that are dragged over the group will stand out because they will be a different color from the nodes in the visual group.

Visual group nodes are created with a default color. However, by right-clicking on the title bar of the visual group, you can select the Color option, which will display a color picker that allows you to choose a new color.

A new node-independent tab is created in the Tool UI for the group node. By default, it will be named "Group (#)," for example, "Group(1)." You can rename the group tab by entering a new name in the Name field in the Tool Options.

To edit a group or enter a super tool:

1. Right-click the group node (to edit a group) or the super tool (to enter the super tool) and select Edit Group, or double-click the node.
2. The Schematic view updates to display the nodes in the group or the nodes of the super tool.

To exit a group or a super tool:

➤ Right-click in the Group Schematic view and select Exit Group, or double-click the background.

To resize or move a visual group node:

➤ The size of the visual group node is determined by the location of the contained nodes. The size of the visual group node will expand or contract as the contained nodes are moved around. To move a visual group node, drag the title bar of the group node.

To lock a visual group or lock the nodes in a visual group:

1. Right-click on the title bar of a visual group node and select Lock, which locks the group node, as well as all the nodes inside the group. This means that all the nodes inside the group, as well as the group itself, are locked in place.

2 To lock just the nodes inside a group, while still allowing the group itself to be moved, right-click on the title bar of the visual group node and select Lock Tools.

To open or close a visual group node:

1 To open a group node, right-click on the node and select Open or press **Shift + O**. Opening a group node displays the contents of the group node while applying the visual group attribute.

2 To close a group node, right-click on the title bar of the group node and select Close or press **Shift + C**. Closing a group node removes the visual attributes of the node and collapses it back to its regular appearance.

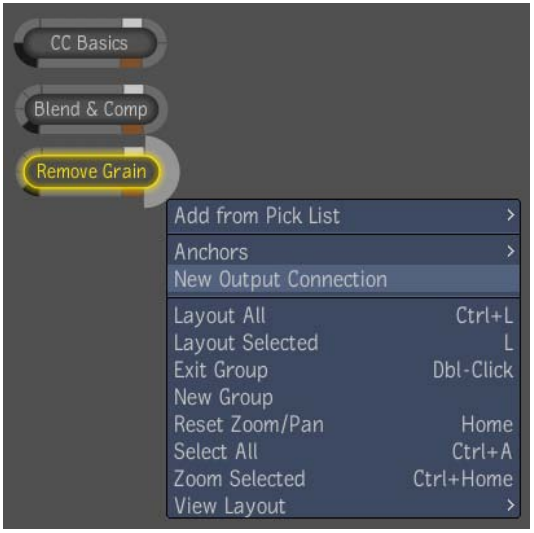
To ungroup a group:

➤ Right-click the group node or the title bar of the visual group node and select Ungroup, or press **G**.

To add input or output connections to a group node:

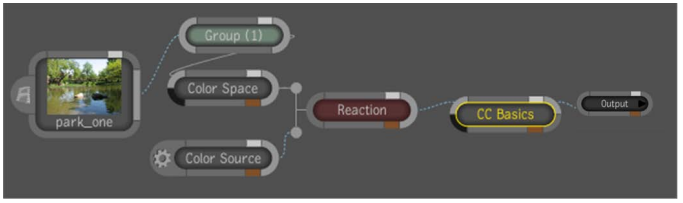
1 Enter the group node by double-clicking the node, then drag the link from an input or output beyond the top border of the schematic. Control will be transferred to the group level one up in the hierarchy. At this point, releasing beyond the border will create a connection node. Moving within the border will allow connection to another node. If this group level is not the top most, moving within the border and back up again will transfer control to the next group up in the hierarchy. Moving down in the group hierarchy is accomplished by dragging beyond the bottom border of the Schematic view. All dragging must be done in conjunction with the Shift hotkey.

2 Enter the group node by double-clicking the node, then right-click and select New Output Connection. Note that the cursor must be over an input or output.



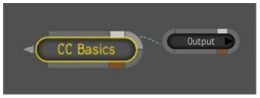
To collapse a branch of the dependency graph:

1 Locate the node at which you want to collapse the branch.



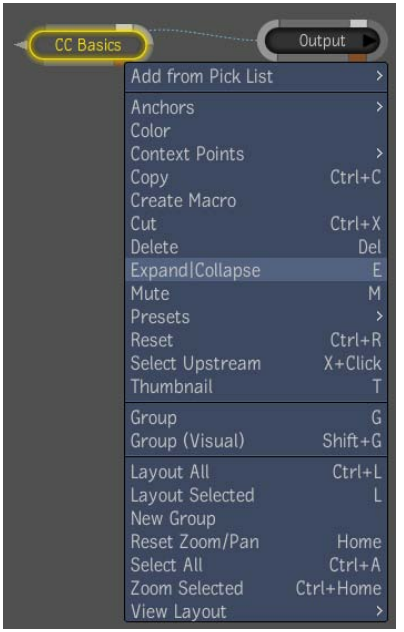
2 Right-click the node and select Expand or Collapse or press **E**.

3 The branch leading into that node collapses. The arrow on the left of the node indicates a collapsed branch.



To expand a collapsed node:

➤ Right-click the node containing the collapsed branch and select Expand or Collapse, or press **E**.



To mute or unmute a node:

1. Select the node.
2. Right-click a node and select Mute or Unmute, or press **M**.

NOTE You can also mute/unmute a node by selecting it and clicking the Mute button in the Tool Options area. This button is located to the left of the Reset button.

To select an entire branch of the dependency graph, do one of the following:

1. Press X and click a node.
2. Right-click and choose Select Upstream.

All nodes upstream are selected.

Hint: You can select multiple branches without clearing the selection.

To turn intermediate results on or off for a node:

➤ Click the orange tab in the lower-right of the node. The orange tab brightens or darkens to indicate results are, respectively, on or off for the node. See Caching on page 226.

NOTE You can also turn intermediate results on/off for anode by selecting the node and clicking the IR button in the Tool Options area.

To set a context point on a node:

1. Press the number of the context point you want to set and click the node on which you want the context point, or right-click the node and select Context # (Set). For example, to set context point 3, press 3 and click the node.
2. The number of the context point, preceded by the letter C (for example C3 for context point 3), appears to the left of the node name and a broken green line appears around the center of the node.
3.

NOTE If you set more than one context point on a node, the numbers of all context points set on the node appear after the C. For example, C134 indicates you set context points 1, 3 and 4 on the node.

To clear a context point on a node:

1. Press the number of the context point you want to clear and click the Schematic background or right-click the node and select Context # (Remove).
2. The context point label (for example C4 for context point 4) is deleted along with the broken green line around the center of the node.

To reset a tool node:

➤ Right-click the tool node you want to reset and select Reset, or press **`Ctrl + R`** (for Windows and Linux) or **`Cmd + Shift + R`** (for Mac OS).

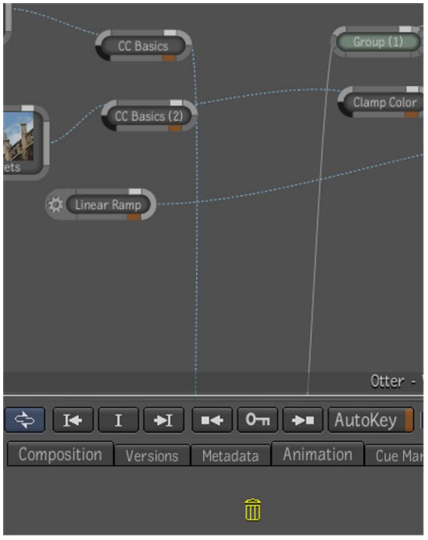
NOTE You can also reset a node by selecting it and clicking the Reset button in the Tool options area.

To cut, copy or paste one or more nodes:

1. Select the node(s) you want to cut or copy.
2. Then, do one of two things. To cut the node(s), right-click one of the selected nodes and select Cut, or press **Ctrl + X** (for Windows and Linux) or **Cmd + X** (for Mac OS). To copy the node(s), right-click one of the selected nodes and select Copy, or press **Ctrl + C** (for Windows and Linux) or **Cmd + C** (for Mac OS).
3. To paste the cut or copied node(s), right-click outside all nodes and select Paste, or press **Ctrl + V** (for Windows and Linux) or **Cmd + V** (for Mac OS).

To delete one or more nodes:

1. Select the node or nodes you want to delete.
2. Then, do one of the following. Either right-click one of the nodes and select Delete, or press **Delete**. Or drag a reasonable distance outside the bottom edge of the Schematic view and release. A garbage icon will appear to indicate the point at which it is possible to release the node.



NOTE You can also delete nodes by selecting then and clicking the Delete button in the Tool Options area of the tool.

To select all nodes, do one of the following:

1. Right-click the Schematic view and choose Select All.
2. Press **Ctrl + A** (for Windows and Linux) or **Cmd + A** (for Mac OS).

All nodes and connection lines are selected in the Schematic view.

Selecting Nodes Upstream and Downstream

Drill-Down is a way to change the currently selected node without depending on a viewer, such as the Schematic view or the Composition browser. It changes the current node by navigating upstream (towards inputs) or downstream (towards outputs) using hotkeys. It only allows navigating through connected nodes.

To navigate through a series of nodes, do one of the following:

1. Press **Alt + Shift + Left arrow** to select the upstream node connected to the current node's primary input.
2. Press **Alt + Shift + Right Arrow** to select the first downstream node connected to the first output.
3. Press **Alt + Shift Down arrow** to select the next node for the last direction taken. For example, after pressing **Alt + Shift + Left Arrow** to go towards the primary input, pressing **Alt + Shift + Down Arrow** will select the node on the second input. It works similarly for outputs: pressing **Alt + Shift + Down Arrow** will visit the next node connected the output, or go to the next output, whichever applies.
4. Repeatedly press **Alt + Shift + Down arrow** to cycle through the candidates in their order of presentation.
5. Press **Alt + Shift + Up arrow** to select the previous candidate for the last direction taken, following the same logic as for **Alt + Shift + Down arrow**.

Cycling Through Inputs and Outputs

For easy display of a node's multiple inputs and outputs in the Player, you can cycle the inputs or outputs shown by the Player using hotkeys. When in tool input mode in the Player, pressing the 5 hotkey a second time cycles to the next image input. Once the last input is reached, pressing the 5 hotkey

again cycles back to the first image input. When in tool output mode in the Player, pressing the 6 hotkey a second time cycles to the next image output. once the last output is reached, pressing the 6 hotkey again cycles back to the first image output.

For added flexibility, you can assign a context point to a tool node. For example, while viewing the output of a tool in one Player, you may also want to view the result of a composition, matte, or other Keyer super tool output in another Player. To do this, add a context point (C1, C2, C3 or C4) to the Keyer super tool and assign a Player view to that context point. Then, use the context point's hotkey number (1, 2, 3 or 4) to cycle through the Keyer's outputs. See Setting Context Points on page 191.

Schematic Options

You can set a number of display and playback option for the Schematic view.

To access the Schematic options:

1. With the cursor over the Schematic view, middle-click or press the tilde (~) key and go through the south gate.
2. The settings for that Schematic view are displayed.

NOTE If more than one Schematic view is open, your cursor must be placed over the Schematic view on which you want to set options.

To set the display options:

- Select the display tab.

- Then set the display options.

Select:	To:
Navigator	Show or hide the Schematic Navigator. By default, the Navigator is displayed.

Select:	To:
Highlights	Expand and highlight the tabs of a node when you pass the cursor over them.
Links	Change the line style of connections. Lines can appear as: Curved, Straight, or Angled.

■ Finally, set the placement options.

Select:	To:
Auto Layout	Automatically organize all of the nodes in the dependency graph.
Auto Zoom	Automatically zoom to keep the complete dependency graph visible in the center of the Schematic view.
Locate Node	Automatically locates a specific node in the Schematic when you select in the Composition browser.

NOTE You can make any of the display settings the default settings for the view by selecting them in your user preferences. See Setting User Preferences on page 168.

To hide the Schematic Navigator in the full screen Schematic:

1. Middle-click or press the tilde (~) key to display the drop gate, and swipe north to view the full screen Schematic.
2. Middle-click or press the tilde (~) key to display the drop gate, and swipe south to the Schematic Options.
3. Under Display, select Navigator to hide the Navigator. By default, the Navigator is visible.

To set the playback options:

■ Select the Playback tab.

■ Then set the ViewerTime options:

Set:	To:
Follow Master	Sets the view to global time. Disable to use its own global time.
Offset	Lets you set a frame offset for the time. The frame offset is with respect to the time you set for the view. For example, if you set a frame offset of 5 and selected the Follow Master option, the proxy in the view always displays the frame that is five frames ahead of the frame currently running in global time. If you deselected Follow Master, the proxy in the view always displays the frame that is five frames ahead of the frame running in the local time of the view.
Update on Play	Updates the proxy at each frame when you play the composition. Deselect to update proxy only when you stop playing the composition, and only at the scrub frames when you scrub through the composition.

■ Finally, set the Frame Rate options:

Set:	To:
Play All Frames	Plays all frames in the composition, regardless of whether it maintains the frame rate set for the composition. Deselect to have the view maintain the frame rate for the composition, at the expense of dropping the frames where necessary.
User	Lets you set the frame rate at which you want the Schematic view to play the composition. Type the frame rate in the field to the right of this button, or click and drag the bottom edge of the field to adjust the value.
Stop on Drop	Stops the Player if a tool is dropped into the dependency graph

Things to Remember:

- Each Schematic view has its own set of options except for the flow (left-right, top-bottom, etc.)

Start Compositing Now!

4

Topics in this chapter:

Try First, Read Later

This chapter will allow you to start experimenting with Composite tools right away. You'll find that many of the tools available in Composite are probably familiar to you, and as soon as you import some media, you can start trying it out. Later on, you'll want to review the Projects on page 95 and Getting Familiar with Your Workspace on page 139 chapters to get a deeper understanding of how best to take advantage of Composite's many capabilities.

Start Composite, Create Composition, Import Media and Go

To get up and running fast with Composite, simply start Composite, create a composition, add media and start working. When you import media into

Composite, it becomes a composition that you can open, or use to insert or link to/from another composition. The Tool UI at the bottom of the Composite desktop shows composition information, and updates to show the Tool UI of the tool you are currently working with.

Composite comes with a set of default user interface presets, called task group presets, that let you begin working effectively, immediately. Presets are grouped according to task; you simply choose the desired task group and a preset displays a UI configuration with the views you need to perform a particular task. After you've chosen a preset and created a composition, press the tilde key (~) or middle-click to display the Gate UI. Swipe to the right (east) and choose a tool from the Tools tab.

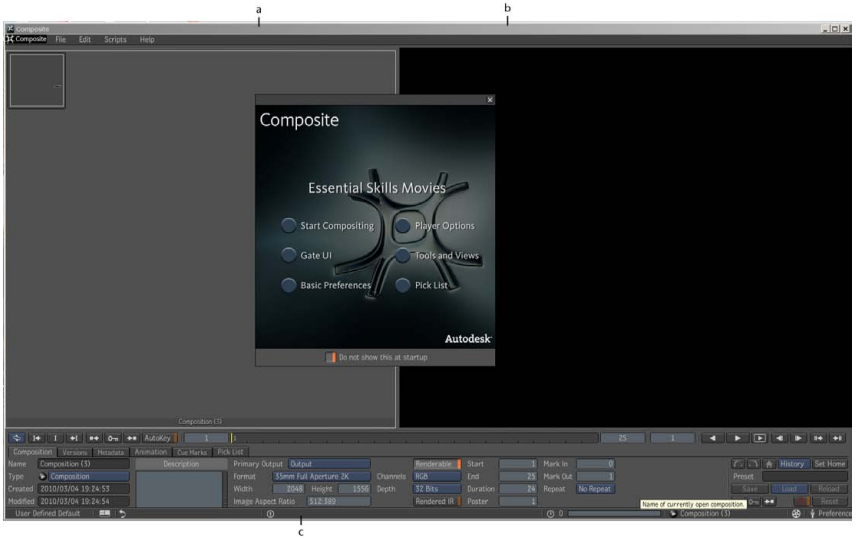
NOTE You can, of course, customize presets or create your own.

To start a new composition in Composite:

1 Start Composite.

NOTE You can choose to watch one of the Essential Skills movies, or just close the dialog box.

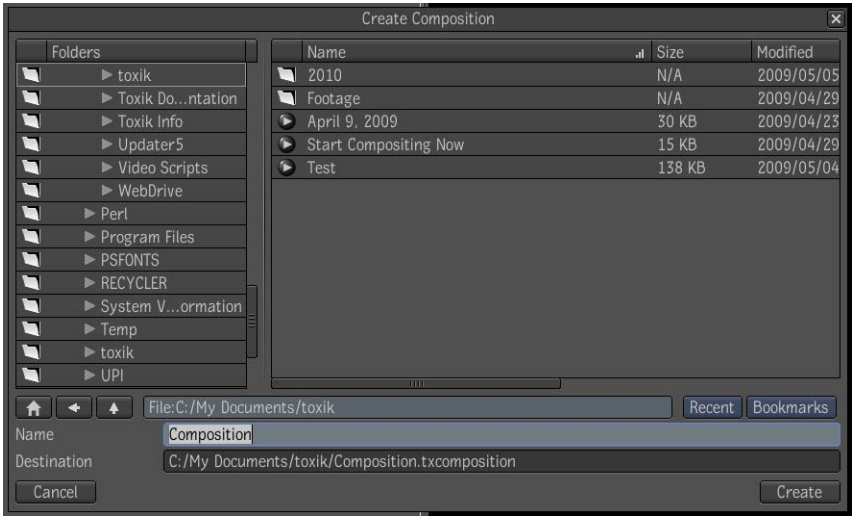
2 The Composite desktop is displayed with the following views: Schematic, Player, and Tool UI/Tool Options.



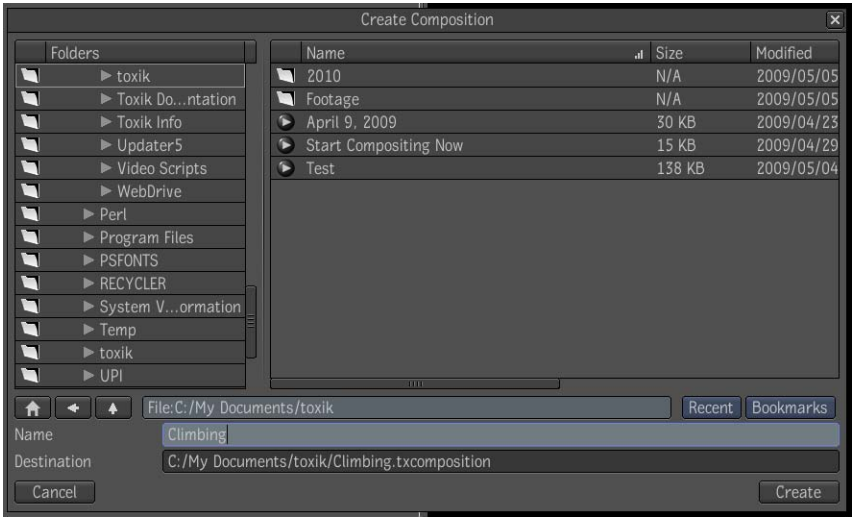
(a) Schematic view (b) Player (c) Tool UI/Tool Options area

3 From the File menu, select “New.”

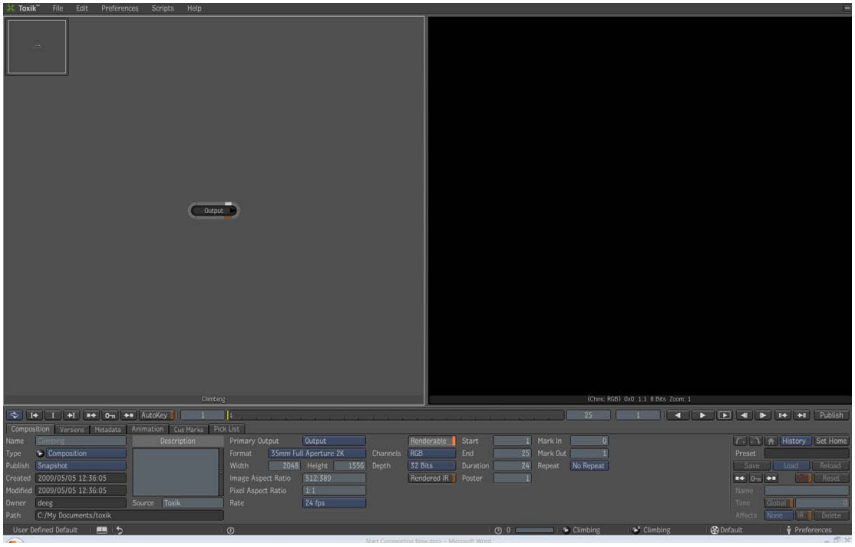
4 The Create Composition dialog appears.



5 In the Name field, enter a name for the composition. You can change the location of the composition in the field above the Name field. Once you have entered the information, click the Create button.



6 A new composition has been created.



Add Media

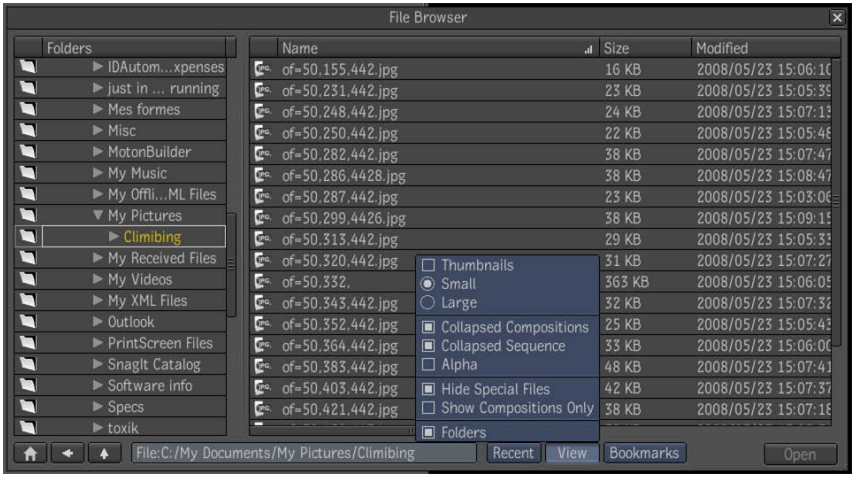
When you have created a new composition, you can import media and then add it to your composition and start working—see Importing Media on page 118.

To import media:

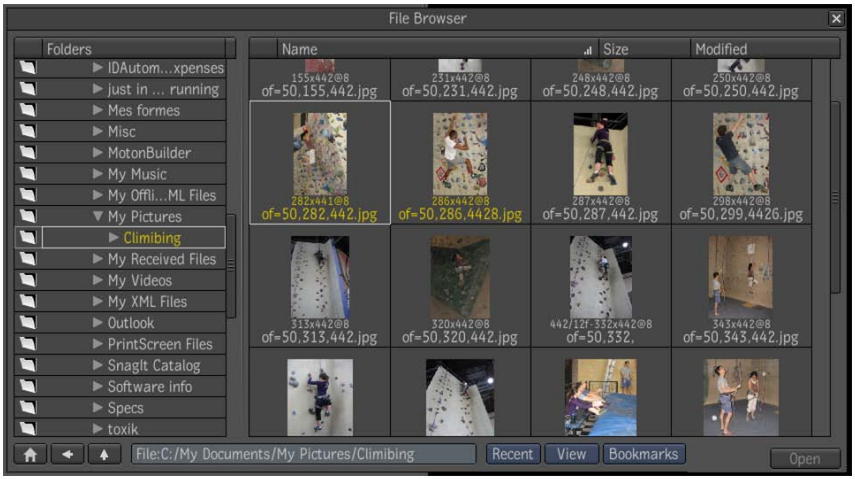
1 Select File > Import or press **Ctrl + I**(for Windows and Linux) or **Cmd + I**(for Mac OS).

The File Browser appears.

2 Navigate to the location (on your machine or network) where your media is stored. The View button at the bottom of the File browser allows you to change the Details view to a Thumbnail display in the Contents area, which is on the right side of the browser.



3 In the Contents area, select the media file(s) you want to import. To select multiple files, press **Ctrl** + click.



4 Click the Import button, or right-click one of the selected media files and select Import File(s).

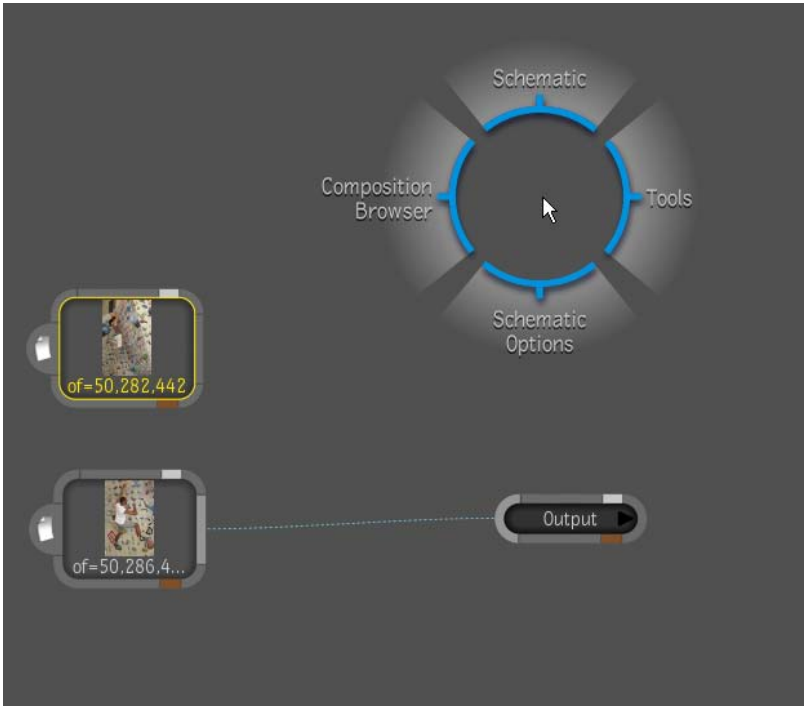
The media files are imported into the composition using an Image Import tool.

Add Tools

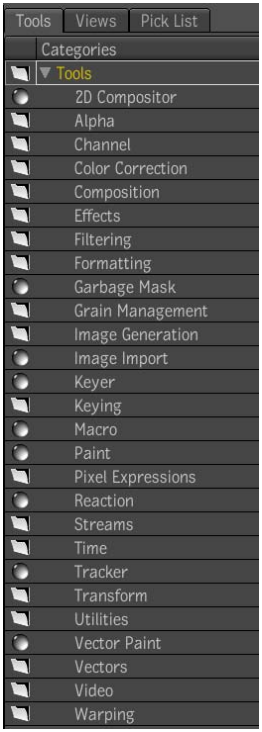
Once your media is added, you can start to add tools to your composition.

To add tools to your composition:

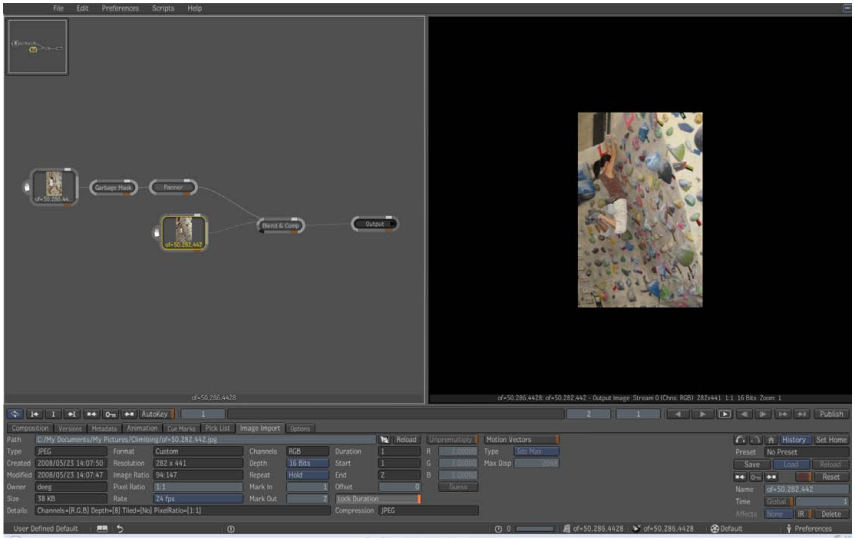
1. Middle-click or press the tilde (~) key to display the Gate UI and swipe east (right).



2. If it isn't already selected, select the Tools tab.

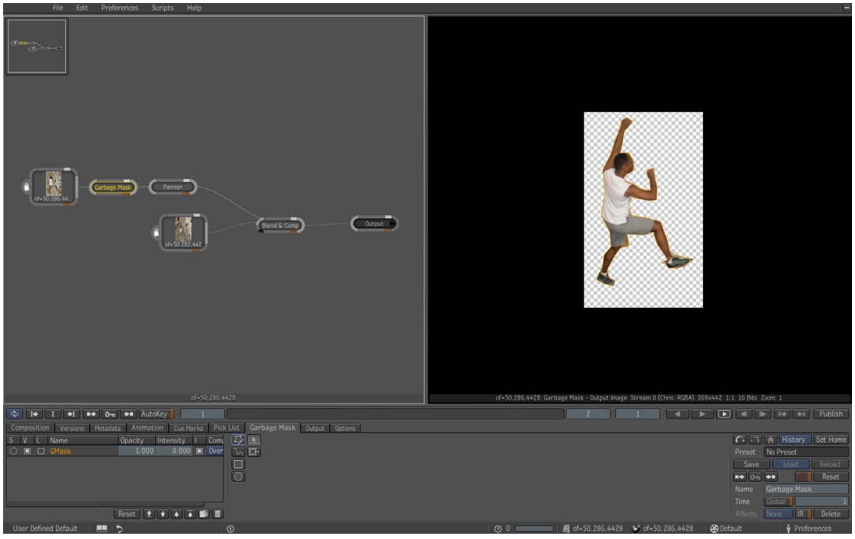


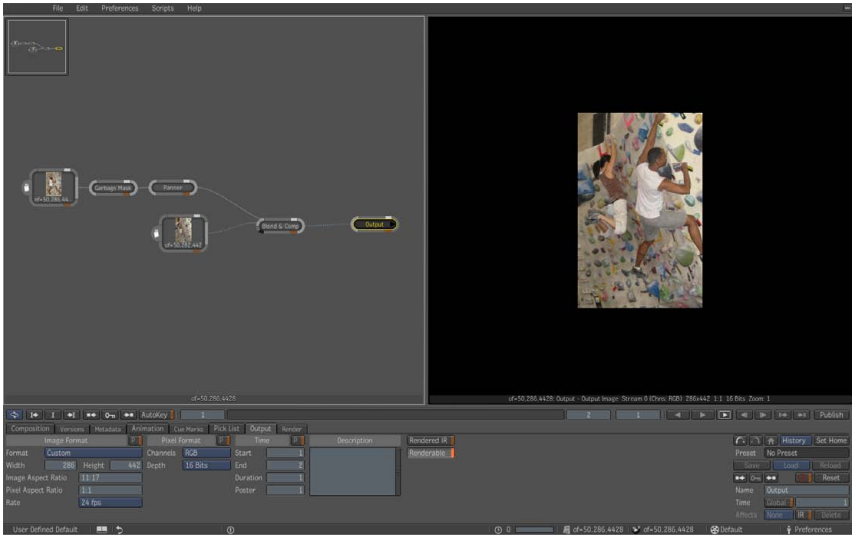
3 Select a tool and add it to your composition by dragging it into the Schematic. For this example, we will add a Garbage Mask, a Panner tool and a Blend & Comp tool.



4 Use the controls on each tool to modify your composition:

- Use the Garbage Mask to key out areas of the image which you don't need. See Garbage Mask Tool Tabs on page 538.
- Use the Panner tool to position the image. See Panner Tool on page 437.
- Use the Blend & Comp tool to blend the two images together. See Blend & Comp on page 367.





To Render the Composition

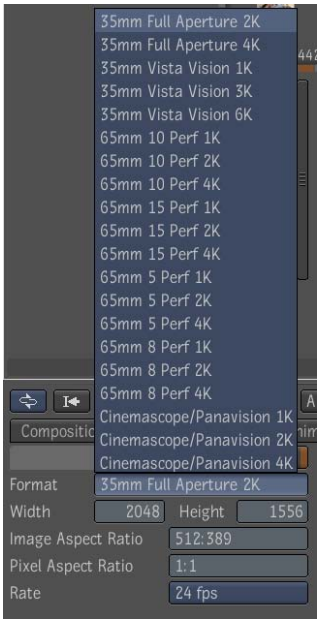
Now that you have completed your composition, you'll want to render it.

To render your composition:

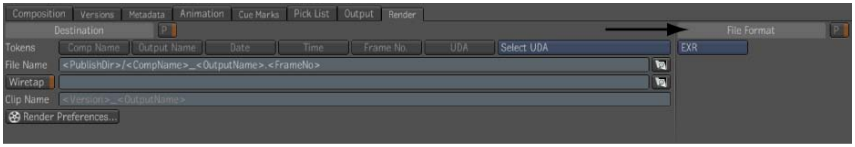
1 Select the output node.

2 In the Tool Details area, select the Output tab.

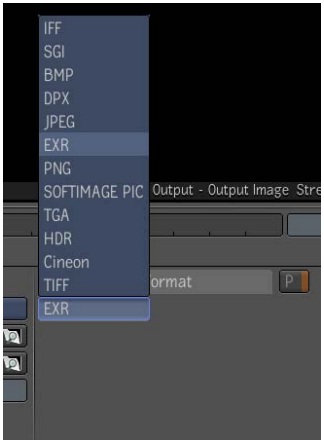
3 Set the image format to the proper format you want to render.



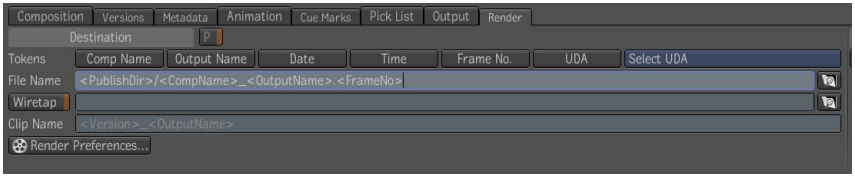
4 Click on the render tab.



5 Under the file format choose the sequence file format to be rendered.

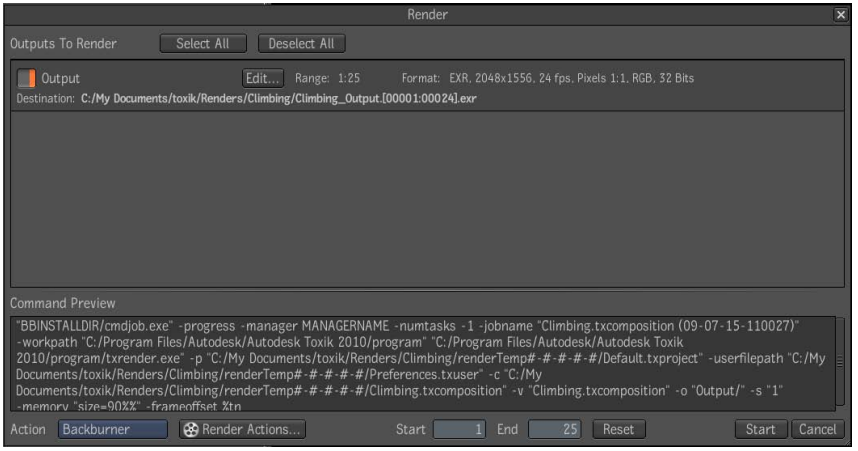


6 The default is to name the rendered file the comp name and the output name. Change the output name if desired.



7 Go to the file menu and choose render.

8 The Render dialog appears. Click the Start button to start the rendering process.



Part 2: Reference

Reference includes these chapters:

About Reference

5

Topics in this chapter:

About the Chapters

The Reference section goes into detail about Composite tools and functionality. This section allows you to build on the Getting Started section by providing in depth information about topics that were introduced in that section, as well as the rest of the Composite application.

The Reference section contains the following chapters:

Projects

6

Topics in this chapter:

About Projects

Projects allow users to keep a part of their preferences separately, in order to better accommodate separation of the work in multiple sub-tasks. For example, a common way of using projects would be to have one project per special-effects shot in a film. To this end, one would want, for example, to specify render directories that will differ from one shot to the other, without having to re-type them for every composition used within this shot. This is the kind of service that project management is intended for. By comparison, the User Preferences are more geared towards controlling the behavior of the user interface.

By default, project management is turned off in Composite, but can be enabled by visiting the User Preferences (Edit->User Preferences). In the tab labeled General, enable the Project Selection Dialog setting and click OK or Apply at the bottom right. The project selection will be enabled for the next time you start the application.

When project management is turned off, Composite uses a default project file that is called Default.txproject. You can view the location of this file by visiting the project preferences (Edit -> Project Preferences) and viewing the Information tab.

Project

The scope of a project depends on your facility's needs or individual projects. It may be a complete film, a special effects shot, an image sequence, or any other collection of shots/images. From an administrative point of view, a project is a set of folders and a set of preferences.

NOTE You can use scripts to automate the creation of projects, either to build a project structure from scratch, or to bring a project structure into Composite (including media). For more information on scripts, see the Composite Python® API online documentation (from the Windows Start menu, choose Programs > Autodesk > Composite > Composite Documentation > Scripting API).

Project Preferences

At the beginning of a project, you can define the following preferences:

- **General** The physical location of the home folder and the command line for the render queue.
- **Composition** The default format for any compositions you create in the project, as well as the name template used when creating new versions for this composition.
- **Render** To select a render folder, file name pattern, file format and optional wiretap location.
- **Render Actions** To create render actions and define the command-line associated with them.
- **Import** To define the folder in which imported media is stored.
- **Post-Import** To define tasks that can be executed after footage has been imported. Caching is one of these tasks, but you can define arbitrary post-import tasks that will become available in the contextual menus for importing footage.
- **Pre-Compositing** To set the properties for working with pre-comp files—see About Pre-Compositing on page 299.

- **Tools** Tool-specific preferences can be set here. Currently only the CTL tool allows these preferences — see CTL Tool on page 463.
- **Archive** The target directory and file name pattern of archives you create in Composite, and the scripts that are executed to perform, archive and restore operations.
- **Metadata** User Defined Attributes (UDA) editor for project-specific attributes. You can add, modify or remove UDAs that can then be used in token substitutions to form paths and file names.

For more information, see Setting Project Preferences on page 97.

Example Workflows

These examples let you perform administrative and project management tasks in Composite.

TIP You can use scripts to perform virtually any task you can accomplish through the user interface. For example, you might have a script that creates projects. Refer to the Composite Python API documentation. In the Windows Start menu choose Programs > Autodesk > Composite > Composite Documentation > Scripting API.

Workflow for administrative tasks during a project:

1. Import media—see Importing Media on page 118.
2. Set up and manage background rendering of the work produced during the project.
3. Archive projects periodically to back them up—see Archiving or Restoring on page 104.

Setting Project Preferences

You typically set project preferences when you first set up a project. If you modify preferences during the project, the changes take effect from that point on. The changes do not affect operations performed using the previous preference settings. For example, if you change the Channel setting in the Composition preferences from RGB to RGBA, the change has no effect on compositions you created previously; those compositions remain RGB.

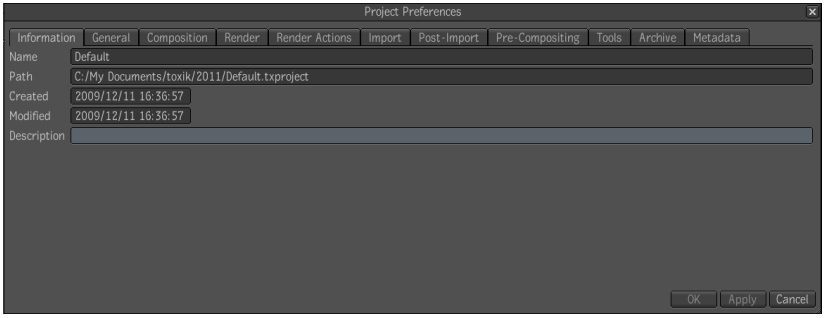
NOTE All operations you perform during a session use the project preferences of the currently open project.

NOTE Project preferences from a read-only file are not modifiable, unless saved under a different location with read/write permissions, and then reloaded.

To set project preferences:

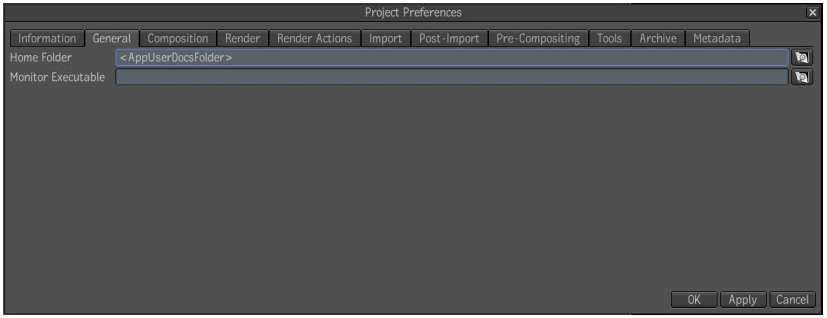
1. Open a project in Composite.
2. Do one of the following:
   - From the menu bar, select Edit > Project Preferences.
   - In the taskbar, click the project name or use the Ctrl + Alt + P hotkey.

   The Project Preferences window is displayed.



   Information on the project is displayed on the Information tab. It contains basic information that is set automatically when the project was created. You can enter a description of your project in the Description field. The remaining tabs contain the preferences you can set for the project.

3. Select the General tab and set the preferences for the home folder and the monitor executable.

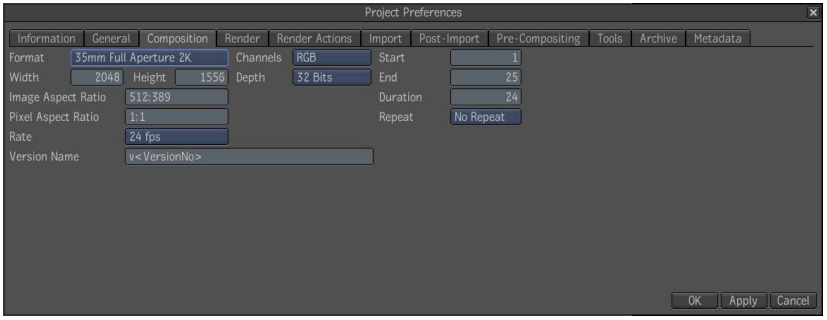


Home Folder Indicate the path for the home folder.

Monitor Executable Enter the command line that launches the render queue monitor.

4 Select the Composition tab and set the media preferences for the project. These preferences set the media format for any compositions you subsequently create.

NOTE These preferences apply only to compositions you create in the project. The compositions Composite creates when you import media retain their original format information.



Format Set the media format for the project. Select Custom to define a custom format.

Width, Height Displays the width and height (in pixels) of the selected format. If you selected Custom in the Format box, you can click in these boxes and edit the values.

Image Aspect Ratio Displays the image aspect ratio of the selected format. If you selected Custom in the Format box, you can click in this box and edit the ratio.

Pixel Aspect Ratio Displays the aspect ratio of each of the pixels of the selected format. If you selected Custom in the Format box, you can click in this box and edit the ratio.

Rate View the frame rate of the selected format. Click to display the list of frame rates available for this format, and select the one you want to use.

Version Name Enter a default name pattern for new versions created in compositions. You can enter any alphanumeric character, or click the Comp Name, Date, Time, or Frame No. buttons to insert any of those variables as part of the name. If you defined custom attributes, you can also insert any of these as variables in the file name. Click the Select UDA button to select the custom attribute you want to insert, then click UDA to insert it. Note that the file name, after all variables are resolved, cannot exceed 126 characters in length.Select a name for the composition version.

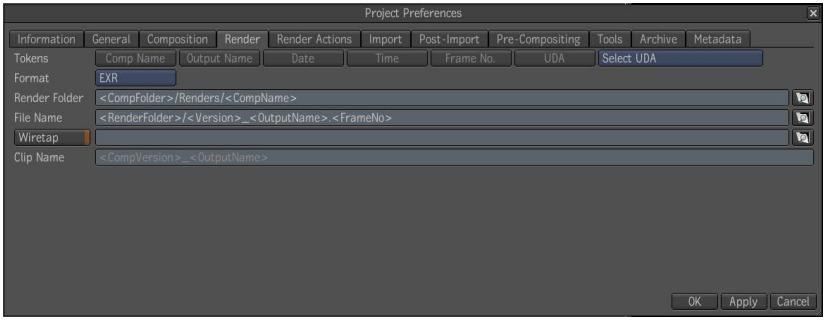
Channels Select the channels you want the media for the new composition to have by default. Select A to only have the alpha channel, RGB to have only red, green, and blue channels or RGBA to have all channels (red, green, blue, and alpha).

Depth Select the bit depth (8, 16, or 32 bits) for new compositions.

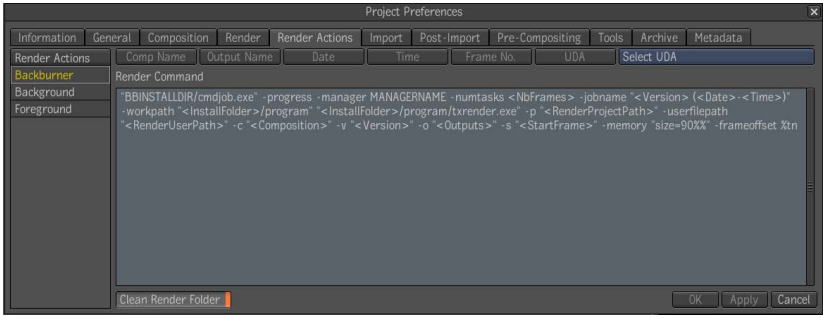
Start, End, Duration Set the start, end, and duration of the footage.

Repeat Select the default repeat mode (Hold, Loop, Ping-Pong, No Repeat) for new compositions when you play them in the Player.

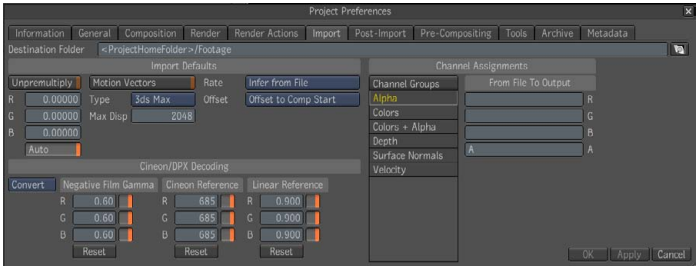
5 Select the Render tab. Select a default file format and set the render folder, file name pattern and Wiretap preferences.



6 Select the Render Actions tab.

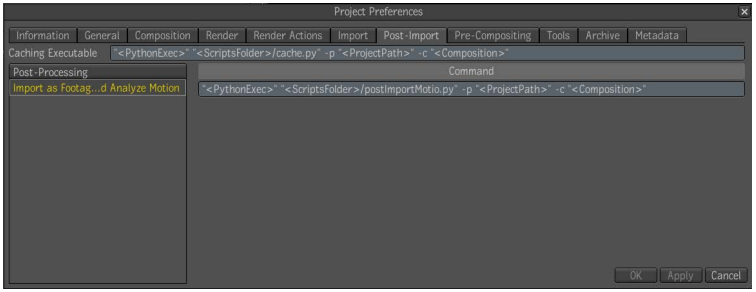


7 Select the Import tab to set preferences for the import operation—see Channel Assignments on page 128 and Channel Groups (Import Preferences) on page 131, Unpremultiply in Import Mode on page 134 and Specifying the Destination Format on page 135.

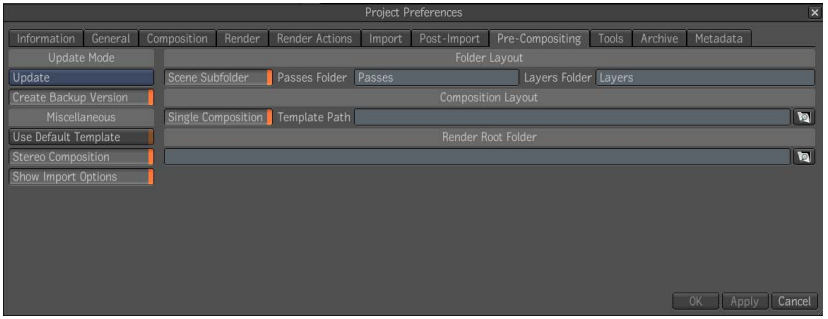


Destination Folder Enter the path to the project folder in which you want to import media, when using the Import as Footage actions in the contextual menus. By default the path is ad rectory inside the project home folder, but you can specify any folder.

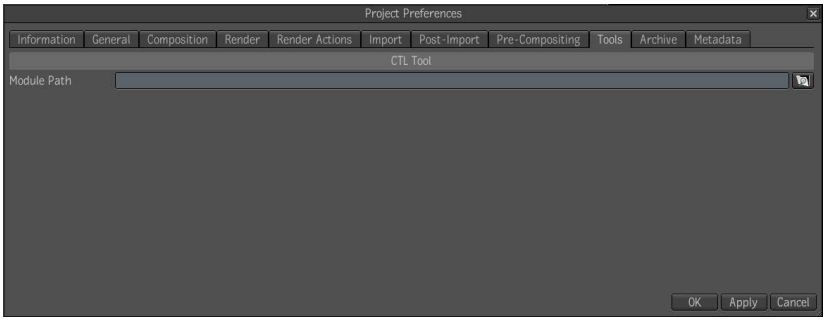
8 Select the Post-Import tab and add any command, such as Python command(s), that will become available as post-import processing actions in the contextual menus.



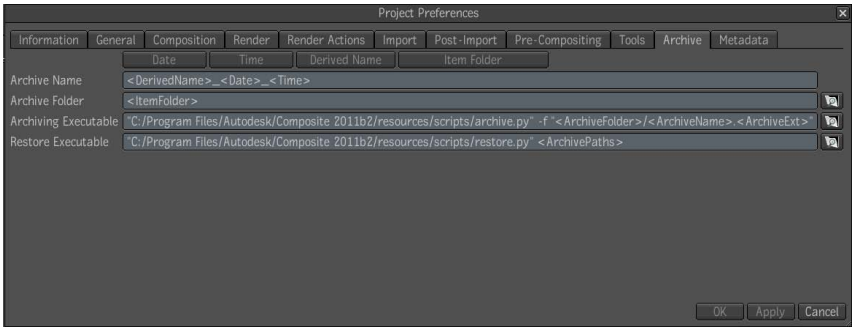
9 Select the Pre-Compositing tab and set the preferences for importing pre-comp files—see About Pre-Compositing on page 299.



10 Select the Tools tab and set the path for the CTL tool—see CTL Tool on page 463.



11 Select the Archive tab and set the preferences for your archive and restore operations—see Archiving or Restoring on page 104.



Archive Name The name template, using substitution tokens, that you select for the archive.

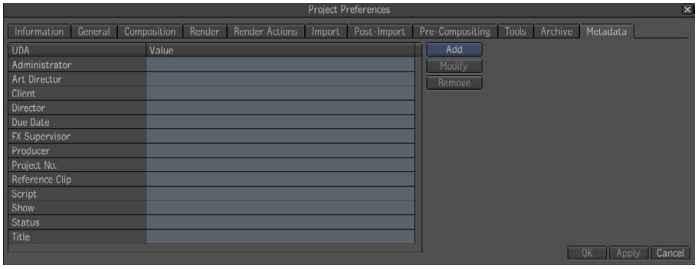
Archive Folder The folder or folder template (using substitution tokens) in which archives are stored.

Archiving Executable The command line that executes when you perform an archive operation in Composite. It starts the *archive.py* script, located in the *\resources\bgTasks* folder of the Composite program folder. Consult the script for an explanation of each of the command line options. You can click in the Archiving Executable field to edit the command line. For example, you can edit the argument for the silent option to prevent Composite from asking what archiving options you want to use (i.e. you are always comfortable with the defaults).

NOTE The following are illegal characters in file names and will cause the archiving to fail: *?:"<> (asterisk, question mark, colon, quote mark, open angle bracket, close angle bracket).

Restore Executable The command line that executes when you perform a restore operation in Composite. This command line launches the *restore.py* script, which is located in the *\resources\scripts* folder of the Composite program folder. Consult the script for an explanation of each of the command line options. You can click in the Restore Executable field to edit the command line (for example to change the script that the command line launches).

12 Select the Metadata tab and enter any information you'd like to describe the project—see Creating UDAs on page 177.



Things to Remember

- If you change preferences during the project, it does not affect operations performed using the previous preference settings.

NOTE Any change done to a project must be confirmed using the OK button (to confirm + dismiss) or Apply (to confirm and stay open).

Archiving or Restoring

An archive can contain an arbitrary selection of folders, compositions and project files. When archiving linked compositions, all versions of the linked composition are included. If you archive a link, Composite archives the object to which the link points, not the link itself.

When archiving a composition, you should save all versions. Otherwise, when you restore the composition, you may be missing some versions. Also, if you are archiving any dependencies, such as linked compositions, media, paint strokes, and LUTs all compositions are opened to locate them.

NOTE You cannot append to an archive.

You can archive and restore elements, such as projects, compositions, and folders. However, to enable you to selectively archive and restore certain elements of the archive (such as linked compositions, media LUTS, and paint strokes), Composite generates not just an *.xml* file, but also an archive folder containing the *.txarchive* file, as well as *.zip* files which store external dependencies.

NOTE Archives created with earlier versions of Composite have the *.dlarchive* extension and are supported for backward compatibility.

When you create an archive, you can produce compressed or non-compressed *.zip* files. The maximum size of a *.zip* file cannot exceed 2 GB. However, if it does, then the file is skipped and the archiving process continues.

The name of each *.zip* file is generated by the archive name, the type of external dependency it contains, and an index. For example, *MyArchive.MEDIA.0001.zip*, *MyArchive.MEDIA.0002.zip*, *MyArchive.PAINT_STROKES.0001.zip*.

The archive output folder uses the archive name you specify. This folder is created by default in the archive folder specified in the Project Preferences window (Archive tab)—see Setting Project Preferences on page 97.

You can configure the archive operation as either a local or remote background task. By default it is a background task on the local machine.

As for most tasks, you can use scripting to improve efficiency. Consult the online Composite Python API documentation for help with scripts (in the Windows Start menu, choose Programs > Autodesk > Composite > Composite Documentation > Scripting API).

To archive the current composition:

1. Select File > Archive.
2. In the dialog that appears, enter a name for the archive and click OK.

   NOTE The Python script for archiving does not recognize file names that contain any of the following characters: *?:"<> (asterisk, question mark, colon, quote mark, open angle bracket, close angle bracket). A file name with any of these characters causes the archiving script to fail.

   If you are archiving a composition, the following dialog appears in which you can choose the archiving options. External dependencies, such as linked compositions, paint strokes, 1D LUTs, and 3D LUTs are on by default for all external dependency types, with the exception of media.

3. Select the archiving options and click OK.

   All external dependency files are stored in one or many *.zip* files named after their type (for example, MEDIA, PAINT_STROKES, 1D_LUT, 3D_LUT).

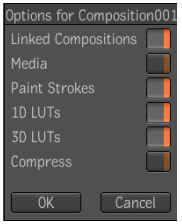
   WARNING Changing the name of a *.zip* file, moving or transferring an archive will result in undefined restoration behavior.

To archive an element:

1. In the File Browser, in the Folder list, navigate to and select the element you want to archive.
2. In the Folder contents area, right-click the element to archive and select Archive.
3. In the dialog that appears, enter a name for the archive and click OK.

NOTE The Python script for archiving does not recognize file names that contain any of the following characters: *?:"<> (i.e. asterisk, question mark, colon, quote mark, open angle bracket, close angle bracket). A file name with any of these characters causes the archiving script to fail.

If you are archiving a composition, the following dialog appears in which you can choose the archiving options. External dependencies, such as linked compositions, paint strokes, 1D LUTs, and 3D LUTs are on by default for all external dependency types, with the exception of media.



4 Select the archiving options and click OK.

All external dependency files are stored in one or many *.zip* files named after their type (for example, MEDIA, PAINT_STROKES, 1D_LUT, 3D_LUT).

WARNING Changing the name of a *.zip* file, moving or transferring an archive will result in undefined restoration behavior.

Archiving Linked Compositions

You can recursively archive the compositions to which a composition is linked. Linked composition archiving is optional and is on by default. Linked compositions are stored in the same *.xml* file as the compositions selected for archiving. You can archive a composition and its linked compositions in the same way that you archive a composition.

To archive a composition:

1 In the File Browser, in the Folder list, navigate to and select the element you want to archive.

2 In the Folder contents area, right-click the element you want to archive and select Archive.

3 In the dialog that appears, enter a name for the archive and click OK.

NOTE The Python script for archiving does not recognize file names that contain any of the following characters: *?:"<> (i.e. asterisk, question mark, colon, quote mark, open angle bracket, close angle bracket). A file name with any of these characters causes the archiving script to fail.

WARNING Changing the name of a *.zip* file, moving or transferring an archive will result in undefined restoration behavior.

4 Select Linked Compositions and any other archiving option, and click OK.

All external dependency files are stored in one or many *.zip* files named after their type (for example, MEDIA, PAINT_STROKES, 1D_LUT, 3D_LUT).

Restoring Archives

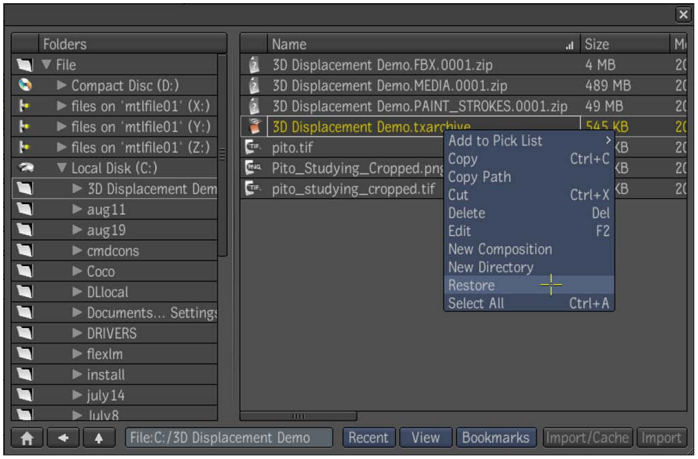
Unlike archiving, all the compositions in the archive, including the linked compositions, are restored. That is, there is no selective restoring of compositions.

NOTE When restoring an archive that contains a user and its settings, the user is skipped and a warning message is displayed.

To restore an archived element:

1 In the File Browser, navigate in the Folder list and locate the archive element in the default archives folder—see Setting Project Preferences on page 97.

2 Right-click the *.txarchive* file in the folder contents area and choose Restore.



Depending on the dependencies (if any), you are prompted to restore the archived dependencies.

3 Select the dependencies to restore.

4 Relocate the restored archive and its dependencies.

The external dependencies in the *.zip* files are restored. If some *.zip* files are missing because they were moved, transferred, or renamed, the external dependency files will not be restored. However, they can be relocated.

Relocating Dependencies

You can change the location of external dependencies in an archive when you restore the archive. The ability to relocate external files allows an archived file created on a Windows environment to be restored in a UNIX environment and vice versa.

Once you choose to restore an archive and any external dependencies, a relocation dialog box appears showing the path used by the external dependency files in the archive.

To relocate external dependencies:

➤ Navigate in the Folders area in the browser, and edit or enter an absolute path to relocate the external dependencies.

NOTE Some external dependencies, such as paint stroke files, do not appear in the relocation dialog box as these are stored by default in the shared storage folder. You can change the default shared storage directory in the Project Preferences window (Site Preferences tab)—see Setting Project Preferences on page 97.

The external files are restored according to the information entered in the relocation browser. The restored compositions are updated to point to the relocated files.

Changing Default Archive and Restore Settings

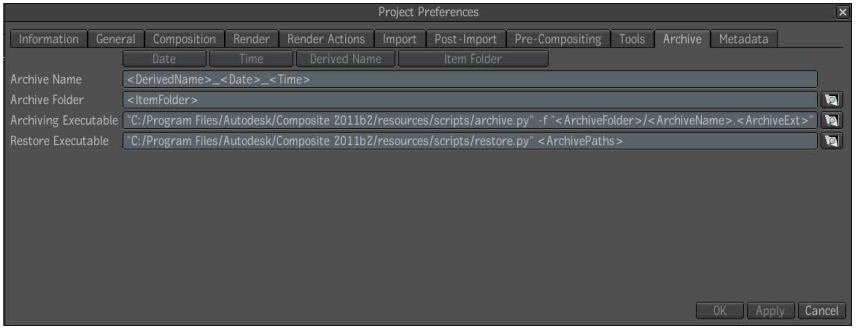
You can change the default settings for archiving and restoring.

To change the default archive and restore settings:

1. Do one of the following:
   - From the menu bar, select Edit > Preferences > Project.
   - In the taskbar, click the project name.

   The Project Preferences window is displayed.

2. Select the Archive tab and set the preferences for archive and restore operations—see Setting Project Preferences on page 97.



Wiretap

7

Topics in this chapter:

Accessing the IFFFS Libraries on the Network Using Wiretap

Composite supports Wiretap, an enabling technology that uses client-server architecture to enable applications running on a client system to access projects, libraries, clips and frames on a remote storage device. The data is presented in a tree-like hierarchy and is accessible via the Wiretap Client API. The ability for Composite to support Wiretap improves interoperability between Composite and IFFFS applications. Wiretap is comprised of a Client API that taps into the remote storage filesystem through a Wiretap Server plug-in running on the remote system via a high-performance network protocol.

To access images on the IFFFS filesystem and network, the Wiretap servers running on IFFFS workstations must be configured to offer Wiretap Path Translation Service—see the *Autodesk® Stone® and Wire® Release Notes*.

Wiretap Concepts

To ensure efficient browsing of the Wiretap network, here are some basic Wiretap concepts:

Wiretap Network: A collection of Wiretap-compliant servers that reside on a network domain.

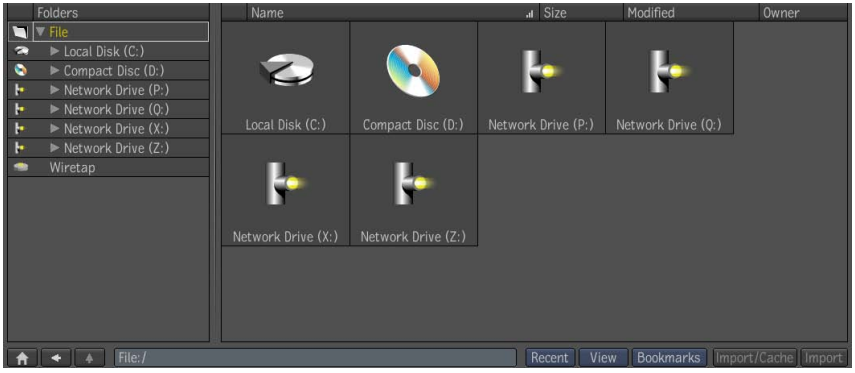
Wiretap Server: A single Wiretap-compliant server that exposes a proprietary database as a tree-like hierarchy of Wiretap nodes. A Wiretap Server is typically a daemon running on a host machine.

Accessing Media Via Wiretap

The IFFFS clips database is identified by the name of the host on which the Wiretap server is running (for example, **Wiretap:/<ServerName>**). You can access Wireptap as you would access any folder.

To access Wiretap:

1. Start Composite.
2. Open a File Browser and navigate to the Wiretap folder.



3. Expand the Wiretap folder. Wiretap displays a list of available servers.

If a Wiretap server does not appear in the list, it may be because it was not running when you started Composite or because an older version of Wiretap is used, in which case you can add the server manually to the list by typing its path in the text field (for example, **Wiretap:/MyServer/...**where MyServer is the name of the Wiretap server).

Accessing IFFFS Clips

When you import media from a Wiretap server, the Information tab displays information about the media and its path.

NOTE The path to clips is displayed, but not the path to images.

a) Clip name b) Clip path

You can access a subclip by specifying the range in the path field.

You can import singe images, as well as image sequences. You can also import certain images from a sequence of images.

In general, it should be possible to import all Wiretap clips. In practice though, it may happen that the media is not supported in Composite. In this case, the images will display a red X to indicate media that is not accessible. See Supported Media Formats on page 116.

An IFFFS clip can be made up of different types of formats. For example, it can be made up of standard file formats from a NAS, or it can be made up of images from a framestore or stonefs, or both. An example of an unsupported file format is the stonefs 12-bit compressed.

NOTE Contrary to standard imported file formats, no Gamma or Cineon correction options are available for Wiretap clips.

Importing Media

8

Topics in this chapter:

About Media

In Composite, media is a sequence of one or more images that you import. All media is represented as RGBA images. When you import media, Composite uses the information in the media file to determine whether the media is RGB, RGBA, or A. When you create a composition, Composite applies the project preferences to create compositions as RGB, RGBA, or A—see Setting Project Preferences on page 97.

Physical Location of Media

The media you work with can reside anywhere on the network. When you import media, you are not physically copying that media to another location, but rather you are creating a composition for that media in Composite; the actual physical location of the media is stored as information in the composition. Composite recognizes both drive letters and Universal Naming Convention (UNC) paths. Composite also supports Wiretap—see Accessing the IFFFS Libraries on the Network Using Wiretap on page 111.

Supported Media Formats

You can import media in any of the following formats:

NOTE When you import *.dpx* media from Autodesk Flame®, you import the media with LUT. To disable the import conversion, see Importing Media on page 118.

Format	File extension	Supported bit depths for imports
Bitmap	.bmp	8
Cineon	.cin	10
DPX	.dpx	8, 10, 16
HDR	.hdr	32
IFF	.iff	8, 16, 32
JPEG /JFIF	.jpg, jpeg	8
Open EXR	.exr	16 bit float, 32
Photoshop	.psd	8
PICT	.pict	8
PNG	.png	8
QuickTime	.mov	8
SGI	.sgi	8
Targa	.tga	8

Format	File extension	Supported bit depths for imports
TIFF	.tif, .tiff	8, 16, 32

NOTE Bit depths 8, 10, and 16 are integer unless otherwise indicated. Bit depth 32 is float.

You can produce results in any of the following media formats. For 4K and higher media, support is limited to tiled formats (OpenEXR, TIFF, and IFF).

Format	File extension	Bit depth of produced result
Bitmap	.bmp	8
Cineon	.cin	10
DPX	.dpx	10
HDR	.hdr	32
IFF	.iff	8, 16, or 32, depending on the composition. The media you create has the same bit depth as the composition. For 16-bit compositions, the bit depth of the created media is 16-bit integer (not 16-bit float).
JPEG /JFIF	.jpg, jpeg	8
Open EXR	.exr	16 bit float, 32
PNG	.png	8
SGI	.sgi	8
TIFF	.tif, .tiff	8, 16, or 32, depending on the composition. The media you create has the same bit depth as the composition. For 16-bit compositions, the bit depth of the created media is 16-bit integer (not 16-bit float).

NOTE Bit depths 8, 10, and 16 are integer unless otherwise indicated. Bit depth 32 is float.

Pixel Aspect Ratio

Composite guesses the pixel aspect ratio of images by comparing an image's resolution with a list of common combinations of pixel aspect ratio and resolution. You can, however, override the guess that Composite makes.

Composite can also store and retrieve the pixel aspect ratio information from file formats, such as *.bmp*, *.jpg*, *.tiff*, and *.dpx*. As a result, you get accurate pixel aspect ratio information of these types of images.

Mixed Resolutions

You can work with mixed resolutions in Composite. If a tool receives multiple inputs which have different image aspect ratios and bit depths, Composite changes the image aspect ratio and bit depth of all inputs to that of the primary input for the tool. When the original image aspect ratio/bit depth of an input does not match that of the primary input of a multiple input tool, the input tab on the tool node turns red to indicate the difference.

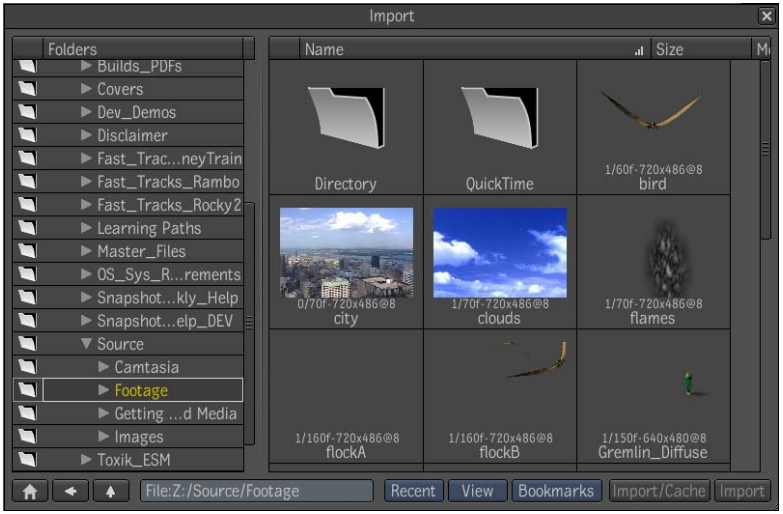
Importing Media

While you are working, you can import media at any time using the File Browser or the floating File Browser. You can browse the network and import media on a shared folder using the UNC path or the Python script. You can also drag and drop the media directly from the floating File Browser into a Schematic view or the Player. To share your imported compositions, you should use the Import button on the File Browser to create a Composite compositions folder or another folder that you use to store your media.

To import media with the floating File Browser:

1. Select File > Import or press **Ctrl + I (Windows and Linux) or Cmd + I (Mac OS)** to display the File Browser.

2 Navigate to the folder that contains the media you want to import.



3 Select the media files you want to import. You can either import, drag and drop, or use the UNC path to import the media.

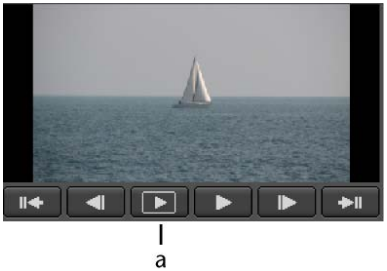
NOTE You can import multiple media files in the same folder, in a single operation. However, when you do this, you cannot set the import options for each file individually. To select multiple files, click the first file to import, then **Ctrl** + click each additional file to add it to the selection. **Shift** + click a file to select it and all files between it and the last file selected.

To import media with the File Browser view:

1. Middle-click or press the tilde key (~) to display the Gate UI. Swipe east and select the Views tab. Drag a File Browser to a view.
2. In the File Browser view, navigate to the folder that contains the media you want to import.
3. If necessary, view individual media files by selecting them and reviewing the Information tab that appears in the Tool UI.

NOTE You can import an image sequence by entering the UNC path in the Path field.

You can also play the media in the mini-Player at the bottom-left of the Tool UI, or click the Full Screen button in the mini-Player to play it in the full screen Player (press **F** to exit the full screen Player).



(a) Full Screen button

4. Select the media files to import. You can select the Options tab in the Tool UI and set a color conversion to be performed during import. The options available depend on the image format of the media file you

selected—see Log to Lin Workflows on page 126, Image and File I/O on page 127 and Channel Assignments on page 128.

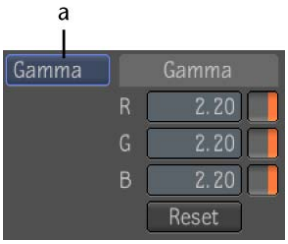
NOTE Values are not clamped when you import media in log format, so no image data is lost. All tools can work with all of the data contained in log format images.

NOTE You can import multiple media files in the same folder, in a single operation. However, when you do this, you cannot set the import options for each file individually. To select multiple files, click the first file to import, then **Ctrl** + click each additional file to add it to the selection. **Shift** + click a file to select it and all files between it and the last file selected.

The following displays the color conversion options for Cineon and DPX image formats.

(a) Select Raw to disable conversion.

The following displays the color conversion options for image formats other than Cineon or DPX.



5 (a) Select Gamma to adjust the Gamma options.

Select Rec. 709 to convert to the Rec. 709 video capture board standard or sRGB to convert to the sRGB standard. Select None to disable the conversion.

6 The destination folder is either the default Footage folder, or the one specified in the Import tab of the Project Preferences window—see Setting Project Preferences on page 97.

Drag the file directly into the Footage folder or click Import. You can create other folders to store the compositions that you create when you import. Dragging the file or using the Import button imports the media quickly and is useful when you do not need to immediately work with the media on the machine on which you are performing the import.

Things to Remember

- You can define the folder into which the media is imported by specifying it in the Project Preferences window. If necessary, you can also modify the Python script or command line used to cache media when you perform an Import/Cache.

The Information Tab

The Information tab in the Tool UI appears when you use the fixed File Browser view. The first and second columns contain properties that belong to a

sequence of image files. The third and fourth columns are the properties of the resulting footage composition.

- **Path** Location of selected file. If you select more than one file, the path of the last selected file is displayed.
- **Type** Displays the type of file that is selected—see Footage Role on page 137.
- **Created** Date the file was created.
- **Modified** Date the file was last modified.
- **Owner** Owner of the file on the file system. The creator of the file.
- **Size** The size of the selected file in megabytes.
- **Details** Information on the selected file, including channels, depth, pixel ratio and whether it is tiled, and so on.
- **Duration** The length of the selected file in frames.
- **Start** The start frame of the selected file.
- **End** The end frame of the selected file.
- **Poster** The thumbnail of the selected file that is displayed in the File Browser (in Proxy view) and in the mini-Player of the Information tab.
- **Format** The destination format of the selected file—see Specifying the Destination Format on page 135.
- **Resolution** The resolution of the selected file.

Composite guesses the image ratio, pixel ratio, and rate of the media you select from the Import view. The image's resolution is compared with a list of

common combinations of pixel aspect and resolution. The aspect ratio is related to the image ratio; changing it affects the image ratio.

- **Image Ratio** The image resolution of the selected media, taking into account the pixel ratio.

- **Pixel Ratio** The pixel ratio of the selected media. You can set the pixel aspect ratio.

- **Rate** The frame rate of the selected media. You can set the rate.

- **Source** Indicates the source from which your composition was originally created. In Composite, this property depends on the actual source; you can edit the source at any time. There are four different sources.

The composition was created	Source Initial Value
In Composite (new)	Composite
From an import operation (not from DND)	File
From Wiretap - IFFFS	The value of the tape name property
From Wiretap - generic	Wiretap

NOTE When you publish to Wiretap/IFFFS, the value of Source is written out to the *Tape Name* property of the IFFFS clip.

- **Channels** The channels in the selected file.

- **Depth** The bit depth of the selected file; you can modify the depth.

- **Repeat** The repeat mode when you play a composition.

- **Mark In** Set a mark in frame, the first frame to start import.

- **Mark Out** Set a mark out frame, the last frame to import.

- **Comp Start** When you create a footage composition, the start time is set after the number of the first frame of the sequence. Sometimes you will

want the start time to be unconditionally set to a different number, which you can do with the Comp Start editable value field.

- **Unpremultiply** see Unpremultiply in Import Mode on page 134.

- **Motion Vectors** see Importing Motion Vectors From Maya on page 712.

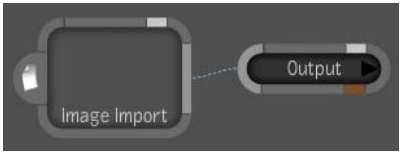
Using the Image Import Tool

The Image Import tool lets you quickly build a composition by dragging files directly into a composition from the file system. You can also drag media files into the composition to replace a media element without having to rebuild the dependency graph.

You can modify the following parameters in the Import Image Tool UI: Path, Pixel Ratio, Repeat mode, Mark In/Mark Out, and Offset (on a tool). You can also modify attributes on the Options tab.

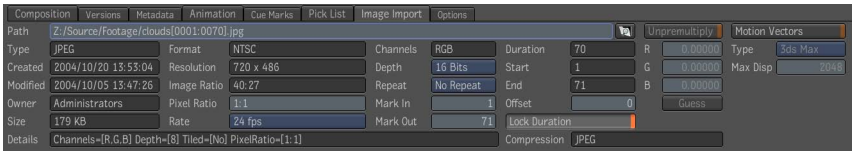
To import an image using the Image Import super tool:

1. In the Schematic view, press **Ctrl + N** (Windows and Linux) or **Cmd + N** (Mac OS) to create a new composition.
2. Middle-click or press the tilde key (~) to display the Gate UI. Swipe east and select the Tools tab.
3. Drag an Image Import tool to the Schematic view and connect it to the Output node.



4. Do one of the following:
   - Drag the media residing on the file system into the Path field in the Image Import Tool UI. When dragging, press **Ctrl + Tab** to switch to the Image Import tab.
   - Type a file path name in the Path field of the Image Import tab and press **Enter**.

- Copy (**Ctrl** + **C** for Windows and Linux or **Cmd** + **C** for Mac OS) a file path, paste (**Ctrl** + **V** for Windows and Linux or **Cmd** + **V** for Mac OS) it in the Path field of the Import Image tab and press **Enter**.



Log to Lin Workflows

There are several tools and controls to construct logarithmic to linear workflows depending on your needs.

Import and Export Workflows

Since Composite processes are optimized for linear data, when you import an image, the Import Options are in their Active state, and logarithmic data (Cineon, DPX) is automatically converted to linear data. If you need to export into Cineon or DPX format, the data is transformed back to logarithmic.

NOTE Since floating point formats (both 16-bit and 32-bit versions) are used, no data is lost going from a logarithmic encoding to a linear one.

In the Active state, you can make adjustments to both the film gamma and to the reference white of the image you are importing or exporting.

You can, however, override these settings by changing the default Active setting on import (and export) to Inactive. You can then make modifications to the image using the 1D and 3D LUT tools, as well as the Log and Delog tools.

Moving Imported Compositions into Project Folders

When you import existing compositions, the resulting compositions are automatically created in the project's Footage folder, or the folder specified

in the Project Preferences window (Import tab)—see Setting Project Preferences on page 97.

To move imported compositions into project folders:

1. In the File Browser, click View and select Folders.
2. In the Folders list, expand the folders for the project so that both the folder containing the imported compositions, and the folders where you want to move those compositions, are visible.
3. In the Folders list, select the folder containing the imported compositions.

   The contents of the folder appear to the right of the Folders list.
4. Select one or more compositions, then drag the compositions to the destination folder in the Folders list and release. To select multiple compositions, click the first composition, then **Ctrl** + click each additional composition to add it to the selection. **Ctrl** + click a composition a second time to remove it from the selection. **Shift** + click a composition to select it and all compositions between it and the last composition selected.

Image and File I/O

Composite supports OpenEXR 16-bit half float, 32-bit float and integer. This improves import workflow, footage versus composition distinction, and data extracted from the media.

Pixel Type	Depth	Usage
Half	16-bit floating-point number	For regular image data
Float	32-bit IEEE-754 floating-point numbers	Used where the range or precision of 16-bit number is not sufficient, for example, depth channel (Z)
UINT	32-bit unsigned integers	For discrete pre-pixel data such as object identifiers

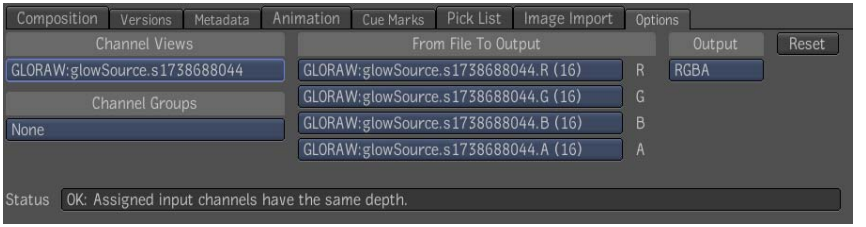
OpenEXR images can contain an arbitrary number and a combination of image channels. For example:

- Red, green, blue, alpha
- Luminance and sub-sampled chroma channels (Y/C)

■ Depth (Z), surface normal directions (XYZ), or motion vectors (XY)

Channel Assignments

Once you have imported your EXR media, you can use the Options tab to assign and import the channels. A channel view shows the channels that are present in the files and channel groups are defined in the project preferences.



NOTE These tools are also available when using the Image Import tool (Options tab).

For both Channel Views and Channel Groups, the None option always exists. Although it's not actually a group, it does allow you to assign any input channel to any output channel. The None option is also a fallback group when the file does not contain a channel that fits into any existing group definition. The None option is the default group selected, and the initial assignment always attempts to match a channel with itself, then assigns it arbitrarily by using an alphanumeric order.

Channel Views

The available channel views are listed. You can select a channel view to be displayed in the mini-player. Or if you are using the Import Image tool, the channel view is displayed on the tool node.

Multi-view OpenEXR files use the convention that channel names are composed of layer names separated by periods, with the final channel name at the end. The view name must be the ultimate layer name, that is, the penultimate period-delimited component in each channel name. In other words, the view name is followed by a period and a final channel name in the format *layer.view.channel* or *view.channel*.

OpenEXR channels are assigned to the Composite RGBA channels according to the following rules:

- If the channel name part of the input channel (the string after the last dot) is named “R”, “G”, “B” or “A” (case insensitive), then it is assigned to its respective R, G, B or A channel.

- If none of the input channels have an “.R”, “.G”, “.B” or “.A” suffix (case insensitive), then they are assigned in alphanumerical order:

To	Input Channels	Result
R, G, B, and A	4 or more	RGBA image
R, G, and/or A	2 or 3	RGB
A	1	A

Examples:

- “.M”, “.N”, “.O”, “.P” input channels are assigned to R, G, B and A respectively, resulting in an RGBA image.

- ".X”, “.Y”, “.Z” input channels are assigned to R, G and B respectively, resulting in an RGB image.

- “.U”, “.V” input channels are assigned to R and G respectively, resulting in an RGB image.

- “.Z” input channel is assigned to A, resulting in an A image.

Channel Groups

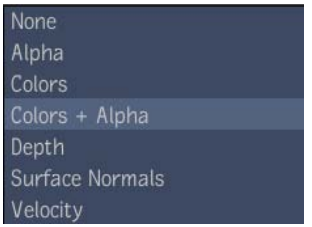
In the Options tab, you can assign any channel to the R, G, B and A channel. When the RGBA channels are available in the files, they are mapped by default to the same channels in the composition or output channel. During import, Composite verifies the mapping to determine that all the channels are the same bit depth.

This list provides the user-defined channel groupings and mappings. Composite provides a number of predefined names. Selecting a group performs the mapping defined by the group. Channels that were identified as being part

of a group in the preferences can be assigned to a corresponding channel group.

Each Channel can be assigned a field from the list, or as groups when using the buttons next to the menu.

The Channel Groups list shows only if the existing group was found in the file. The groups are determined by your project preference—see Setting Project Preferences on page 97.



The Channel Group options include:

Channel Group	Description
None	See Channel Assignments on page 128.
Alpha	Alpha channel only
Colors	Red, Blue, and Green channels only
Colors+Alpha	Red, Blue, Green and Alpha channels
Depth	Z-axis only
Surface Normals	X, Y, and Z-axis
Velocity	X and Y-axis

From File to Output Maps a channel present in the file to a channel (output) in the resulting node or composition.

You can assign a channel from the file to an output channel by clicking the Output Channel menu and selecting the channel wanted. As shown, the channels NX, NY and NZ have been grouped under the name Surface Normals, and were assigned to R, G and B.

Output Specifies the channels that will be present in the resulting node or composition. A reasonable default is set according to the channels present in the file. It is possible to change the output to any output of RGB, RGBA, or A.

Reset Resets the mapping in the File Browser to the original state, or to the last persistent state in the Import Image tool.

Status Verifies that the selected input channels all have the same depth. If the depths do not match, the images will still be imported properly.

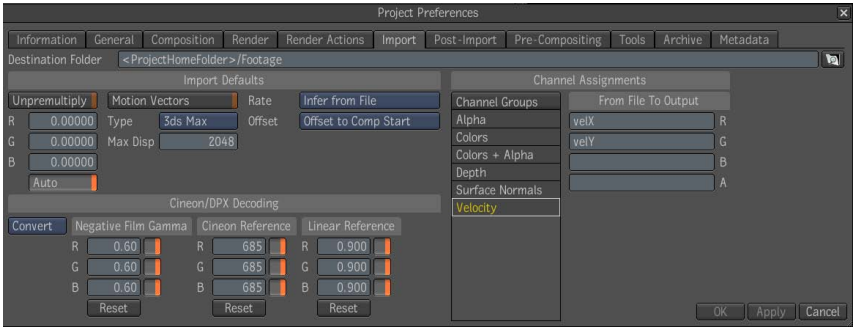
Channel Groups (Import Preferences)

In the Project Preferences window, you can define new groups in the Import tab. A number of default channel groups and predefined assignments are provided. Groups can be added, renamed, removed, and channel mapping can be updated.

The predefined groups are:

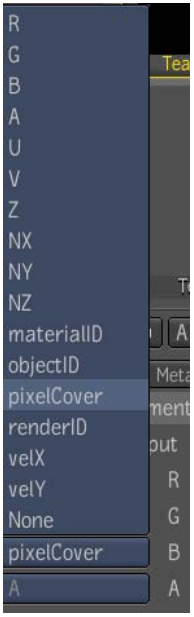
Group	Mapping
Alpha	A to A
Colors	RGB to RGB
Colors + Alpha	RGBA to RGBA
Depth	Z to R
Surface Normals	NX, NY and NZ to R, G and B
Velocity	velX and velY to R and G

NOTE The groups map to a subset of the channels available from the Splutter Fish plugin used by Autodesk 3ds Max: Velocity, Depth, and Surface Normal.



The selected group, Velocity, is predefined by Composite. It defines a mapping of channels named velX and velY to output channel R and G respectively. No channels are mapped to B and A.

Under From File To Output are four fields labeled R, G, B, and A. Each contains all the channels available in the EXR file.



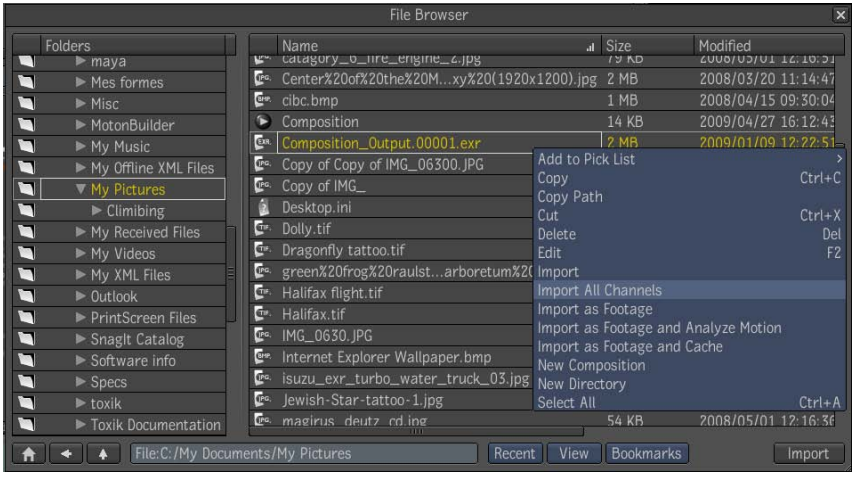
Channel	Description
R	Red Intensity
G	Green Intensity
A	Alpha Opacity
U	Equivalent to X-axis in a 2D plane
V	Equivalent to Y-axis in a 2D plane
Z	Z-axis or depth in a 3D plane
NX	X-axis Normals
NY	Y-axis Normals

Channel	Description
NZ	Z-axis Normals
materialID	A surfaces material ID is the value that determines which sub-material the surface will use when you apply a Multi/Sub-Object to a material to the object to which the material belongs.
objectID	The value that determines which object will be used when you apply it to the object.
velX	X-axis Motion Vector
velY	Y-axis Motion Vector
None	No channels selected

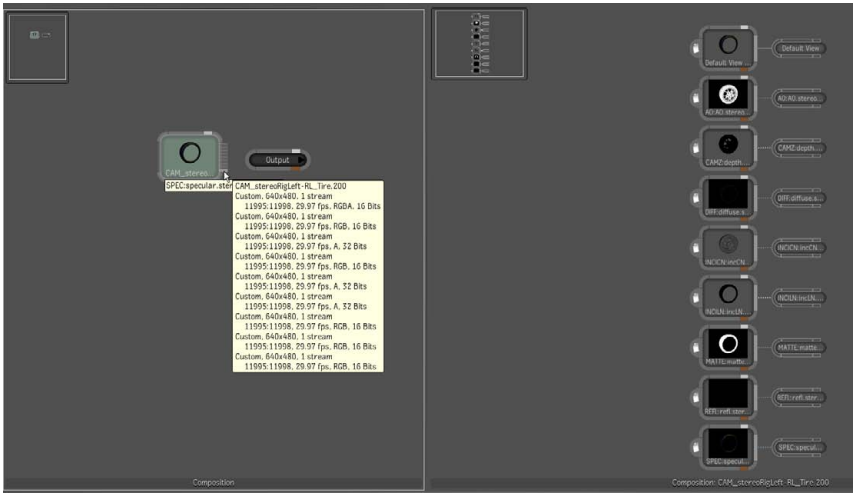
Import OpenEXR File

To import all channels of an OpenEXR file, do the following:

1. From the File menu, click Import.
2. When the File Browser opens, select an OpenEXR file.
3. Right-click on the file and select “Import All Channels” from the context menu.



Once you have selected “all channels,” a new node will appear in the Schematic, which displays tabs that each correspond to an output connection. In other words, all channels are imported into different import nodes which creates a group with multiple output connections that corresponds to each import node’s output. The following image displays a UI with two schematics. The Schematic on the left displays the group node, the Schematic on the right displays the import nodes with their output connections.



Unpremultiply in Import Mode

In Composite, it is typically not desirable to work with images that have their alpha premultiplied with the colors. If you are working with such an image, then you should manually add an Unpremultiply node from the Image Import tab or create a post-import script that does.

The Unpremultiply button is off by default for most formats, unless there is information in the file that indicates the presence of an alpha channel that

was premultiplied. OpenEXR files have their alpha channel premultiplied, so it is set to on by default. The following table summarizes the different cases.

Format	Unpremul-tiply	Value
Format without alpha	Off	--
Format with alpha, without premultiplied info	On	Off
Format with alpha, with premultiplied info	On	Depends on info
Format with alpha, conventional premulti-plied	On	Depends on convention

Specifying the Destination Format

When importing an image sequence, Composite determines the resulting composition properties with respect to channels and bit depth. Some controls that let you override these default properties include:

- Output channels RGB, RGBA, and A.
- Bit depth either 16-bit or 32-bit regardless of the files original depth.

Your initial bit depth values can vary depending on the properties of the image files.

- From a jpeg file, the default proposed is 16-bit
- From a dpx 10-bit int file, the default is 16-bit
- From a dpx 16-bit int file, the default is 32-bit
- From an hdr 32-bit, the default is 32-bit

You can always create an RGB, RGBA or A Output Channel composition. Any missing channels are filled with black.

Using the Gate UI to Create Footage

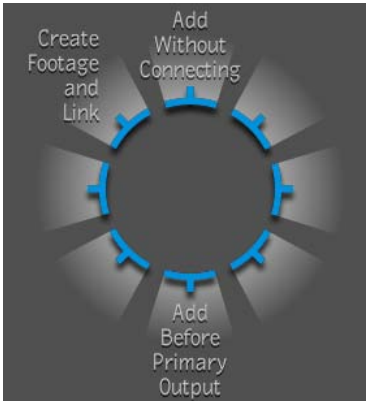
When you import media directly into a Schematic view or in the Player by dragging and dropping, a Gate UI appears with a number of choices depending

on the context and node selected. This allows you to create a footage composition in the Footage folder, and at the same time create a link node to that footage in the current composition.

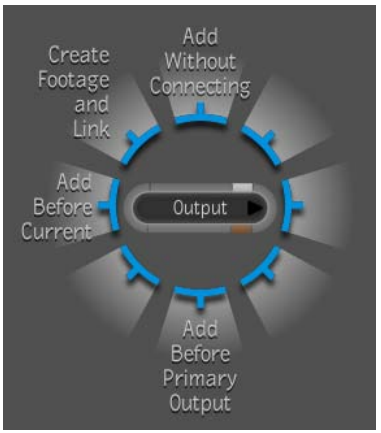
If you drag an image sequence to the Schematic and drop it on an input of a node, the Gate UI does not appear. However, if you drop the sequence anywhere in the Schematic or the Player, the Gate UI appears with several options.

To create footage and link in the Gate UI:

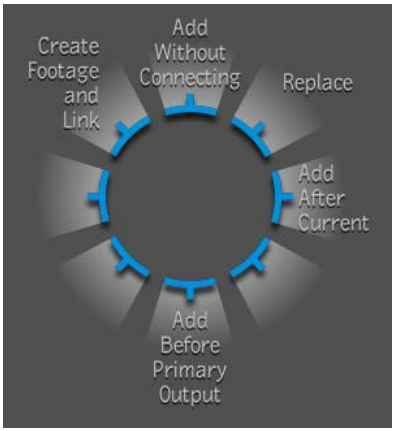
➤ Drag and drop your first image into the Player.
Three options are available from the Gate UI.



If you drag and drop your first image directly onto an Output node, you have a choice of four options.



If you drag and drop your second image into the viewer (instead of an Output node), you have a choice of five options.



Footage Role

How do you distinguish a composition representing footage from a composition you are using for compositing? Composite has a role for a composition named *Footage*.

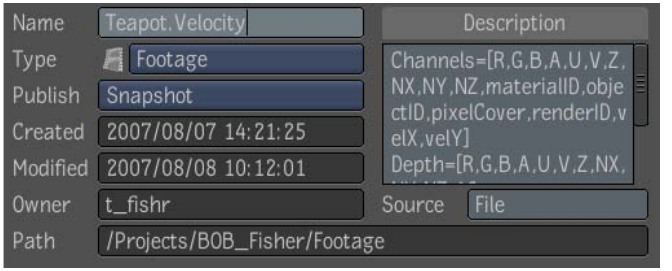
A composition with a Footage role displays a different icon in the Tool UI.

This role is not permanent and can be changed at any time. You can even create a composition from scratch and assign it a Footage role. When a sequence is imported, either by clicking the Import button or by dragging the sequence into a library folder, a composition is created with the Footage role. The role of a composition does not change its behavior at all in the application, and is only used as an indicator of the intended use of the object.

Footage Composition

When an OpenEXR image is being imported, a suffix corresponding to the Channel group is added if the selected Channel group is not a predefined group, such as Colors or Colors+Alpha.

The name would be *FileName.Alpha, FileName.Velocity*. This is useful when you need to import the same file twice, first to import the image channels and then to import the motion vectors.



NOTE The description of the footage corresponds to the description in the Information tab in the Tool UI—see The Information Tab on page 122.

Premultiplication

Premultiplication is the process whereby an image's RGB channels are multiplied by its alpha channel. Composite nodes expect unpremultiplied images. Furthermore, if you plan to use color correction, the images must be unpremultiplied. See the Premultiplication on page 337 chapter for a full discussion of premultiplication.

Getting Familiar with Your Workspace

9

Topics in this chapter:

Working with a Task-Based UI

Composite provides a flexible workspace environment that lets you work within the context of the current task at hand. You can choose from the available task presets to populate the user interface with the views you need to begin working immediately, or you can create your own presets, as well as customize existing presets according to your facility's workflow. When you are ready to begin working, you can immediately access tools and views using the Gate UI—see The Gate UI on page 12 and The Drop Gate on page 14.

What are Presets?

A preset is a saved configuration of views and settings appropriate for a particular task. For example, if you want to quickly assemble media and tools into a loose composition, you would build a three-view layout that displays a configuration that included a File Browser, a Layer Editor and a Player, as well as the Details area for displaying information about the composition or currently selected tool. Now, if you want to add and tweak layers in your composition, you could then switch to an editing preset to display only the Layer Editor and Player, and continue working.

You can also create new presets, and rename or delete existing ones as needed.

NOTE You can modify the default presets simply by selecting a different layout, or replacing a view by dropping a different view on the viewport.

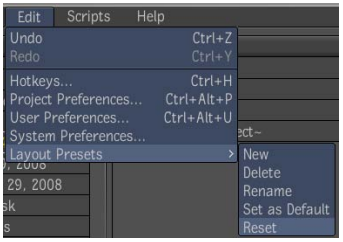
To choose default presets:

1. In the taskbar at the bottom-left, click the preset menu to display the list of presets.

2. Select a preset from the list or use its numeric keypad hotkey.

To select, delete, rename, set as default, or reset to a default preset:

1. Select the Edit Layout Presets and choose the New, Delete, Rename, Set as Default, or Reset command.
2. If you chose the New or Rename command, enter a name for the new/renamed preset and click OK or press **Enter**. If you chose the Delete command, select Yes or No in the deletion confirmation dialog.



NOTE You can create up to 18 presets that will have hotkeys to access them. The first nine will use the single number 1 through 9 on the keypad and the next nine will use the hotkey combination of **Ctrl** and 1 through 9.

To change the order of the layout presets:

➤ Hold **Ctrl** and drag the name of the preset to a new location higher or lower within the menu.

NOTE You must designate or set the default state of a preset. When you select the Reset to Default option, the preset will return to the state that you have designated as the default.

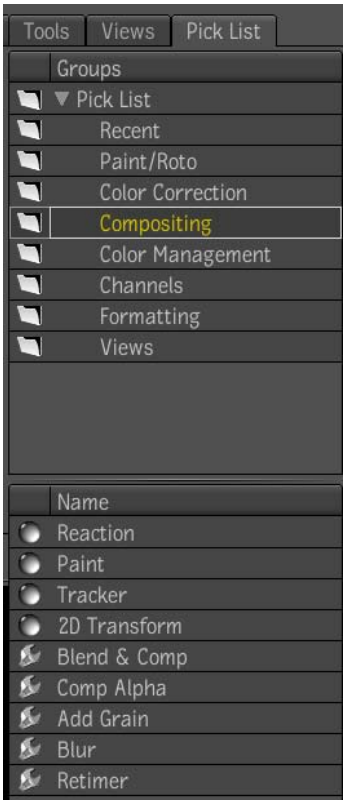
Pick List

The Pick List is a convenient way to access frequently used tools and other elements. You can add tools, views, and compositions to the Pick List. For example, if you add a Schematic view to the Pick List, then drag and drop it into a view, a Schematic view opens in that view. This is the same behavior that occurs when you drag the Schematic view from the Views tab into a view.

Each user has a Pick List. Its contents are independent of any project and are persistent across sessions.

The Pick List exists as a tab in two locations: the tool UI and through the east gate of the Gate UI.

Pick List tab in the tool UI



Pick List tab accessed through the east gate

Working with the Pick List

You can add and remove items in your Pick List, create a new Pick List group, delete Pick List groups you no longer use, and reorder Pick List groups. You can color code Pick List groups for easier recognition, as well as rename them. You can also set a Pick List as a default, reset a layout to that default, and restore the Pick List to the factory default group layout.

To add items to the Pick List:

➤ Do one of the following:

- Drag the item to the Pick List tab and drop it into a group.
- Right-click the item and select Add to Pick List. The Pick List group which is currently highlighted in the Pick List tab will be the destination Pick List group.
- Use **Ctrl** + drag to copy and paste an item from one Pick List group to another.

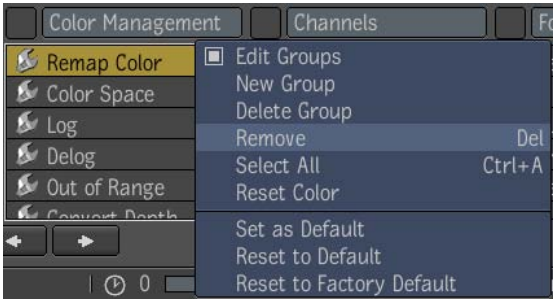
To use an item in the Pick List:

➤ Do one of the following:

- Drag an item from the Pick List to the dependency graph or into a view. If you add an item from the Pick List into a Player view, the drop gate appears. Add the item as per the selections in the drop gate. If you drag an item from the Pick List to a Schematic view, then connect it to the dependency graph as if you were dragging the item from the Tools or Views tab.
- Right-click in the view (Schematic, Player, Composition Browser, or Layer Editor) and select Add from Pick List.

To remove items from the Pick List:

1. Right-click and select the Edit Groups checkbox.
2. In the Pick List, right-click the item you want to remove and select Remove or press the **Delete** key.



NOTE You cannot undo the removal of an item from the Pick List. You can, of course, add that item to the Pick List again at any time.

To create a new Pick List group:

➤ Do one of the following:

- Right-click anywhere in the Pick List tab and select New Group.
- Drag an item to the Pick List tab and drop it into the tab's background.

A new Pick List group is added at the end when dropping on the background. Right-clicking and adding will add it after the currently selected group. This will allow new groups to be placed in-between existing groups.

To move the groups within the Pick List in the Tool UI:

1. Right-click and select Edit Groups.
2. Click anywhere in a group's window.
3. Use the right and left arrow buttons.

To delete a Pick List group:

➤ Right-click inside a Pick List group and select Delete Group.

The Pick List group is deleted.

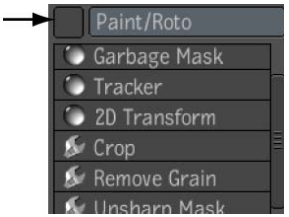
To rename a Pick List group:

1. Right-click and select Edit Groups.
2. Select the name of the Pick List group in the Name field at the top of the Pick List.
3. Select the current group name, type in a new name and press **Enter**.

To color code a Pick List group:

1 Right-click and select Edit Groups.

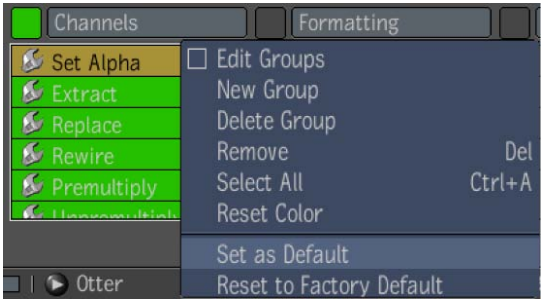
2 Click the color pot next to the Pick List group name field.
The color picker appears.



3 Select a color from the Color Picker and press OK—see Color Picker on page 154.
The Pick List group assumes the selected color.

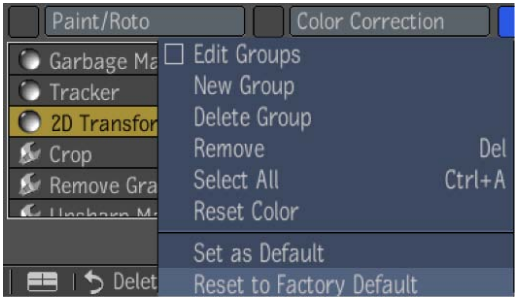
To set a Pick List group as the default group:

➤ Right-click inside the group and select Set as Default.



To reset a Pick List group to the factory default group layout:

➤ Right-click inside the group and select Reset to Factory Default.
All Pick List groups are reset to their factory default states.



Tool Presets

A tool preset is a tool you save with a customized set of properties. You can then use the tool preset as you would any other tool in a dependency graph. Sharing tool presets among a team is a great way to streamline a project. You can simply email tool presets to anyone involved in a particular project.

The tool presets you create are stored in the ToolPresets folder in the following location: <UserData>\toolPresets. When you create a tool preset, it is appended with the following extension: .txpreset. For example, in the following illustration, three tool presets were created from the Glow tool which reside in the<UserData>\toolPresets\Effects\Glow folder.

Working with Tool Presets

You can create a tool preset from the current tool after customizing it for your project. Once you create a tool preset, you can load it into the dependency graph and starting using it. Reloading the tool preset returns it to the settings in which you saved the tool preset.

NOTE You cannot create a tool preset from a group in the Schematic, but you can create a tool preset from a Macro.

The controls for using tool presets are located in the Tool Options area on the right side of the Tool UI.

(a) Tool preset controls

To create a tool preset:

1. From the Schematic, select a tool node from the dependency graph.
2. Do one of the following:
   - Right-click the tool and select Presets > Save.
   - In the Tool options area, click the Save button.
3. Type in a name for the tool preset and click OK.

   The name of the current tool preset is displayed in the Preset field in the Tool Options area.

To load a tool preset:

1. From the Schematic, select a tool node from the dependency graph.
2. Do one of the following:
   - Right-click the tool node and select Presets > Load.
   - In the Tool options area, click the Load button.
   - Middle-click or press the tilde key (~) to display the Gate UI and swipe east to display the Pick List, Tools, and Views tabs. Select a tool, right-click and select Presets. Then select a preset from the list and drag it over the selected tool node in the Schematic.

   The tool preset is loaded into the tool node and becomes the current tool.

TIP This is also a good way to reset the values of a tool preset back to its set values.

To reload a tool preset:

1. From the Schematic, select the tool node with the tool preset you want reload.
2. In the Tool options area, click the Reload button.

   The tool preset is reloaded into the selected tool node.

To delete a tool preset:

1. Navigate to the following folder: <UserData>/toolPresets.
2. Locate the tool preset you want to delete.
3. Select the tool preset and press Delete on the keyboard.

Using Tool Presets as a Display Modifier

You can apply any tool preset you created to a Player view or the fullscreen Player. This affects only what that particular Player displays; it has no effect on the composition data.

NOTE You can apply a tool preset to the fullscreen Player only if you entered it through a Player view.

To use a tool preset as a display modifier:

1. In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and swipe east to display the Pick List, Tools, and Views tabs.
2. Select the Tools tab.
3. Select a tool preset to apply to the Player, drag it outside the Tools tab, and do one of the following:
   - Pause over the Player view in which you want to apply the tool.
   - In the fullscreen Player, pause anywhere.
4. In the Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.

   The Player Options appear.
5. Position the cursor over the last tab of the Player Options and release.

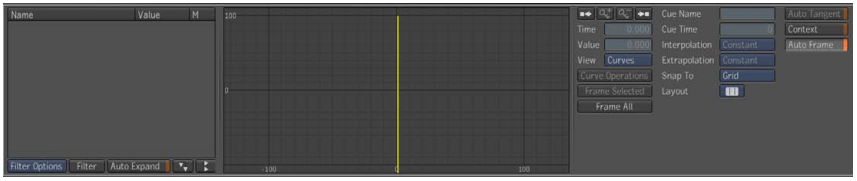
The name of the tab changes to that of the dropped tool preset. The Player updates to reflect the result of applying the tool preset to the image.

Using Tool Presets in the Pick List

For greater workflow efficiency, you can add tool presets to the Pick List and access your favorite or most frequently used tools—see Pick List on page 141.

Animation Editor

The Animation editor displays the animation curves and the animation tracks of a composition. The view has the same features as the The Animation Tab on page 661 of the Tool UI.



Setting the Animation Editor Options

You can set the way the Animation Editor is displayed, as well as the playback options.

To set the Animation Editor options:

1. With the cursor on the Animation Editor, middle-click or press the tilde (~) key to display the Gate UI and go through the south gate. The Animation Editor options appear in the Tool UI.
2. Select the Display tab to set the guide options.

   Pin Lock the time of the view to either Composition or Tool. If you lock the time to Composition, the view always displays animation curves in global time. If you enable Tool, the view always displays the local time of the tool node currently selected in the Schematic view.

   Composition Display animation curves or tracks in global time.

   Tool Display animation curves or tracks in the local time of the tool node selected in the Schematic view. If you did not select the Pin option

for a specific tool, the Animation Editor view updates each time you select a different tool node in the Schematic view to reflect the animation curve of that tool node

3 Select the Playback tab to set the time and frame rate.

4 Set the Viewer Time options.

Follow Master Uses global time. Deselect to use local time.

Offset Lets you set a frame offset for the time. The frame offset is with respect to the time you set for the view. For example, if you set a frame offset of 5 and selected the Follow Master option, the view always displays the frame that is five frames ahead of the frame currently running in global time. If you deselected Follow Master, the view always displays the frame that is five frames ahead of the frame running in the local time of the view.

Update on Play Updates the proxy at each frame when you play the composition. Deselect to update the proxy only when you stop playing the composition, and only at the scrub frames when you scrub through the composition.

5 Set the Frame Rate options.

Play All Frames Plays all frames in the composition, regardless of whether it maintains the frame rate set for the composition. Deselect to have the view maintain the frame rate for the composition, at the expense of dropping frames where necessary.

User Lets you set the frame rate at which you want the view to play the composition.

Stop on Drop Stops playing if a tool is dropped into the dependency graph.

Layer Editor

The Layer Editor gives you another view in which to perform compositing tasks with Reaction. It shows all layers for the currently selected Reaction node in your composition. Layers are displayed as rows in a table format and are stacked in the order in which you create the layers. You can choose to show or hide columns in the table, as well as show/ hide layers in your composition.

The Layer Editor contains columns that you can reorder by dragging a column heading. You can also right-click a column heading to display show/hide options. Any hidden columns are listed in the menu. The Layer Editor includes the following:

Column/Button	Description
V (visible)	Turns the visibility of the layer on and off. It is on by default.
Proxy	Displays a proxy of the image connected to the layer's source input.
Generator	Displays the name of the image generator connected to the layer's source input, if applicable.
Source	Displays the source name. You can use the **F2** hotkey to edit the source name. If you create a source without a layer, the text is red.
Layer	Displays the Layer name. You can use the **F2** hotkey to edit the layer name.
Layer Type	Click to select the surface geometry type for the current layer: Bicubic, Bilinear, Box, Frustum, Plane or Sphere. The default is Bicubic—see Working with Geometric Surfaces on page 254.
	Lets you move the selected layer up, down or to the bottom or top of the stack.

Column/Button	Description
	Duplicates a layer. This is particularly useful when you want to use the same image generator or media for multiple layers. When there is a source in the Layer Editor that does not have a layer assigned to it, the text is displayed in red—see Duplicating Layers on page 252.
	Deletes the selected layer.
Compositor	In a composition with multiple reaction compositors, selects which reaction compositor to work with.
Row Height	Lets you modify the row height in pixels for all rows in the Layer Editor.

Working with the Layer Editor

You can quickly build and edit multi-layer compositions using the Layer Editor. As in the Schematic view, you can use the Reaction drop gate to quickly build your initial composition, and can drop tools directly onto layers.

The Layer Editor's context menu also lets you access the Pick List, select the layer's component nodes, such as materials, and other tool nodes associated with the selected layer. When you select a Reaction node or tool, the appropriate panel in the Tool UI is displayed, so you can modify properties interactively. For more information, see Creating Sources and Layers on page 248.

To add tools to a layer in your composition:

1. Do one of the following:
   - Right-click to access the Pick List, or select the Pick List tab.
   - Middle-click or press the tilde (~) key and go through the east gate and display the Pick List, Tools, and Views tab.
2. In the Tools tab, drag the tool to the layer you want to modify.

   The tool is added before the source input for the layer, and its Tool UI is displayed.

NOTE Any tools added to a layer are added to the context menu. When you right-click the layer, you will see any tools in its branch.

Player

The Player lets you play or preview a composition, or preview media. There are three different versions of the Player:

- **Player view** Lets you play compositions and view the changes you make to compositions as you work—see Player View Default Settings on page 190.
- **Mini-Player** Lets you preview compositions located in the File view—see Using the Mini-Player on page 212.
- **Fullscreen Player** Is useful when you want a more immersive environment. You can use it to play compositions, work on compositions, and preview media—see Using the Fullscreen Player on page 215.

View Layouts

You can save up to four zoom and pan settings for the Schematic and Player viewers. These view layouts can then be recalled with a hotkey or using the **F1**-**F4** keys. This lets you switch back and forth between multiple areas of the viewer and is especially useful in the Schematic view.

View layouts are stored with the version of the composition. This allows for example, multiple Schematic viewers to share the same layouts. It also allows the layouts to be persistent from one UI preset to another and one session to another.

To set a view layout in a viewer:

➤ Do one of the following:

- Zoom and pan to the layout you want, and press **Ctrl** + **F1**. Repeat using **F2**, **F3**, **F4** as needed. Activate the view using its corresponding hotkey or by right-clicking and selecting Activate (corresponding view) from the View Layout sub-menu.
- Zoom and pan to the layout you want, right-click and select Set View Layout (1, 2, 3, 4), then select Set from the submenu. Activate the view using its hotkey or by right-clicking and selecting Activate (corresponding view) from the View Layout submenu.

NOTE When a view layout is saved for a Schematic viewer, a small number (1 to 4) appears in the view. This number represents the center point of that view layout.

To delete a saved view layout setting:

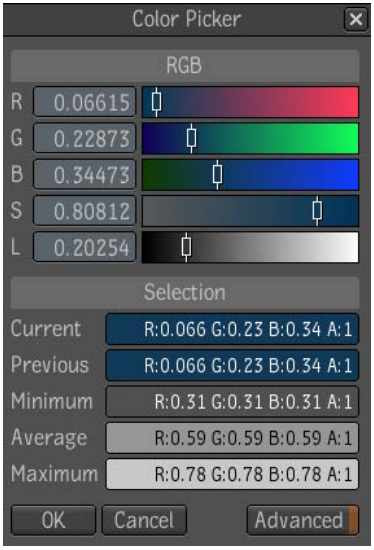
➤ Do one of the following:

- Right-click in the view and select Delete (corresponding view layout) from the View Layout sub-menu.
- Press **Ctrl** + **Shift** + (**F1**, **F2**, **F3**, **F4**). The saved view layout is deleted.

NOTE Composite saves the zoom/pan settings for viewers that have entered fullscreen mode so that the next time they go fullscreen, they maintain the same zoom/pan as before. When exiting fullscreen, the viewer's zoom/pan will be reset to the previous values.

Color Picker

There are two ways to launch the Color Picker. You can display the color picker by pressing **Alt** + **P**, or by selecting a color for a color pot from the screen, such as selecting a color for the Hue Shift color pot in the Color Curves tool.



To expand the Color Picker and see more options, click the Advanced button.

Working with the Color Picker

When working with the color picker, you can:

- Examine the values of each component of a color that appears on screen.
- Examine the minimum, maximum, or average color value in a selection.
- Select and store a color without launching the Color Picker.
- Create, delete, and rename a color palette.
- Adjust the colors in a color palette.
- Revert a color palette to its original colors.

NOTE You will always be in pick mode whenever the color picker is active. You can only pick outside of the floating window. When inside the window the cursor will return to the normal cursor for interaction with the UI controls.

Cursor in pick mode

To select a color for a color pot from the screen:

1. **Ctrl** + click the color pot for which you want to select a color.
2. Position the cursor over the color you want to select and click.
   The color pot updates to the selected color.

To select a color for a color pot:

1. Click the color pot for which you want to select a color.
   The Color Picker appears.
2. At the bottom center of the Color Picker UI, click the bit depth button and select a color depth for the Color Picker. Select % to work with percentage values (0.00% to 100.00%). The default is Float.
3. Do any of the following:
   - Use any of the color model areas to select a color.
   - Click anywhere on the screen to select a color.
   - Click any of the color pots on the right of the UI to select the color in that pot.
4. Click the OK button. That color pot now contains the color you selected.

To examine the values of an on-screen color:

1. Start the Color Picker by pressing the **Alt + P** hotkey combination.
2. At the bottom center of the Color Picker UI, click the bit depth button and select a color depth for the Color Picker.
3. Position the cursor over the on-screen color whose values you want to examine, and click.

   The HSV and RGB areas of the Color Picker tab, as well as the Current field of the Selection area all update to reflect the values of the selected color.

To examine the minimum, maximum, or average color value in a selection:

➤ Create a selection by pressing **Ctrl** + click, and dragging a marquis around an area of the screen.

The fields in the Selection area update to reflect the color values in that selection.

Field:	Displays the:
Current	Currently selected color.
Previous	Previously selected color.
Minimum	Minimum color value in the current selection.
Average	Average color value in the current selection.
Maximum	Maximum color value in the current selection.

To select and store a color without launching the Color Picker:

1. Press the **P** key to set the cursor to pick mode.
2. Click the color in the image you want to store.

   The color that you picked will be stored as the current color the next time the Color Picker is launched.

To select a color using the pop-up Color Picker without using a hotkey to display the Color Picker:

1. Click the color pot for which you want to select a color.

The pop-up Color Picker appears.

2 Click the Advanced button to expand the Color Picker and to display more options.

3 If necessary use the Bit Depth button at the bottom of the Color Picker to adjust the number of colors the Color Picker displays.

4 Do any of the following to pick a color:

- If you want to use the RGB model, use the slider bars to adjust the color.
- On the HSV tab, use the HSV wheel to pick a color.
- Use the cursor to pick an on-screen color.
- Enter the numeric value of the color in the fields next to the RGB or HSV color models.

The color pot you clicked to enter the Color Picker updates to reflect the new color, as does the color pot for the currently selected color in the Color Picker.

To exit the Color Picker and update the color pot:

➤ Press OK.

To exit the Color Picker without updating the color pot:

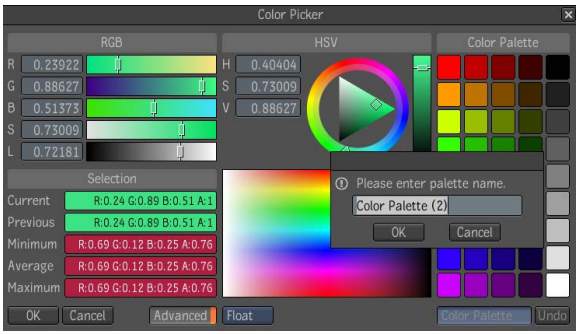
➤ Do one of the following:

- Press Cancel.
- Press the **Esc** key.
- Click the close button at the top-right corner of the Color Picker UI.

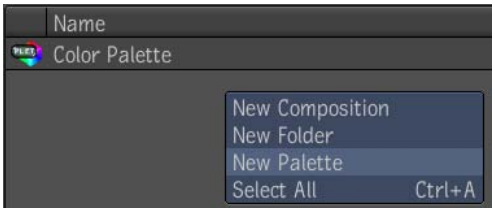
To create a color palette:

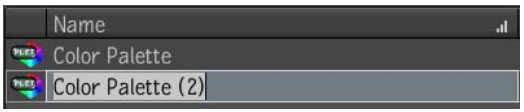
➤ Do one of the following:

- Click the palette name button and select New from the menu. Enter a palette name and click OK.



- Open your User folder located at the top of the Library folder structure and open the Color Palettes folder. Right-click anywhere in the view and select New Palette. A new palette is created. Type in a name and press Enter.





- Open your User folder located at the top of the Library folder structure and open the Color Palettes folder. Right-click a palette and select Copy. Right-click anywhere in the Color Palettes folder and select Paste. Click the new palette, press **F2**, and enter a name for the new palette. Press **Enter** again to complete the rename.

To delete a Color Palette:

➤ Do one of the following:

- Open the Color Picker by pressing **Alt + P**, then click the Color Palette name button you want to remove and select Delete from the menu.
- Open your User folder located at the top of the Library folder structure and open the Color Palettes folder. Right-click the palette you want to remove and select Delete.

To modify colors in a color palette:

1. In the Color Picker UI, use the HSV and RGB areas or the color spectrum area to set the color you want to insert into the color palette.
2. In the color palette, click in the color pot whose color you want to change, and hold until the pot updates to the new color.

To revert a color palette to its original colors:

➤ Do one of the following:

- Click the Undo button.
- Click the name of the color palette and select Reset.

To switch color palettes:

➤ Click the Color Palette button and select a color palette.

Calculators

There are two calculators available: the standard calculator that you can use to perform calculations and enter values into fields, and the Expresso Calculator that is displayed when you click the Expresso button or click in a numeric field that allows expressions. The Expresso Calculator extends the standard calculator to allow you to create expressions that generate values used in numerical fields.

(a) Expresso button

Not all fields accept expressions. Expressions can include composition parameters as variables.

In the Expresso calculator, you can add comments to the expression, allowing you to document what an expression does—see Animation on page 659.

To display the calculator or the Expresso calculator:

➤ Do the following:

- Press **Shift** + click in a numeric field to display the normal calculator or the Expresso Calculator.
- Press **Alt** + **C** at any point to display the normal calculator.
- Right-click a numeric field that allows expressions and select Set Expression.

The Expresso Calculator appears.

- **Shift + Alt** + click a field that allows expressions. The Expresso Calculator appears.
  The standard calculator appears if there is no expression.

NOTE If you use one of the last three methods and the calculator does not appear, the field cannot accept a value from the calculator. You must use an alternative method to enter the value.

To close the calculator:

➤ Click the "x" box in the upper-right corner of the calculator.

To input a value from the calculator into a numerical field:

1. **Shift** + click in a numerical field. The calculator appears.
2. Type a new value into the calculator.
3. Click in the numerical field again or press **Enter**. The value is updated.

Hotkey Editor

The Hotkey Editor enables you to save different Hotkey Catalogs, and share these Catalogs with other users. Use the Hotkey Editor to manage hot keys (keyboard shortcuts).

The Hotkey Editor allows you to:

- Quickly and easily modify the existing default hotkeys.
- Keep a copy of your user defined hotkeys for sharing and backup or updating your copy of Composite.
- Regroup or Document a list of all hotkeys.

Hotkey Catalogs

Composite introduces a new Hotkeys Catalog concept. Catalogs are place holders for user defined hotkeys. In other words, when a user modifies a hotkey through the Hotkey Editor, the hotkey entry is added to the active Catalog, keeping track of all customized hotkeys.

How Catalogs Work

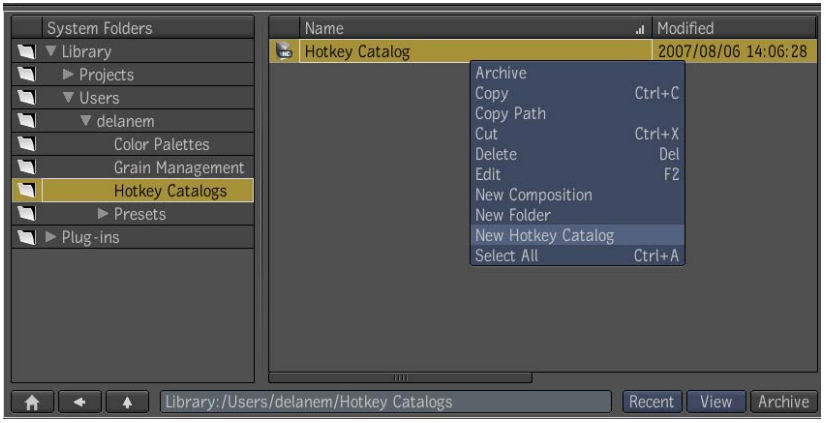
Catalogs are stored in the Hotkey Catalogs subfolder of the user home folder. Catalogs can be subsequently moved to the */Project* folder and shared with other users. Like Color Palettes, only one Catalog can be active at any time. The active Catalog can be chosen through the Hotkey Editor. When modifying hotkeys, if no Catalog exist for the current user, a new Catalog will be created automatically.

When starting Composite, the default hotkeys are registered in the hotkey registry and organized into the specified domains. If you have a Hotkey Catalog active, the hotkeys in the registry will be overwritten by the hotkeys found in the Catalog. When resetting a hotkey or a hotkey domain, Composite will remove these hotkeys from the active Catalog, and reset the hotkeys to the Composite defaults from the default hotkeys resource file.

Managing Catalogs

Creation and deletion of Catalogs can be done through the Hotkey Catalogs subfolder of the user home folder found in the File Browser. However, deleting a Catalog will not automatically reset all the hotkeys in the registry. Remember that a Catalog is simply a place holder of your user defined hotkeys.

Also, like any other database objects found in the user home folder, you can move Catalogs around, copy them, and archive them for sharing.

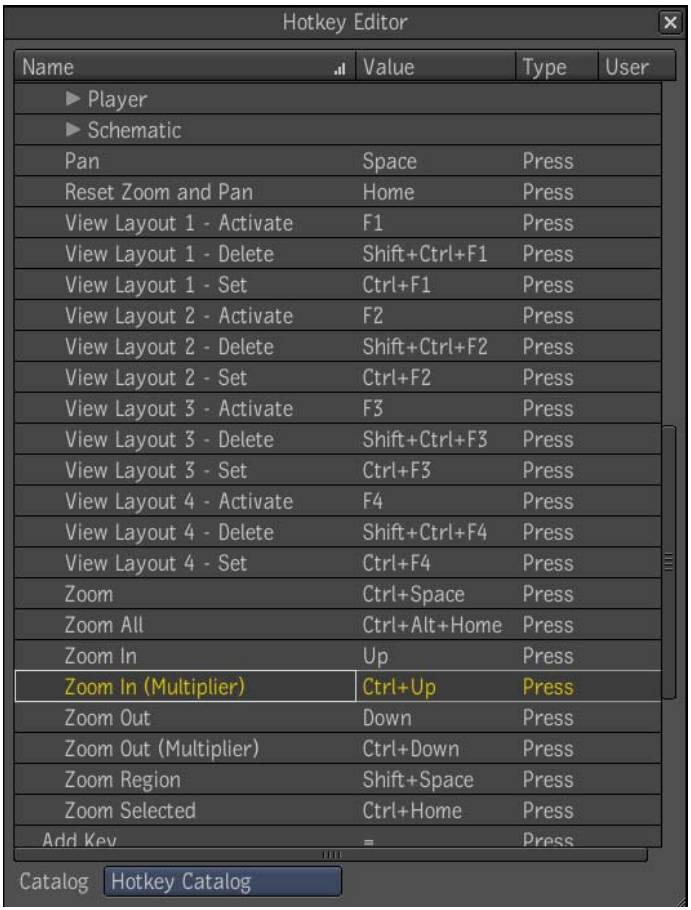


Hotkey Editor

The Hotkey Editor allows you to modify all the hotkeys described in the default Composite hotkey resource file.

Hotkey Editor Overview

The Hotkey Editor is a browser displaying all the hotkeys grouped into domains. Domains act as folders, and hotkeys as leafs of these domains.



The hotkeys displayed are those found in the default Composite hotkeys resource file and the current active Hotkey Catalog—see Hotkey Catalogs on page 163.

The active Hotkey Catalog is displayed in the Catalog menu at the bottom of the Hotkey Editor.

NOTE You can manage the Catalogs from the File Browser in the hotkeys subfolder of the user home folder, which is similar to how Color Palettes work.

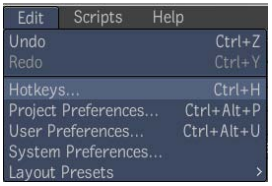
The browser section of the Hotkey Editor is composed of four columns.

- **Name** Represents the name of the domain or the hotkey action. This cannot be modified.

- **Value** Represents the keystrokes composing the hotkey. This can be edited by clicking the cell or by typing F2 after selecting the row.
- **Type** Represents the type (Press/Release) of the hotkey. This can be edited by a right-click edit option on the cell or by typing F2 after selecting the cell.
- **User** Indicates if a hotkey has been user defined. This would be indicated by a mark icon which is read only.

Accessing the Hotkey Editor

The Hotkey Editor can be accessed through the global hotkey **Ctrl + H**, which may also be changed by the user. The Hotkey Editor can also be accessed through the Composite menu bar (Edit > Hotkeys).



Editing a Hotkey

Editing hotkeys can be done in two ways.

To edit a hotkey:

➤ Do one of the following:

- Clicking on a VALUE cell representing the hotkey.
- Clicking F2 after selecting the row representing the hotkey.

Either way the Hotkey Editor will switch to “Learn Mode”. Every key pressed will be part of the new hotkey, see Hotkey Format on page 167, for the hotkey rule formats.

NOTE If the user tries to set a hotkey that already exists in the current domain or in the “Global” domain, they will be notified with a warning. The hotkey will be set, but remember that a local hotkey has precedence over a Global hotkey.

Conflicting Hotkeys

Conflicting hotkey rows in the Hotkey Editor are displayed in red.

To resolve a conflicting hotkey:

1. Right-click a conflicting hotkey to see the "Go To Conflict" option.
2. Choosing this option on the menu, or pressing the **F3** hotkey will take you to the conflicting hotkey, select it, then turn the learning state ON. This allows you to edit a conflicting hotkey on the fly.

Hotkey Format

Composite has a hotkey format standard that needs to be followed in order for a hotkey to be valid. When creating and editing hotkeys, the following keystroke conventions apply:

- All modifiers **Ctrl**, **Alt**, and **Shift** must come before any other keystrokes.
- Modifiers must be entered in the above order.
- A hotkey can be composed of one to three modifiers and one normal key.
- You are limited to four keystrokes per hotkey operation.
- Key strokes in a hotkey operation are pressed at the same time.
- You cannot use a combination of keystrokes more than once.

The formatting will be handled by Composite automatically, meaning that as soon as you release the last key, the hotkey string is automatically generated, and entered in the value editor.

Resetting Hotkeys

To reset a hotkey:

1. Right-click a hotkey in the Hotkey Editor.
2. Select Reset.

   This will reset to the Composite default and remove the hotkey entry from the active Catalog.

To reset all hotkeys:

1. Right-click a hotkey in the Hotkey Editor.

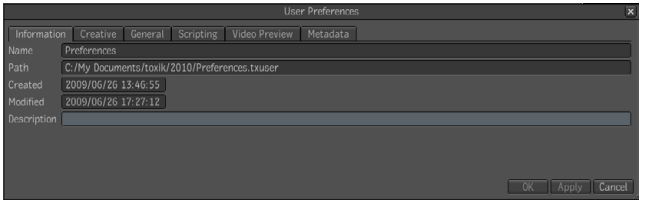
2 Select All.

3 Select Reset.

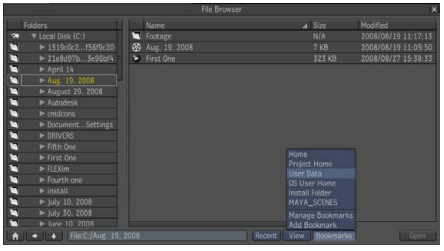
All Composite hotkeys are reset and hotkey entries are removed from the active Catalog.

Setting User Preferences

A user is defined as a group of settings that belong to a user. The settings include: color palettes, hotkey catalogs, presets, and bookmarks. Preferences are stored in a file named *.txuser* under your user home directory. User preferences never need to be saved as it is done automatically any time you make a change to your preferences.



In the File Browser, there is a bookmark set to your User data.



Each user has a set of preferences. When you login, you can set these preferences to tailor the user interface to your liking. There are two ways to access the user preferences: from the menu bar at the top of the Composite application and from the taskbar at the bottom of the application.

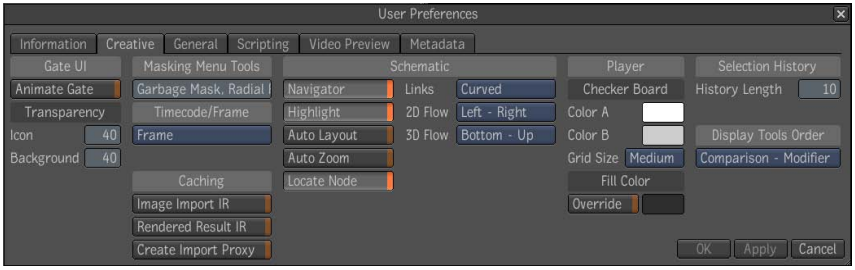
To set user preferences:

1 To open the User Preferences window, do one of the following:

- From the menu bar, select Edit > User Preferences.

2 ■ In the taskbar, click your user name.

3 Select the Creative tab.



4 Set the Gate UI preferences.

Animate Gate Determines the way the panels move into the interface when you swipe through the Gate UI. Select this option to have the panels slide into the interface. Deselect to have the panels display immediately in their final position in the interface.

Icon Set the transparency of the nodes in the Schematic overlay that appears when you swipe through the north gate of the Gate UI. A value of 100 means completely transparent and a value of 0 means completely opaque.

Background Set the transparency of the background of the Schematic overlay that appears when you swipe through the north gate of the Gate UI. A value of 100 is completely transparent; a value of 0 is completely opaque.

5 Under Masking Menu Tools, add masking tools to the Masking tab Create list.

6 Under Timecode/Frame, display the composition in frames or timecode.

7 Under Caching, turn on or off Image Import intermediate results, Rendered Result intermediate results, and the Create Import Proxy option.

8 Set the Schematic preferences. Set these default values for options that appear when you right-click outside of a node in the Schematic view:

Navigator Disable the north gate schematic mini viewer.

Highlight Highlight tabs and the connections between nodes when the cursor is passed over it.

Auto Layout Automatically organize all of the nodes in the dependency graph.

Auto Zoom Automatically zoom the view to include all nodes when you drag a node outside of the visible area.

Links Select the style (Curved, Angled, or Straight) of the lines that connect nodes in the

Schematic view.

2D Flow Build the dependency graph in the Schematic view in a left-to-right or top-down direction.

3D Flow Build the dependency graph in the Schematic view in a left-to-right or bottom-top direction.

9 Set the Player preferences.

Checkerboard Foreground/Background Set the foreground and background colors of the checkerboard pattern that appears in the Player when you enable RGBA in the Player preferences. This pattern makes it easier to see the alpha channel of the media. To change the color, click the color pot and use the Color Picker that appears to select a new color.

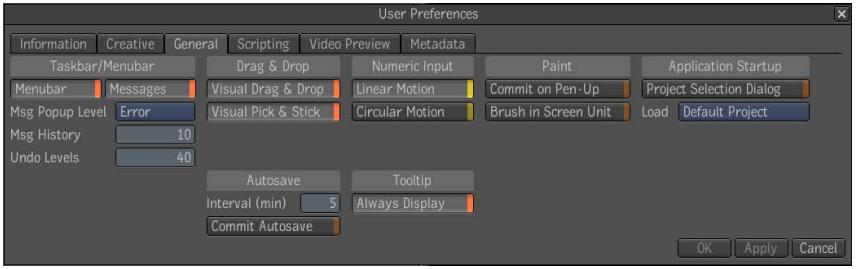
Grid Size Set the size (Small, Medium, or Large) of the individual squares in the checkerboard pattern.

Fill Color Set the color of the Player by clicking the Override button. Click the color picker and select a color.

10 Under Selection History, set history length up to 50 selections. The default is 10. Set the selection history preferences.

11 Under Display Tools Order, place the Comparison tool before or after the display modifier—see Comparison Tool on page 806.

12 Select the General tab, which contains the taskbar, drag and drop, numeric input, paint, application startup, Autosave, and tooltip preferences.



13 Set the taskbar preferences.

Menubar Show or hide the menu bar.

Messages Display messages in the messages area of the taskbar. You can click the messages area to display a list of messages received to date.

Msg Popup Level Select the types of messages to appear as pop-ups. The order of the types in the list is meaningful. Each type includes all types that appear above it in the list. For example if you select Information, a popup message appears for all information, warning, error and fatal error messages. If you select Fatal Error, a popup message appears only for fatal error messages.

Msg History Set the maximum number of messages in the list that appears when you click the message area of the taskbar.

Undo Levels Set the maximum number of items in the Undo list that appears when you click the Undo list in the taskbar.

14 Set the drag and drop preferences.

Visual Drag & Drop See a ghosted image of the selected item as you drag and drop it in the interface.

15 Set the numeric input preferences.

Linear Motion Use a mouse or pen with a linear motion to increase or decrease numeric inputs when in a value editor. This is the default setting.

Circular Motion Use a mouse or pen with a circular motion to increase or decrease numeric inputs when in a value editor.

16 Set the paint preferences.

Commit on Pen-Up Commit strokes when the pen is lifted from the tablet. The default setting commits paint strokes when the curser exits the viewer or when any key is pressed.

Brush in Screen Unit Retain the brush when zooming in the Player view.

17 Set the Application Startup preferences.

Project Selection Dialog View Project Selection dialog at start-up.

18 **Load** Choose to load either a default project or the most recent project.

19 Set the autosave preferences.

Destination The location of your autosave files.

Interval The length of time between auto saves, between 0 and 240-minute intervals.

20 Set the tooltip preferences.

Tooltip Turn tooltips on or off.

21 Select the Scripting tab.

Script Output Choose to show the script output. You can also enable the time, stream and context for the script output, and define the maximum number of lines.

22 Select the Video Preview tab. See Previewing Video on page 172.

Device Choose a video preview device: Mono Preview, DLP-3D preview or Segmented Frame Preview.

Show Settings Turn this option on or off.

23 Select the Metadata tab.

UDA Add, modify or remove UDAs. For more information on UDAs, see Creating User Defined Attributes on page 176.

Previewing Video

If your system has the appropriate hardware, you can preview your work on a broadcast video monitor. Video previewing is available for Composite on Windows 32-bit. Depending on the level of performance of the video output card and the graphics card, as well as the host system CPU, Composite can achieve real-time previewing of 1920x1080 24P resolutions (and below) of 8-bit RGB or RGBA material.

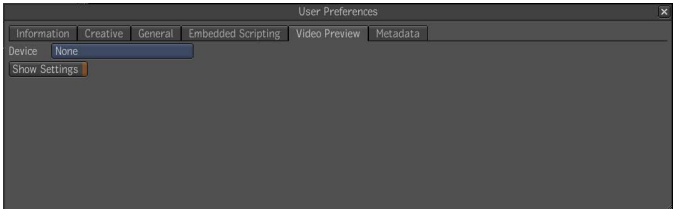
Composite will only display 8-bit pixel formats on the video preview device. To display 16-bit float and 32-bit float images, you must convert them to 8 bits. You can do this by using a display modifier in the Player that converts to 8 bits, such as the sRGB, 3D LUT, or Convert Depth tools—see About Utilities on page 805. Or you can add and connect such a node inside your composition to explicitly convert your material to 8 bits. Also, see Initialization Variables on page 873 for the initialization variable for previewing video.

Setting Up Video Previewing

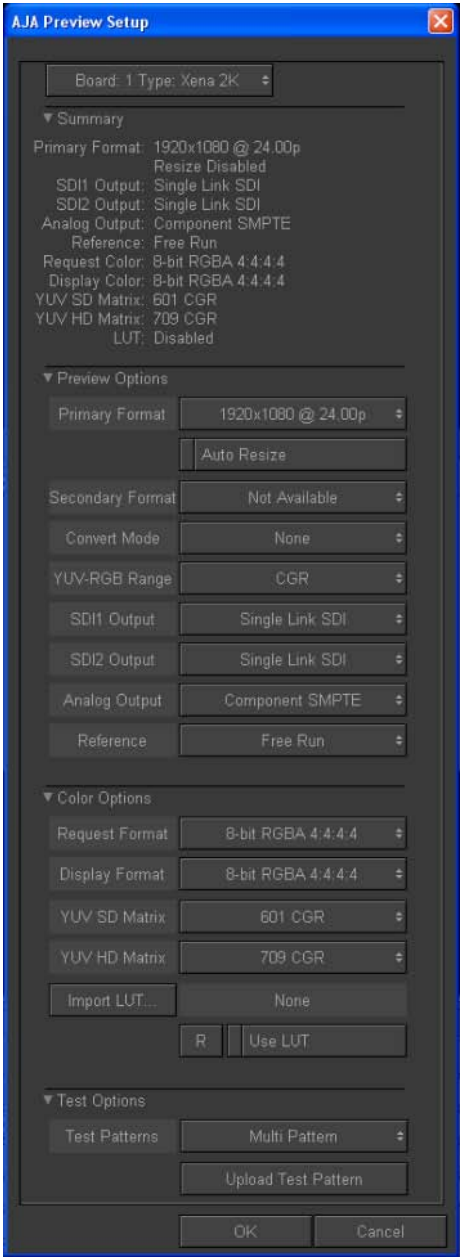
To preview video on a broadcast monitor, you will need the AJA Xena 2K video card and its accompanying driver installed on your system. For more information, refer to the Composite Installation Guide for Windows.

To set up video previewing:

1. Open the User Preferences window by doing one of the following:
   - From the menu bar, select Edit > User Preferences.
   - In the taskbar, click your user name.
2. In the User Preferences window, select the Video Preview tab.



3. For the device, select AJA Preview and click Show Settings.


AJA Preview Setup
Board: 1 Type: Xena 2K
Summary
Primary Format: 1920x1080 @ 24.00p
Resize Disabled
SDI1 Output: Single Link SDI
SDI2 Output: Single Link SDI
Analog Output: Component SMPTE
Reference: Free Run
Request Color: 8-bit RGBA 4:4:4:4
Display Color: 8-bit RGBA 4:4:4:4
YUV SD Matrix: 601 CGR
YUV HD Matrix: 709 CGR
LUT: Disabled
Preview Options
Primary Format
1920x1080 @ 24.00p
Auto Resize
Secondary Format
Not Available
Convert Mode
None
YUV-RGB Range
CGR
SDI1 Output
Single Link SDI
SDI2 Output
Single Link SDI
Analog Output
Component SMPTE
Reference
Free Run
Color Options
Request Format
8-bit RGBA 4:4:4:4
Display Format
8-bit RGBA 4:4:4:4
YUV SD Matrix
601 CGR
YUV HD Matrix
709 CGR
Import LUT...
None
R
Use LUT
Test Options
Test Patterns
Multi Pattern
Upload Test Pattern
OK
Cancel

4 In the AJA Preview Setup window, select a primary format. This should match the resolution and update rate of the material you want to preview. If the resolution does not match, the image will be centered in the video preview device. If the update rate does not match, the video preview device will not be updated with a new image at every video refresh.

5 In the Preview Options section, deselect the Auto Resize option.

6 Set any other parameters you would like and click OK. Refer to the AJA documentation for information on the preview options and color options.

7 In the User Preferences window, click Show Settings if you want to edit the settings the next time you select the Video Preview tab.

Previewing Video

When previewing a composition's video on a broadcast monitor, you can work with multiple Players.

NOTE When you zoom out the Player to 0.5, the video image is resized.

To preview video on a broadcast monitor:

1 Open a composition.

2 Create, import, or link to material that is RGB or RGBA.

TIP If the material is 16 or 32 bits, you can convert it to 8 bits by using a display modifier in the Player or by adding a Convert Depth node to the process.

3 Click the Player which contains the video you want to display on the broadcast monitor.

The Player displays the image generated by the tools in your dependency graph.

NOTE Manipulators are not shown in the video preview.

4 In the transport controls, press Play to play the composition.

Creating User Defined Attributes

Create user defined attributes (UDAs) to let you expand the limited number of already defined attributes associated with the different data types. There are three types of UDAs:

- Composition (version)
- Project
- User

Composition (Version) UDAs

Composition UDAs are the most important type of UDA as this allows you to associate data you have in your facility with projects, users and versions of compositions in Composite. This data could be used to find compositions or help determine information about the material referenced in the setup, such as camera setup. Composition UDAs are associated with the versions of compositions, and can have different information in the UDAs.

Project UDAs

Project UDAs allow your facility to store job related information within the Composite workflow. This could be used by an artist to find out who is responsible for a given shot or it could be used to store project information to be used in a Python script.

User UDAs

User UDAs let you store information related to a specific user in Composite. This could be used to send a user an email as part of a Python script or to store job information.

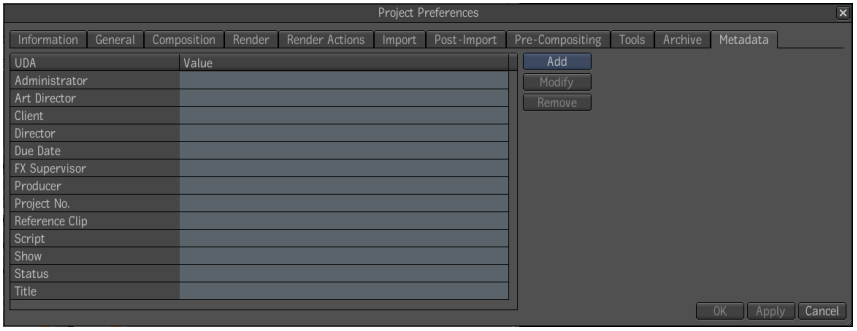
Scoped UDAs

A scoped UDA is a UDA that has an artificial scope defined by the user. For instance, a UDA called Status has no scope whereas a UDA called Project:Status has a scope. These UDAs are just a way for you to keep track of what the UDA applies to; Status applies to all data types while Project:Status should only

apply to the project data type (although there is nothing stopping you from referencing a project UDA from the metadata tab in your project preferences).

Creating UDAs

Creating a scoped UDA for a project, user, or composition can be done in one of two ways: through the metadata tab in the Tool Details, Project Preferences or User Preferences, or with the XML editor. If you're using the XML editor, you must specify a UDA in the XML file associated with the given data type. Each UDA in the XML resource file will be created automatically in the database, if not already present, when the XML file is loaded—see XML Resource Files on page 182.



To create a UDA using the Metadata tab:

➤ ■ From the Tool Details, Project Preferences or User Preferences, select the Metadata tab.
This will reveal a list of user defined attributes. You will now be able to add, remove or modify UDAs in your project.

You can add information to each of the UDAs by clicking in the value field next to the UDA.

NOTE The Add, Modify, and Remove buttons are disabled unless you have read/write privileges for the project you are working in. Furthermore, even with read/write privileges, these buttons will only be enabled when a UDA is selected.

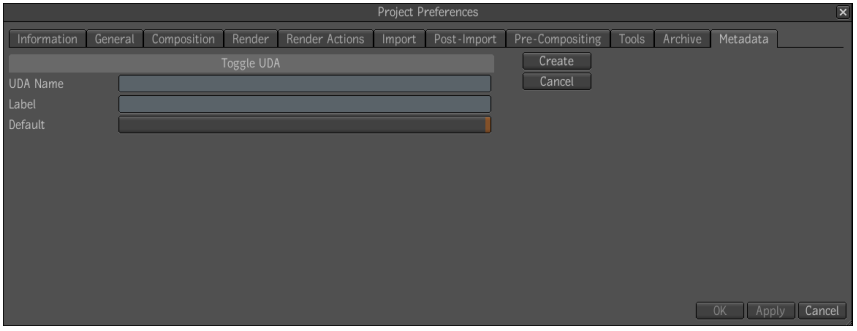
Add button — clicking this button reveals four selections: Toggle, Menu, Numerical and Text.

Modify button — clicking this button modifies the information in the UDA field.

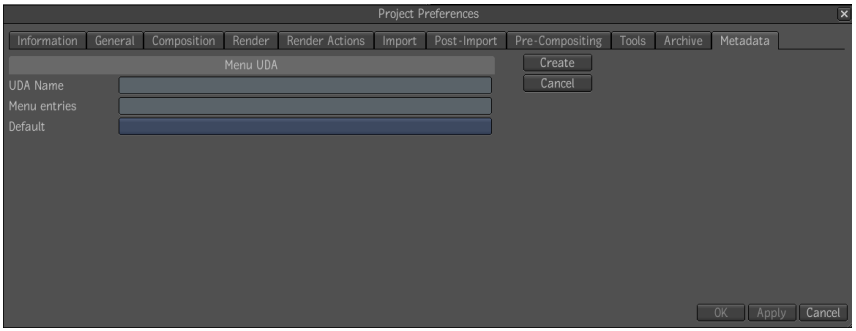
Remove button — clicking this button disables the selected field and removes it from the list of UDAs, but not from the Metadata tab. If you remove a pre-created UDA, the control associated with that UDA in the Metadata tab will be disabled.

■ **Add button**

Toggle: this control allow you to toggle between two states.

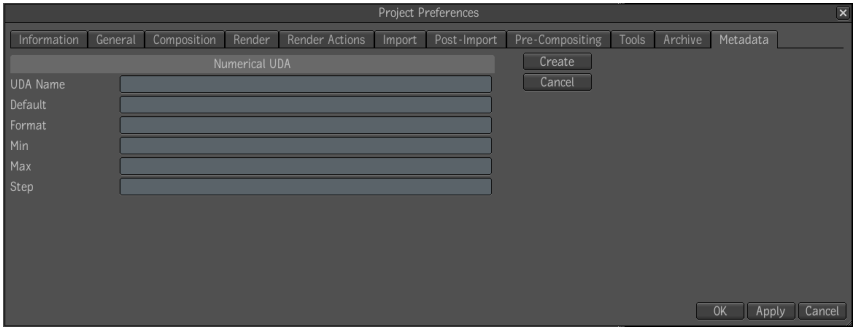


■ Menu: this control allows you to choose between multiple states. You can specify the valid values by listing items separated by a comma. You can also decide which item will be the default by clicking Default.

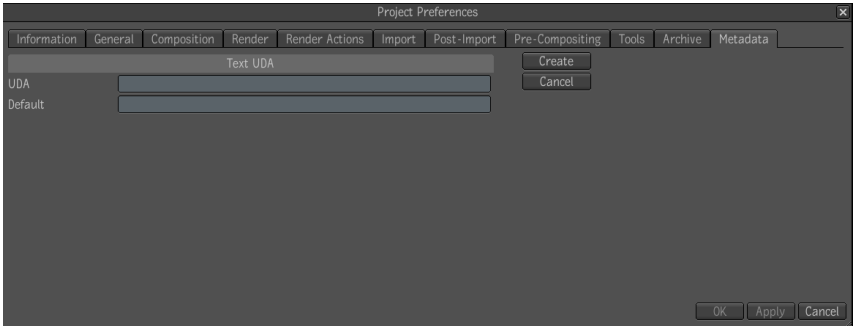


- Numerical: this control allows you to define a numerical UDA.

NOTE It is not necessary to fill out all the fields, only the ones that apply to your project, with the exception of the name field, which must be filled.



- Text: this is a text field for data entry.

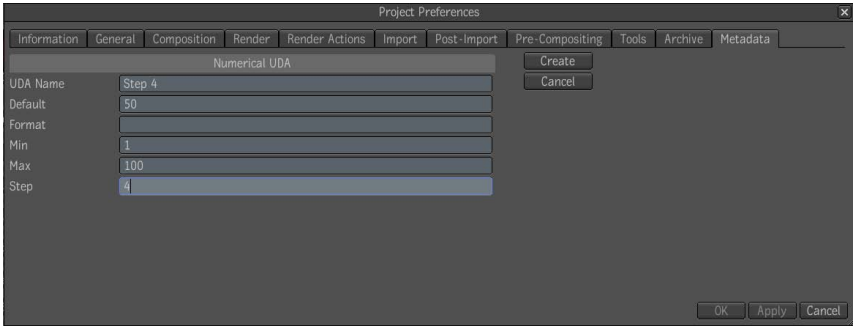


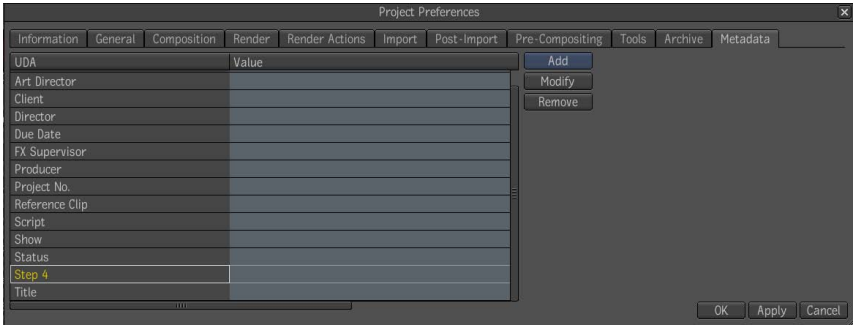
Create — implements the changes.

Cancel — cancels the changes and brings you back to the list of UDAs.

Once you have completed your changes, they will be visible in the original list of UDAs.

NOTE The metadata tab will not reflect the changes made in the manage mode. If you delete or rename the UDA that appears in the metadata tab, the associated control will be disabled.





Displaying a UDA

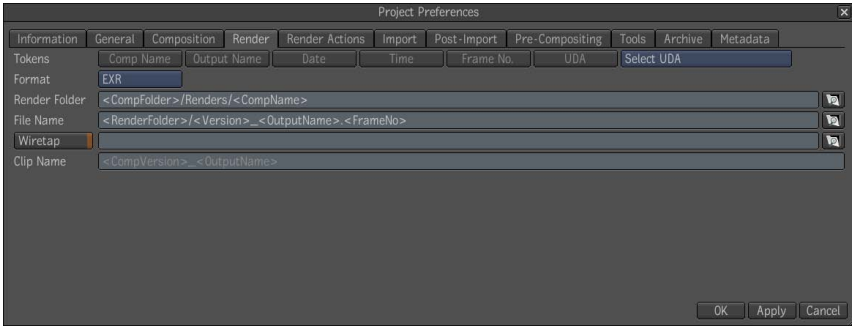
Since it is possible to have more than one data type visible in the browser, UDAs that have the form Version:..., Project:..., User:... will not be displayed by default as individual columns in the browser. If the UDA is created by use of the XML resource file, it will be shown in the Metadata tab in the detail view area of the tool UI. The UDA can be displayed by right-clicking on the title bar and selecting the UDA if desired, but the preferred method of displaying and editing scoped UDAs are through the Metadata tab.

Using a UDA

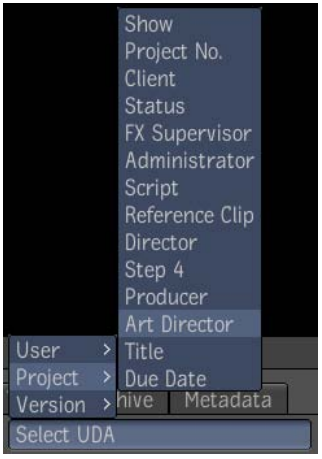
When specifying, for instance, a render file name, there is an option for entering UDAs in the string. The UDAs may be scoped or not and can be selected from the default lists available or can be created by the user.
See—Creating UDAs on page 177.

To add a UDA to a render file name:

1 Open the Project Preferences window by clicking on the project icon in the task bar then select the Render tab.

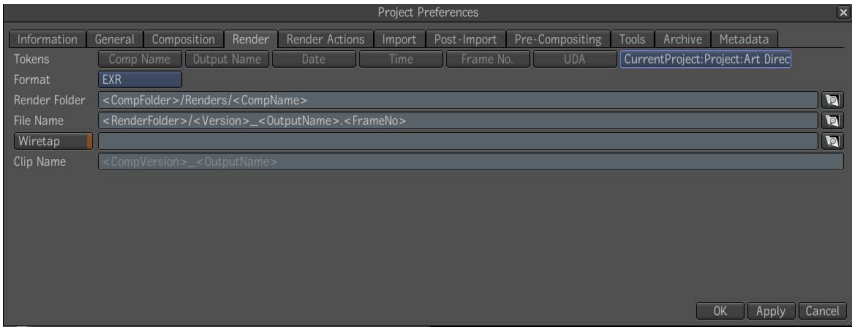


2 Click open the Select UDA menu and select the UDA you want to add to the File Name string. In this example the scoped Project:Art Director UDA is selected, loaded and ready to insert into the string.



3 Place the cursor in the File Name string where you want the UDA to appear and select UDA.

The UDA is added to the string.



4 Repeat steps 2 and 3 to add more UDAs to the string.

XML Resource Files

Composite is shipped with a default set of XML resources files, one for each of the above mentioned data types; metadata.composition.xml, metadata.project.xml, etc. These XML files can be found in the resTables subfolder of the resources folder of the Composite application folder (udas.xml file contains the UDAs declaration; the metadata.*.xml file contains the UI metadata tab UI).When Composite is launched, the udas.xml file is parsed and any UDA that does not already exist will be created.

These resource files are only used to create the layout to display and edit the UDAs within the application. If you wish to modify these files and share them with other workstations running Composite, there is a site preference called Shared Settings that indicates the path to these XML files. This path, or more specifically, directory, can be located on the network so that each workstation uses the same resource file for displaying and creating UDAs. If the files exist in this directory, they will be used, otherwise the resources files in the default location will be used.

Shown below are examples of how to create a text field, value editor, menu, and toggle button. Note the <Uda> name. To make the UDA unique, it should be prefixed with the data type.

UdaTextField

```
<Label>
   <Widget Name="studioLabel">
      <Layout>
         <Constraint>
            <X1 Value="0" Type="u" Target="l"/>
            <Y1 Value="4" Type="u" Target="h"/>
            <X2 Value="14" Type="u" Target="w"/>
            <Y2 Value="-4" Type="u" Target="t"/>
         </Constraint>
      </Layout>
      <LabelString>
         <Text>Studio</Text>
      </LabelString>
      <LabelAlign>
         <Align H="Left" V="Center"/>
      </LabelAlign>
   </Widget>
</Label>
<UdaTextField>
   <Uda>Composition:Studio</Uda>
   <Widget Name="studio">
      <Layout>
         <Constraint>
            <X1 Value="0" Type="u" Target="s" Sibling="studioLabel"/>
            <Y1 Value="4" Type="u" Target="h"/>
            <X2 Value="40" Type="u" Target="w"/>
            <Y2 Value="-4" Type="u" Target="t"/>
         </Constraint>
      </Layout>
   </Widget>
</UdaTextField>
```

UdaValueEditor

```
<UdaValueEditor>
    <Uda>Composition:Shot Number</Uda>
    <Widget Name="number">
       <Layout>
          <Constraint>
             <X1 Value="0" Type="u" Target="s" Sibling="numberLabel"/>
             <Y1 Value="4" Type="u" Target="h"/>
             <X2 Value="15" Type="u" Target="w"/>
             <Y2 Value="-4" Type="u" Target="t"/>
          </Constraint>
       </Layout>
       <Limits>
          <MinMax Min="0.1" Max="1000" Step="0.1"/>
       </Limits>
       <FormatString>
          <Text>%.3f</Text>
       </FormatString>
    </Widget>
 </UdaValueEditor>
```

UdaMenu

```
<UdaMenu>
    <Uda>Composition:Set</Uda>
    <Widget Name="set">
       <Layout>
          <Constraint>
             <X1 Value="0" Type="u" Target="s" Sibling="setLabel"/>
             <Y1 Value="4" Type="u" Target="h"/>
             <X2 Value="40" Type="u" Target="w"/>
             <Y2 Value="-16" Type="u" Target="t"/>
          </Constraint>
       </Layout>
       <MenuString>
          <Text>Interior - Early Morning</Text>
          <Text>Interior - Morning</Text>
          <Text>Interior - Afternoon</Text>
          <Text>Interior - Late Afternoon</Text>
          <Text>Interior - Early Evening</Text>
          <Text>Interior - Evening</Text>
          <Text>Interior - Night</Text>
       </MenuString>
    </Widget>
 </UdaMenu>
```

UdaPushedToggle

```
<UdaPushedToggle>
    <Uda>Composition:HDR</Uda>
    <Widget Name="hdr">
       <Layout>
          <Constraint>
             <X1 Value="0" Type="u" Target="s" Sibling="statusLabel"/>
             <Y1 Value="4" Type="u" Target="h"/>
             <X2 Value="15" Type="uu" Target="w"/>
             <Y2 Value="-20" Type="u" Target="t"/>
          </Constraint>
       </Layout>
       <LabelString>
          <Text>HDR</Text>
       </LabelString>
    </Widget>
 </UdaPushedToggle>
```

UDA Declaration

udas declaration is as follow:

```
<Udas>
  <Uda>
    <Scope>Project</Scope>
    <Name>Administrator</Name>
    <Type>UdaTextField</Type>
  </Uda>

  <Uda>
    <Scope>Version</Scope>
    <Name>F-Stop</Name>
    <Type>UdaValueEditor</Type>
    <Limits>
      <MinMax Min="1" Max="64" Step="0.1"/>
    </Limits>
    <FormatString>
      <Text>%.1f</Text>
    </FormatString>
  </Uda>

  <Uda>
    <Scope>Version</Scope>
    <Name>HDR</Name>
    <Type>UdaPushedToggle</Type>
    <DefaultValue>1</DefaultValue>
    <LabelString>
      <Text>HDR</Text>
    </LabelString>
  </Uda>

  <Uda>
    <Scope>Version</Scope>
    <Name>Set</Name>
    <Type>UdaMenu</Type>
    <DefaultValue>Interior - Early Evening</DefaultValue>
    <MenuString>
      <Text>Interior - Early Morning</Text>
      <Text>Interior - Morning</Text>
      <Text>Interior - Afternoon</Text>
      <Text>Interior - Late Afternoon</Text>
      <Text>Interior - Early Evening</Text>
      <Text>Interior - Evening</Text>
      <Text>Interior - Night</Text>
    </MenuString>
  </Uda>

</Udas>
```

Default UDA Tag

An optional XML tag in the resource file is used to specify the default value for each of the UDAs when a data type that has that UDA is created. If no default tag is specified, the UDA will be empty. The XML tag is <DefaultValue>.

```
<UdaTextField>
    <Uda>Composition:Studio</Uda>
    <DefaultValue>Image Works</DefaultValue>
    <Read-Only/>
    <Widget Name="studio">
       <Layout>
          <Constraint>
             <X1 Value="0" Type="u" Target="s" Sibling="studioLabel"/>
             <Y1 Value="4" Type="u" Target="h"/>
             <X2 Value="40" Type="u" Target="w"/>
             <Y2 Value="-4" Type="u" Target="t"/>
          </Constraint>
       </Layout>
    </Widget>
 </UdaTextField>
```

Read-Only UDA Tag

An optional XML tag in the metadata.*.xml resource file is used to specify whether the UDA is read-only. If no read-only tag is specified, the UDA will be read-write. The XML tag is <Read-Only>. For example <Read-Only></Read-Only> or just plain <Read-Only/>. Don't forget the '/'. See the example above. The read-only tag is only for the UI element so that the user cannot modify its value.

Other Workspace Functions

There are additional functions available in the workspace, including hotkeys and zooming and panning a view.

Composite Hotkeys

You can perform many operations using hotkeys—see Hotkeys on page 817 for a complete list.

The Player

10

Topics in this chapter:

About the Player

As stated earlier, the Player is used to play or preview a composition, or preview media. There are three different versions of the Player: the Player View, the Mini-Player and the Fullscreen Player view. For more information on each of these views, please see The Player on page 34 section in the Interface Fundamentals chapter.

Player View Default Settings

When you add a Player view to a viewport, it has the following settings by default. You can adjust all of these, except for the repeat mode and f-stop, through the Player options. You adjust the repeat mode in the Player controls—see Using the Player Controls on page 36 in the Interface Fundamentals chapter.

- Displays the output of the composition.
- Follows master time.
- Plays all frames and updates at each frame.
- Sets the f-stop to 0.00.
- Plays at the native rate of the composition.
- Does not display any guides apart from the frame outline, or any masks.
- Displays RGB.
- Has repeat mode set to No Repeat.
- Does not correct pixel aspect ratio.
- Does not use a display modifier (LUT).

Calculating Image Size in Composite

All Composite images live on a 2D plane.Composite and PXL use a Cartesian coordinate system to identify points on the plane.This plane and reference system are called the Image Reference Frame (IRF). This reference system has an origin and perpendicular X and Y axes. Units along the X and Y axes have the same length.By default, Composite images are centered at the origin of the IRF.In Composite, pixels at the highest resolution have a height of 1 IRF unit, and have a width equal to their pixel aspect ratio, i.e. 1 IRF unit for 1080P HD material, 0.9 for NTSC material, and 16/15 (1.0667) for PAL material.

For images with non unit pixel aspect ratios, the height of a pixel is 1, but the width of a pixel is equal to its aspect ratio (e.g. 0.9 for NTSC). Using IRF units simplifies image manipulation: for example, translating an image by 10 IRF units will move the image by the same amount, regardless of the direction of the translation, horizontal, vertical, or any combination thereof.

Setting the Target

The target for the Player view is what it displays. You set the target for an individual view in the Player options. The target can be the output node of the dependency graph, the input or output of a tool node, a specific rendered output of the composition, or a context point (the output of a specific tool node)—see Setting the Target for the Player on page 194.

You can specify the channels of the target you want the Player view to display (RGBA, RGB, A, or any other combination of channels)—see Setting the Channels for the Target on page 195.

You can set the resolution you want a Player view to use for its target (full, half, quarter, or proxy). You can use this to improve playback performance when full resolution playback is not necessary—see Setting the Resolution for a Player on page 201 and Playback Performance on page 192.

You can set the aspect ratio of the pixels the Player uses for the target, to either square or non-square pixels—see Setting the Pixel Aspect Ratio of the Player on page 206.

Setting Context Points

A context point is a specific tool node you mark as the target for a Player view. No matter which tool node you select, the view always displays that context point.

Context points are especially useful when you want to see how changes to a tool node that precedes the node with context points affect the context point. You can have the Player view display the output from a context point. You can set a maximum of four context points.

You set a context point in the Schematic view—see Working with Schematic on page 50. You use Player options to set the target for a Player view to a context point—see Setting the Target for the Player on page 194.

Playing Back in Multiple Views

You can have any combination of workspaces including the Player, Schematic, and Animation Editor views, the three views affected by the Player controls. You can use the Player controls to play the composition in only the view currently in focus, or all the views simultaneously.

Master time is an abstract time you can use to synchronize playback among two or more Player, Schematic, and Animation Editor views. You synchronize playback between two views by setting both to follow master time. You can also set an offset for each view, so you can see different parts of the composition playing at the same time. For example, if you are creating a mask, you might have two Players, one in which to create the mask, and one to preview what's ahead, so you can take changes into account as you create the mask.

Master time always starts at frame 0, and has a duration equal to the length of the composition. If a composition starts at frame 215 and ends at 564, master time starts at frame 0 and ends at frame 349.

Standalone time is the opposite of master time. In standalone time, the view responds to the Player controls only if it currently has the focus; if another view has focus at that time, the view set to standalone time remains static. You can also set an offset for standalone time; in this case the offset is relative to the composition time.

You set a view to use master or standalone time using the Follow Master preference in the Playback tab of the view options. By default the view is set to use master time—see Synchronizing or Separating Playback between Views on page 199.

Playback Performance

The ability to play compositions and media in real time depends on both hardware configuration and software optimizations.

Internally, the two key features used to improve playback performance are tiling and the media cache. The media cache keeps images that have been loaded close at hand on the local drive for faster retrieval.

To improve playback performance, you can turn on hardware rendering (if you are compositing with Reaction) and/or define a region of interest. Each of these has trade-offs (usually between quality and speed) and therefore may or may not be feasible in a given situation.

Tiling

Images are automatically divided into tiles and processed. Instead of processing an entire frame, only those tiles that are part of the result of the operation need processing.

Each tile is 512 by 512 pixels. The number of tiles loaded in the Player at any point depends on the resolution of the image, the zoom level of the Player, and the region of interest, if one is defined—see Region of Interest on page 194:

- In general, a zoom level that displays only a detail of the complete image requires fewer tiles than a zoom level that displays the complete image. For example, if you zoom in so that the Player displays only the top-left corner of the frame, it loads only the tiles needed to display that area of the frame. Note that the resolution optimization performed during zooms may have an influence on the number of tiles—see Resolution on page 193.
- A region of interest that contains only part of the image, requires fewer tiles than a larger one or than the complete image. Only the tiles that are part of the region of interest are loaded. You can set the Player to show or hide the area of the loaded tiles that falls outside the region of interest—see Showing or Hiding Tiles on page 204.

Resolution

You can set the resolution that a Player view or the fullscreen Player uses—see Setting the Resolution for a Player on page 201. When full resolution is not required, working at less than full resolution can improve playback performance.

When you change the resolution of a Player, a new copy of the image is retrieved and added to the media cache at the new resolution and tiled for that resolution.

NOTE When you zoom out in the Player, the resolution is automatically adjusted to the optimal one for that zoom level. This reduces the load on the computer.

Hardware Rendering

You can switch to hardware rendering when using Reaction to improve playback performance when the quality of the output is secondary to the speed of playback—see Turning Hardware Rendering On or Off on page 205.

NOTE Only Reaction nodes benefit from hardware rendering. For all other tool nodes, switching to hardware rendering has no effect.

Region of Interest

You can define a subsection of the frame processed for playback by defining a region of interest. This can improve playback by limiting the area of the frame that needs to be processed for the playback.

The region of interest has no effect on rendering an output. When you render an output, the entire frame is always rendered. The Region of Definition (ROD) is used to determine the area to render when you render an output.

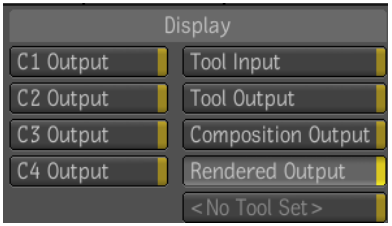
Setting the Target for the Player

You can set the target for a Player view or the fullscreen Player, as well as set the channels of the target that the Player view or fullscreen Player displays—see Setting the Channels for the Target on page 195.

NOTE You can set the target and channels for the target in the fullscreen Player only if you entered it through a Player view.

To set the target for a Player:

1. Do one of the following to display the Player options:
   - In the Player view or fullscreen Player, press the tilde key (~) or middle-click and go through the south gate.
   - In the Player view or fullscreen Player, right-click to display the Player menu.
2. In the Player options, select the Display tab, or choose Display mode in the menu.



3 Select the target you want the Player view to display.

Select:	To display:
C1 Output	The output of the node on which you have set context point 1.
C2 Output	The output of the node on which you have set context point 2.
C3 Output	The output of the node on which you have set context point 3.
C4 Output	The output of the node on which you have set context point 4.
Tool Input	The input of the tool node currently selected in the Schematic view. When you select a tool node in the Schematic view, the Player updates to display what that node receives as input.
Tool Output	The output of the tool node. When you select a tool node in the Schematic view, the Player updates to display the output of that node.
Composition	The composition as it appears in the final node of the dependency graph (the Output node in the Schematic view).
Rendered Output	One of the rendered outputs of the composition. You use the Results tab of the Tool UI to select which rendered output the Player displays.
Tool Set	Sets the player in focus to view the output of the tool for which the mask is being edited.

4 To exit the Player options, move the cursor to another area.

Setting the Channels for the Target

You can specify the channels you want the Player view or fullscreen Player to display for its target.

NOTE You can set the channels for the target in the fullscreen Player only if you entered it through a Player view.

To set the channels for the target:

1. Do one of the following to display the Player options:
   - In the Player view or fullscreen Player, press the tilde key (~) or middle-click and go through the south gate.
   - In the Player view or fullscreen Player, right-click to display the Player menu.

2. In the Player options, select the Display tab or select View Mode in the menu.

3. Set the channels for the target.

Select:	To display:
Chns: RGB	All channels (RGBA, RGB, or A) contained in the target. For example, if the target is the output of an RGBA composition, it displays RGBA channels. If the target is the output of an RGB composition, it displays RGB channels.
RGB	The red, green, and blue channels.
RGBA	The red, green, blue, and alpha channels.
A	The alpha channel.
<user-defined>	Any combination of RGBA channels. Enter the combination you want as a four-character string in which you specify each channel either as the channel letter (if you want to display it), or as a blank (if you do not want to display it). Position is significant: the first position is R, the second G, the third B, and the fourth A. For example if you wanted to see only the blue and alpha channels, you would enter two spaces or 2 'Ns' followed by BA.

Setting the Mark In and Mark Out Points for Playback

You can set the mark in and mark out points that the Player views use to play back a composition. By default the Player view uses the mark in and out points set for the composition, or, if none are set, uses 0 as mark in and 1 as mark out.

To set the mark in and mark out points:

1. Open the composition for which you want to set mark in and mark out points.
2. In the Tool UI, select the Composition tab.

3. Edit the Mark In and Mark Out fields.

   The Current Frame field in the Player controls updates to reflect the changes you make to the fields. The area between the mark in and mark out points (the playback area) appears in light gray.

Setting the Repeat Mode

You set the repeat mode in the Player controls—see Using the Player Controls on page 36.

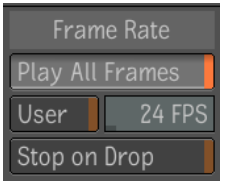
Setting the Frame Rate for Playback

You can set the frame rate that the Player view or fullscreen Player uses for playback.

NOTE You can set the frame rate for the fullscreen Player only if you entered it through a Player view.

To set the frame rate for playback:

1. In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.
2. In the Player options, select the Playback tab and locate the Frame Rate area.



3. Set the playback frame rate for the Player view by deselecting the User button and specifying a frame rate in the field beside the button. Select the User button to use the frame rate set for the composition.

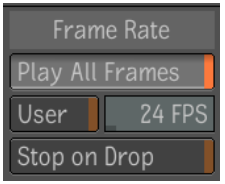
Setting the Real-Time Playback Preference

In a Player view or in the fullscreen Player, you can set a Player preference that determines whether real-time playback has priority over a playback that includes every frame or vice-versa. Real-time playback is also influenced by the hardware on which you are running Composite, and the load on that machine at the time you perform the playback.

NOTE You can set the real-time playback preference for the fullscreen Player only if you entered it through a Player view.

To set real-time playback preferences:

1. In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.
2. In the Player options, select the Playback tab and locate the Frame Rate area.



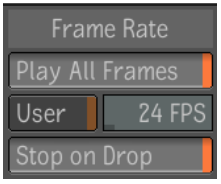
3 Set the real-time playback preference.

Play All Frames Select this button to play every frame, regardless of the impact on real-time playback. Deselect to have the Player view drop frames if necessary to perform playback as close to real-time as possible.

NOTE A busy cursor appears in the lower-left corner of the Player when Composite is processing the composition during playback.

---

4 If you want the Player to stop playing if a tool is dropped into the dependency graph, select the Stop on Drop button.



Synchronizing or Separating Playback between Views

You can set a Player, Schematic, or Animation Editor view to use either master or standalone time. Master time is used to synchronize playback among views—see Playing Back in Multiple Views on page 192.

To set a view to use master or standalone time:

1 In the Player view or fullscreen Player, go through the south gate.

2 In the view options, select the Playback tab and locate the Viewer Time area.

3 Set the time the view uses.

Follow Master Select this button to use master time. Deselect to use standalone time.

Offset Select this button to set an offset, in frames, for the time. Type the offset in the field to the right of this button, or click and drag on the bottom edge of the field to adjust the value. The frame offset is with respect to the time you set for the view. For example, if you set a frame offset of 5 and selected Follow Master, the view always displays the frame that is five frames ahead of the frame currently running in master time. If you deselected Follow Master, the view always displays the frame that is five frames ahead of the frame running in the standalone time of the view.

Setting the Playback Point when the Player Updates

You can set the point when a Player view or the fullscreen Player updates its display.

NOTE You can set the playback point when the fullscreen Player updates only if you entered it through a Player view.

To set the playback point when the Player updates its display:

1. In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.
2. In the Player options, select the Playback tab and locate the Viewer Time area.

3. Set the point when the Player updates its display.

   Update on Play Select this button to update the image at each frame when you play the composition. Deselect to update the image only when you stop playing the composition, and only at the scrub frames when you scrub through the composition.

Setting the Resolution for a Player

You can set the resolution that a Player view or the fullscreen Player uses. This is useful for improving playback performance when a high resolution image is not necessary.

NOTE You can set the resolution for the fullscreen Player only if you entered it through a Player view.

To set the resolution the Player uses:

1. In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.
2. In the Player options, select the Display tab, and locate the Streams area.

3. Select the resolution you want the Player view to use.

   The Player immediately updates to use the resolution you enable.

Things to Remember

- When you launch the fullscreen Player from a Player view, it uses the resolution set for that Player view. In all other cases it uses full resolution.
- The mini-Player always uses proxy resolution.
- When you zoom out in the Player, the resolution is automatically adjusted to the optimal one for that zoom level. This reduces the load on the computer.

Setting the Grid in the Player

See Setting the Player Settings on page 244.

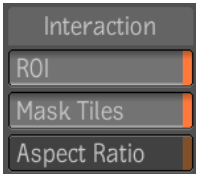
Setting a Region of Interest (ROI) in the Player

You can set a region of interest in a Player view or in the fullscreen Player.

NOTE You can set a region of interest in the fullscreen Player only if you entered it through a Player view.

To set a region of interest in the Player:

1. Do one of the following to display the Player options:
   - In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.
   - In the Player view or fullscreen Player, right-click to display the Player menu.
2. In the Display tab of the Player options, click the ROI button to enable a region of interest. Or, in the Player menu, select ROI.



The Player view updates to display a yellow box with resize handles.

3 Use the resize handles to adjust the region of interest.

You can resize the box to adjust the region of interest at any point. It remains visible as long as ROI is enabled.

Things to Remember

- When you define a region of interest (ROI), you can choose to show or hide the areas of the loaded tiles that fall outside the ROI—see Showing or Hiding Tiles on page 204 and Tiling on page 193.

Displaying Masks and Guides

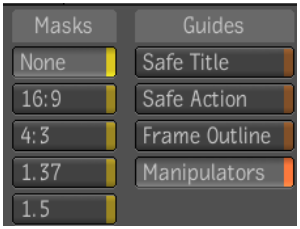
You can use masks to preview the target of the Player view or the fullscreen Player in different film format aspect ratios. You can use guides to show the frame outline, as well as the safe areas within the frame for titles and action.

NOTE You can display masks and guides in the fullscreen Player only if you entered it through a Player view.

To display masks/guides in the Player:

1 In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.

2 In the Display tab of the Player options, locate the Masks and Guides areas.



3 In the Masks area, select the mask you want the Player to display, or select None if you do not want to display a mask. Each mask represents the aspect ratio of a specific film format.

The Player updates immediately to reflect the mask you select.

4 In the Guides area, enable the guides you want the Player to display.

The Player updates immediately to reflect the guides you enable or disable.

Select:	To display the:
Safe Title	Safe title guide. This guide indicates the area of the frame in which it is safe to place titles and graphics (i.e. in which there is no risk of having titles or graphics cropped by the device on which the shot is viewed). The safe title area is inside the safe action area.
Safe Action	Safe action guide. This guide indicates the area of the frame within which there is no risk of having any of the image cropped by the device on which the shot is viewed. The safe action area is larger than, and encloses, the safe title area.
Frame Outline	Frame guide. This guide indicates the edges of the frame.

Showing or Hiding Tiles

When you define a region of interest (ROI), playback performance can be improved by loading only the tiles required to display that ROI. You can choose to show or hide the areas of the loaded tiles that fall outside the ROI.

You can show or hide tiles in a Player view or the fullscreen Player.

NOTE You can show or hide tiles in the fullscreen Player only if you entered it through a Player view.

To show or hide the area of the tiles that fall outside the region of interest:

1. In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.
2. In the Player options, select the Display tab.
3. Under Interaction, select Mask Tiles.

   NOTE Show or hide tiles is only available if you defined a region of interest in the image.

Turning Hardware Rendering On or Off

If you are compositing with Reaction, you can turn hardware rendering on or off for a Player view or the fullscreen Player. For all other tool nodes, switching to hardware rendering has no effect.

NOTE You can turn hardware rendering on or off in the fullscreen Player only if you entered it through a Player view.

To turn hardware rendering on or off in the Player:

1. In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.
2. In the Player options, select the Tools tab and locate the Reaction area.

3. Select Interactivity to use hardware rendering.

Deselect Interactivity to have the view use software rendering. Note that the Reaction tool disregards any display modifier set for the Player when Interactivity is enabled. Recall that you set a display modifier for a Player in the third tab of the Player options—see Applying a LUT or Color Conversion Tool to the Player on page 210.

4 Select View Name to display the name of the Reaction view currently being shown (reference or camera).

Things to Remember

- Hardware rendering is only used with Reaction nodes.

Setting the Pixel Aspect Ratio of the Player

You can set the aspect ratio of the pixels that the Player view or fullscreen Player uses, either square and non-square pixels.

NOTE You can only toggle the pixel aspect ratio in the fullscreen Player if you entered it through a Player view.

To set the pixel aspect ratio in a Player view:

1 In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.

2 In the Player options, select the Display tab.

3 Under Interaction, select Aspect Ratio to use non-square pixels. In this case the Player displays the image using the pixel aspect ratio defined for the composition. You can examine this ratio in the Pixel Aspect Ratio field in the Composition tab of the Tool UI. Deselect Aspect Ratio to use square pixels.

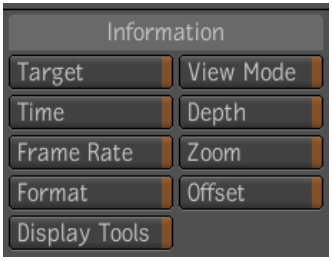
Displaying Player Option Information in the Player view

You can set the Player view or the fullscreen Player to display information in a bar at the bottom of the Player. This is a convenient reminder of the options set for that Player, and can be particularly useful when you using more than one Player view.

NOTE You can display target information in the fullscreen Player only if you entered it through a Player view.

To display information about the target, in the Player:

1. In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.
2. In the Display tab of the Player options, locate the Information area.



3. Set the information you want the Player to display.

Select:	To:
Target	Display the name of the composition followed by a colon and the name of the target set for the Player. Remember, you set the target for the Player view in the Display area of the Display tab of the Player options. Targets are represented as follows: ■ Output indicates that the target is a composition. ■ <toolname> Out indicates the target is a Tool Output. ■ <toolname> In indicates the target is a Tool Input. ■ A context point is represented as the name of the context point, followed by the name of the tool node on which it is set, followed by either In for Input or Out for Output.
Time	Display the position within the composition, of the frame that currently appears in the Player. The position is represented as a frame number.
Frame Rate	Display the frame rate of the target.
Format	Display the resolution and pixel aspect ratio of the target.
Display Tools	Display the name of the tool(s) used in this composition.

Select:	To:
View Mode	Display the channels (RGB, RGBA, A, or another combination of these) that appear in the Player. You set the channels you want the Player to display in the View area of the Player options—see Setting the Target for the Player on page 194.
Depth	Display the bit depth of the target.
Zoom	Display the current zoom level of the Player.
Offset	Display the time offset set for the Player. Recall that you set the time offset in the Playback tab of the Player options—see Synchronizing or Separating Playback between Views on page 199.

Offset Display the time offset set for the Player. Recall that you set the time offset in the Playback tab of the Player options—see Synchronizing or Separating Playback between Views on page 199.

Displaying Manipulators in the Player

You must enable the display of manipulators in the Player view or in the fullscreen Player to see manipulators related to individual tools. Manipulators for individual tools include the eyedropper for the Color Picker, axes in the Reaction tool, and control points in the Garbage Mask tool.

NOTE You can enable the display of manipulators in the fullscreen Player only if you entered it through a Player view.

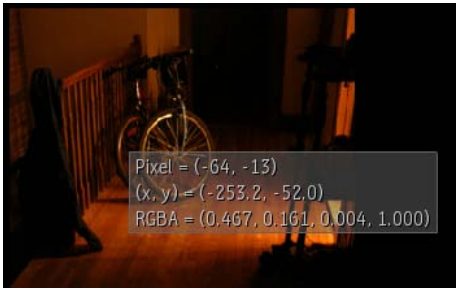
To display manipulators in the Player:

1. In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.
2. In the Display tab of the Player options, select Manipulators (at the bottom of the Guides area).

In-player Pixel Display

To display positional and color value information about a particular pixel in an image:

1. Zoom the Player to view the pixel.
2. Place the cursor in the player over the pixel and press **Z**.



Adjusting the Zoom or Pan

You can zoom or pan in the Player view, the fullscreen Player, or the mini-Player.

To zoom in the Player:

➤ Do one of the following:

- In the Player, press **Up Arrow** to zoom in or **Down Arrow** to zoom out.
- Press **Ctrl** + **Spacebar**(for Windows and Linux) or **Cmd** + **Spacebar**(for Mac OS) and drag right to zoom in or left to zoom out.
- To zoom a region Region zoom; User drag out a box, and the view is zoomed (uniformly or not, depending on the viewer) to fit the viewer extents. You can also use the **Shift** + **Spacebar** + drag hotkey combination.
- Press **Home** to reset the zoom factor to 1. Pressing **Home** again will set zoom to previous zoom factor. This is not available in all viewers.

To pan in the Player:

➤ In the Player, press **Spacebar** and drag.

Things to Remember

- When you zoom out in the Player, the resolution is automatically adjusted to the optimal one for that zoom level. This reduces the load on the computer.

Applying a LUT or Color Conversion Tool to the Player

You can apply any LUT or color conversion in the Tools tab to a Player view or the fullscreen Player. This affects only what that particular Player displays; it has no effect on the composition data. You can also apply a tool preset as a display modifier in the Player—see Using Tool Presets as a Display Modifier on page 148.

NOTE You can apply a LUT or color conversion tool to the fullscreen Player only if you entered it through a Player view.

To apply a LUT or color conversion tool to the Player:

1. In the Player view or fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and swipe east to display the Pick List, Tools, and Views tabs.
2. In the Tools tab, select a tool to apply to the Player. Drag it outside the Tools tab, and do one of the following:
   - Pause over the Player view in which you want to apply the tool.
   - In the fullscreen Player, pause anywhere.
3. In the Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.

   The Player Options appear.
4. Position the cursor over the last tab of the Player Options and release.

   The name of the tab changes to that of the dropped tool. The Player updates to reflect the result of applying the LUT or color conversion tool to the image.

To remove a display modifier from the Player:

1. In the Player, middle-click or press the tilde key (~) to display the Gate UI and go through the south gate.

   The Player Options appear.

2. Select the Display tab.

   In the Tool Options area on the right, click Delete.

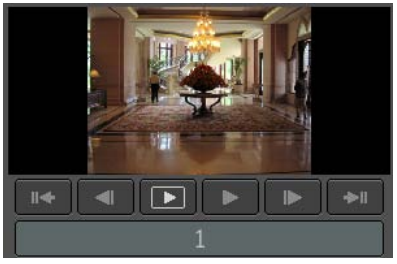
Things to Remember

- You can display the name of the tool you use, in the information bar at the bottom of the Player view—see Displaying Player Option Information in the Player view on page 206.
- The Reaction tool disregards any display modifier set for the Player when the Interactivity option is selected.

The Mini-Player

The mini-Player appears on the left in the Information tab area when you select a media file or composition in the File Browser. The mini-Player lets you preview compositions, as well as media prior to importing. You must have the Information tab selected to see the mini-Player.

The mini-Player contains a limited subset of the Player controls available for Player views. This reflects the context in which you use the mini-Player (to browse compositions in the project folders or preview media prior to import)—see Using the Mini-Player on page 212 for help using the controls.



The mini-Player

Using the Mini-Player

The mini-Player lets you preview media compositions in project folders and lets the project administrator preview media prior to import—see The File Browser on page 41. The mini-Player includes controls to play the composition or media. You can also scrub or move to a specific frame of the composition or media.

To display a composition or piece of media in the mini-Player:

➤ In a File browser, select a composition.

The composition is loaded in the mini-Player.

To play the composition or media in the mini-Player:

➤ Use any of the playback controls.

Click:	To:
	Play the composition forward.
	Display the fullscreen Player.
	Step backward one frame.
	Step forward one frame.
	Move to mark in frame.
	Move to mark out frame.

To scrub through the composition or media in the mini-Player:

➤ Click and drag in the Current Frame field.

(a) Current Frame field

To move to a specific frame of the composition or media:

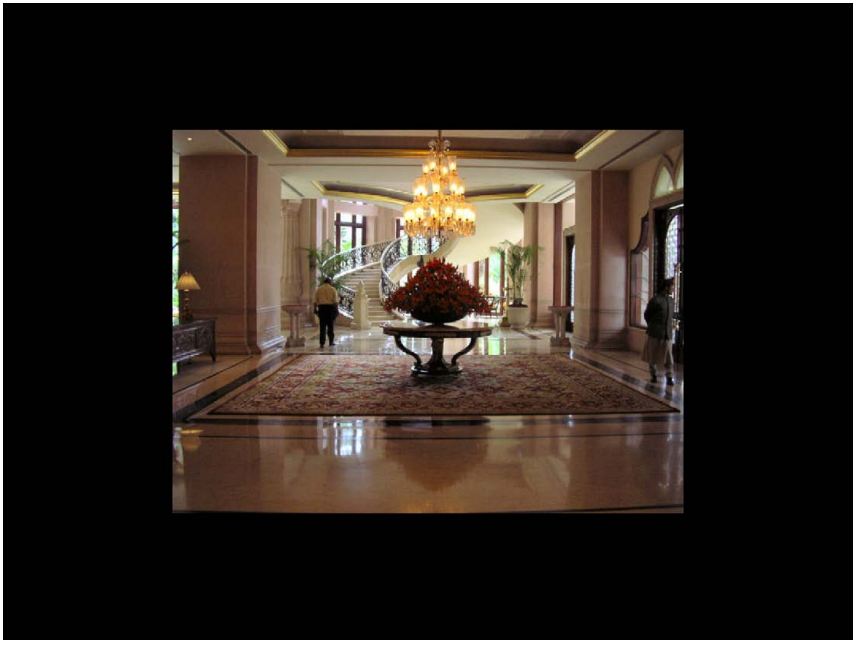
➤ Click in the Current Frame field, enter the frame number to which you want to move, and press **Enter**.

(a) Current Frame field

The Fullscreen Player

The fullscreen Player occupies the entire screen (the taskbar is also hidden) and is available from any Player view, as well as from the mini-Player.

When you enter the fullscreen Player, it plays the composition or media forward from the frame at which you entered the fullscreen Player.



The fullscreen Player

Fullscreen Player and the Gate UI

When you enter the fullscreen Player through a Player view, the Gate UI is available. You can thus add tools to a composition by going through the east gate, display and work in the Schematic overlay by going through the north gate, set Player options by going through the south gate (or from the context menu), or use the composition browser by going through the west gate.

NOTE When you set Player options in the fullscreen Player, they are saved and applied to the Player view you were in when you entered the fullscreen Player. For example, if you set a region of interest while in the fullscreen Player, when you exit, that region of interest appears in the Player view.

Player Controls in the Fullscreen Player

The fullscreen Player occupies the entire screen and does not, by default, display any Player controls. You can access the Player controls by displaying the Player options from within the fullscreen Player (go through the south

gate). The Player controls appear at the bottom of the Player options. You use them as you would in the Player view—see Using the Player Controls on page 36.

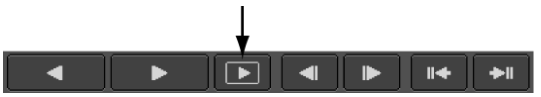
You can also use the Player hotkeys to navigate frames, set in and out marks, set a region of interest, and so on—see Player on page 829.

Using the Fullscreen Player

You can enter the fullscreen Player from either the Player view or the mini-Player. If you enter the fullscreen Player from a Player view, you can add tools to the composition, either directly from the Views tab, or by dropping the view from the Views tab into the Schematic overlay. If you enter the fullscreen Player from the mini-Player, you can only access the Player options from the Gate UI.

To enter the fullscreen Player:

➤ In the playback controls, click the Fullscreen button or press **F**.



To exit the fullscreen Player:

➤ Press **F**.

To control the fullscreen Player:

1. Press **L** to play forward, press **L** again to stop.
2. Press **J** to play backward, press **J** again to stop.

NOTE When you enter fullscreen mode, the Player plays forward by default.

To add a tool to the composition in the fullscreen Player:

1. In the fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and swipe east to display the Pick List, Tools, and Views tabs.
2. In the Tools tab, drag the tool you want to add to the composition, and drop it in the fullscreen Player. A drop gate appears and you can specify

where to add it in the dependency graph (the options that appear depend on the current tool node selected when you perform the drop).

To add a tool to the Schematic overlay in the fullscreen Player:

1. In the fullscreen Player, middle-click or press the tilde key (~) to display the Gate UI and swipe right to display the Pick List, Tools, and Views tabs.
2. In the Tools tab, drag the tool you want to add to the composition outside the Tools tab, but do not release it.
3. Go through the north gate to display the Schematic overlay, and drop the tool in the position in the dependency graph you want it to occupy.

To Navigate Views:

➤ **Press Ctrl + Tab**.

Working with Compositions

11

Topics in this chapter:

Creating Compositions

A composition is a file that contains both the current work and the different versions, if any, of that work. Compositions are stored in your file system.

What's in a Composition File?

A composition is stored as a file or a group of files, depending on its state. For a newly created composition, a file named *<compname>.txcomposition* will exist on the file system. Within Composite, you will see only the name part of the file, not the extension. Whenever a new version is created, a new directory named *<compname>.txversions* is created and contains additional files, one for each version. Also, if you happen to have some Raster Paint nodes in your

composition, another directory, named *<compname>.txpaintstrokes* is created to store the paint strokes. By default, the browsers in Composite do not show the versions and paint strokes directories. To see them, you must deselect the Collapse Compositions option.

Creating a New Composition

You can create a new composition and have it open automatically.

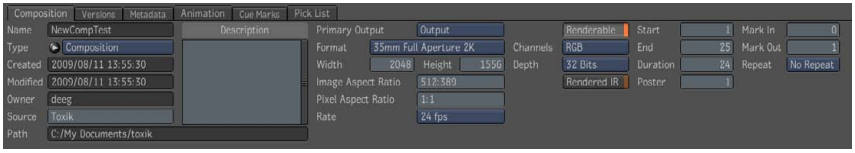
To create a new composition:

1. Do one of the following:
   - Select File > New or press **Ctrl + N** (for Windows and Linux) or **Cmd + N** (for Mac OS). Type in a name for your composition.
   - In a browser, right-click an empty area and select New Composition. Type in a name for your composition. Open your new composition by double-clicking it, or by selecting it and clicking Open.

     An Output node appears in the Schematic view (if you have one visible).

2. In the Composition tab of the tool UI, update the properties for the composition. Some properties are editable.



Name The name of the composition.

Type Designate the composition as footage or as a composition—see Footage Role on page 137.

Created Date the composition was created.

Modified Date the composition was last modified.

Owner User name associated with the composition.

Source Identifies the source application from which the composition media is derived. Usually it is Composite, but if you are using files

generated by another application, such as Maya, the source would be set accordingly. You can edit this field.

Path Location of the composition.

Description Information you can enter about the composition.

Primary Output Select the output to use as the principal output of the composition. The format, width, height, image aspect ratio, pixel aspect ratio, and rate settings are those of the primary output—see Primary Output on page 58.

Format Set the resolution of the composition by specifying a known film or video format. The format you select is linked to the following: width, height, image and pixel ratio, and rate. Select Custom to define a non-standard format.

Width, Height Width and height (in pixels). These values, together with ratios and rate, may correspond to a predefined format, in which case the format is automatically selected. Otherwise, the format selected is Custom.

Image Aspect Ratio and Pixel Aspect Ratio The ratio is expressed as a fraction representing the proportion, width:height, of the image. This value, together with width, height, pixel ratio and rate, may correspond to a predefined format, in which case the format is automatically selected. Otherwise, the format selected is Custom. Image and pixel ratio are linked; changing one updates the other.

Rate Frame rate expressed as frames per second (fps). For a given format, a limited number of rates are available. To set a non-standard rate for a given format, change the format to Custom.

Channels Set the image channel for the primary output: RGB, RGBA, A.

Depth Select the bit depth (8, 16, or 32 bits) for the primary output.

Start, End, Duration Set the start, end, and duration of the primary output. End is exclusive, so Duration = End - Start. Example Start=0, End=30, Duration=30. Start=1, End=31,Duration=30. And so on.

Poster Frame used for proxies, defaults to start value.

Mark In/Mark Out Set the in point and out point for the primary output.

Repeat Select the default repeat mode (Hold, Loop, Ping-Pong, No Repeat) for primary output when you play it.

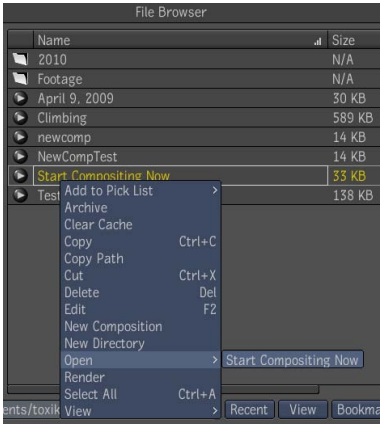
Opening and Viewing Compositions

There are two modes in which you can load a composition into Composite: Open and View. The Open mode allows you to modify the composition (read/write mode). The View mode allows you to view the dependency graph and all attributes in the tool UI, but without being able to modify anything (read-only mode). There is a circumstance where a composition can only be viewed, and that is whenever another user has already opened the same composition for modification. This is indicated in the browser by a yellow read-only icon.

(a) Read-only icon

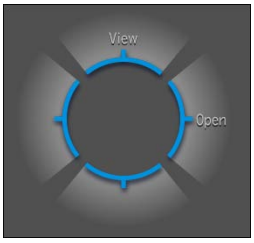
To open a composition:

1. Select File > Open or press **Ctrl + O** (for Windows and Linux) or **Cmd + O** (for Mac OS).
2. In the File Browser, do one of the following:
   - Double-click a composition.
     The composition is opened in Open mode (read/write). If the composition is in use by another user, it will open in View mode (read only).
   - A more advanced technique would be to right-click a composition and select Open. If the composition has more than one version, they are displayed in a list. Select a version to open.

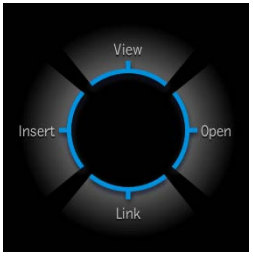


The version of the composition is opened in read/write mode. If the version is in use by another user, it will open in View mode (read only).

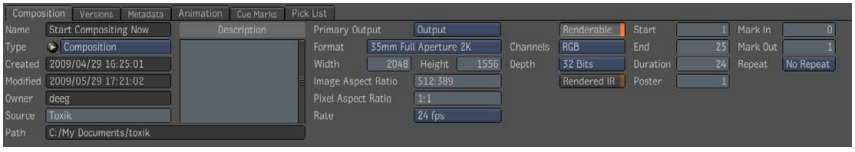
- Select a composition and drag it to the Schematic or Player. In the Gate UI that appears, swipe east to open the composition.
  If the composition is not in use and there is no composition currently open, the following Gate UI is displayed. Swipe through the east gate to open the composition.



If the composition is not in use and there is already an open composition, the following Gate UI is displayed. Swipe through the east gate to open the composition.



The composition opens in Open mode or View mode. In the tool UI, the Composition tab is displayed.



3 The name of the currently open composition is displayed in the taskbar. Click beside the Composition icon to view the list of open compositions.

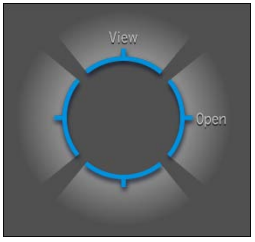
(a) Composition icon

To view a composition:

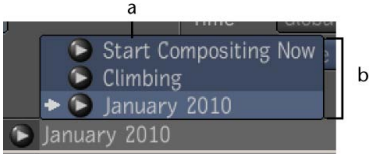
1. Select File > Open or press **Ctrl + O** (for Windows and Linux) or **Cmd + O** (for Mac OS).
2. In the File Browser, do one of the following:
   - Select a composition, then right-click and select View.

     NOTE Advanced technique: if the composition has more than one version, they are displayed in a list.

     You can view any of these versions.
   - Select one or more compositions, and drag them to the Schematic or Player. In the Gate UI that appears, swipe north to view the compositions.



   The name of the composition you are viewing is displayed in the taskbar.
3. Click the composition icon to view the list of opened or viewed compositions.



(a) Composition icon (b) List of opened or viewed compositions

Saving Compositions

To save a composition:

➤ Select File > Save or press **Ctrl** + **S** (for Windows or Linux) or **Cmd** + **S** (for Mac OS).

To save a composition with a different name:

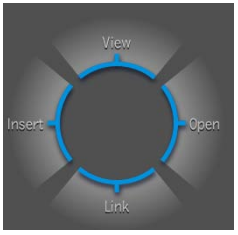
1. Select File > Save As.
2. In the Save Composition As browser that opens, enter a different name for the composition and click Save.

Inserting a Composition

When inserting a composition, you are actually copying its nodes into the currently open composition. The composition being inserted is not affected by the insert operation.

To insert a composition:

1. Select File > Open or press **Ctrl** + **O** (for Windows and Linux) or **Cmd** + **O** (for Mac OS).
2. In the File Browser that opens, select a composition and drag it to the Player or Schematic view.
3. In the Gate UI that appears, swipe west to insert the selected composition in your current composition. All nodes from the selected composition are inserted, except its output nodes.



Checking the Status of Compositions

You can check whether a composition is open or whether it is view-only.

To check the status of compositions:

1. In the taskbar, click beside the composition icon to display the list of compositions.
2. Look at the version icon for each item in the list. A yellow icon indicates a view-only composition. Icons that are not colored indicate compositions you have open for modification.

Rendering, Caching and Versioning

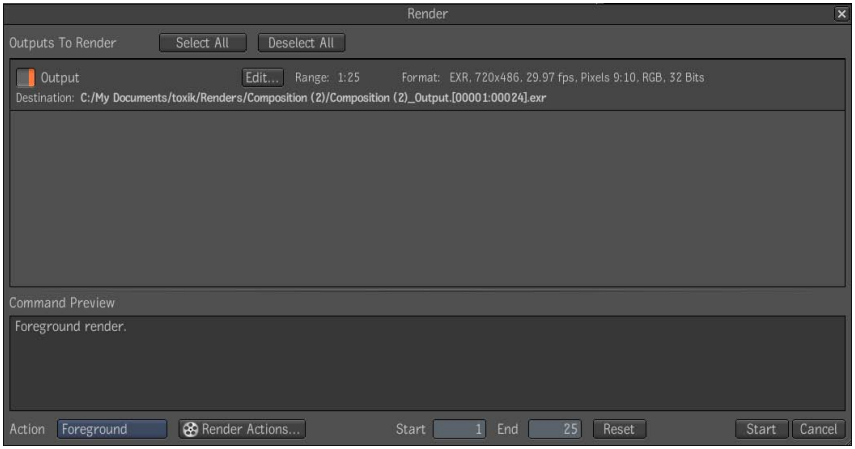
The following is a description of rendering, caching and versioning techniques.

Rendering a Composition

You can render a composition at any time, to save the final output of a composition. When you render a composition, you choose the output(s) you wish to render, and the way in which they will be rendered, which is called the render action, as either the foreground or one of the many background actions. A progress bar displays the rendering of frames for all outputs. The outputs are rendered in parallel, one frame index after another alternating between each output. This can speed up the rendering process because some intermediate results are reused.

To render a composition:

1. Go to the File menu and click Render or click Alt+R. The Render dialog will display.

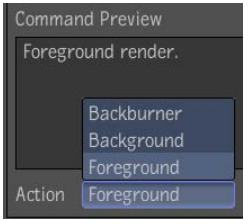


2 Click the Select All or Deslect All button to enable or disable rendering for all outputs.

3 Click the Output button to enable or disable the render for an individual output.

4 Click the Edit button to map the UI for this node. This will dismiss the Render dialog.

5 Click the Action button to select the way in which you want to render your composition. By default, Composite has three render actions, to the foreground, background or backburner.



Foreground will perform the render while blocking further interaction with the Composite UI. It will only use the local machine, but is the fastest single-machine rendering option. Its disadvantage is that you cannot use Composite while rendering proceeds.

Background will also perform the render on the local machine, but allows you to continue using Composite during rendering. It is slower than Foreground rendering.

Backburner dispatches rendering to Autodesk Backburner(TM), Autodesk's distributed network rendering solution. As for Background rendering, you can use Composite while rendering proceeds. When you choose Background or Backburner rendering, the render is done on a copy of the composition: modifying the composition while it renders will not affect the result of the rendering.

6 Click the Render Actions button to dismiss the Render dialog, and display the Render Actions tab in the Project Preferences dialog. This allows you to edit the existing render actions, or define new ones.

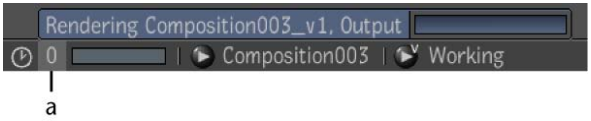
7 Set the Start and End values to set the start and end frames for this render.

8 Click the Reset button if you wish to modify the Start and End frames to include frames of all the currently selected outputs.

9 Finally, click the Start button to begin rendering, or click the Cancel button.

To display the list of background tasks:

➤ In the taskbar, click the number representing the number of ongoing render tasks.



(a) Number of ongoing render tasks

Caching

Composite can keep track of previously-computed images through an image cache, called the media cache. While building your composition, you will frequently connect new nodes to the output of existing nodes. It is beneficial for performance to avoid re-computing the image results of existing nodes, and rather read previously-computed results from a cache.

Composite supports this through its media cache, which features two caching levels: an automatically managed in-memory cache, which is limited by the size of available memory, and a disk-based cache that you can turn on per node.

Disk cached results are called intermediate results. It is counter-productive to enable intermediate results at every node in your composition: the increased disk usage in reading and writing to the disk cache will negate the gains from caching. Instead, try to identify the subgraph parts of your composition that are costly to compute and are heavily used, and cache only the outputs of those subgraphs.

For example, if you have nine subgraphs connected to nine different image sources of a single Reaction node, you can cache the outputs of those nine subgraphs. In this way, working in Reaction will only involve reading disk cached results for the nine subgraphs, rather than re-computing each one of them at potentially very high cost.

Intermediate results are the results rendered at the output of any node, other than the output nodes, in the dependency graph. When you set a node to create intermediate results, the frames displayed in the Player when you preview the results are saved to your local media cache. These frames do not need to be reprocessed as you continue adding tool nodes and viewing the results.

For example, you can add a glow tool to your composition and set it to create intermediate results. As you play the composition (or scrub through the frames), the glow is displayed as it processes. At the same time, the result is stored in your local media cache. If you add another node after the glow, Composite will not reprocess the frame.

Intermediate results are stored in the media cache as long as there is room. When the media cache is full, Composite will start discarding the images that are least recently used. You should consider this and the type of work you are doing when you turn on intermediate results for a node.

Because intermediate results are disk-based, they are available from one Composite session to the next: closing a composition (or closing Composite itself) does not clear the media cache. In contrast, memory cached results are discarded when a composition is closed (or Composite itself is closed).

Composite's in-memory cache is read before attempting to read from the intermediate results. If a frame is generated and reused before the memory cache is exhausted, there won't be any need to visit the IRs in the media cache.

To create intermediate results on a node:

➤ Do one of the following:

- In the Schematic, click the orange IR tab at the lower-right of a node.
- Select a node from the dependency graph and click the IR button in the Tools Options area.

To clear the media cache of intermediate results:

➤ Open a File Browser (**Ctrl** + **O** for Windows and Linux, **Cmd** + **O** for Mac OS), right-click a composition and select Clear Cache.

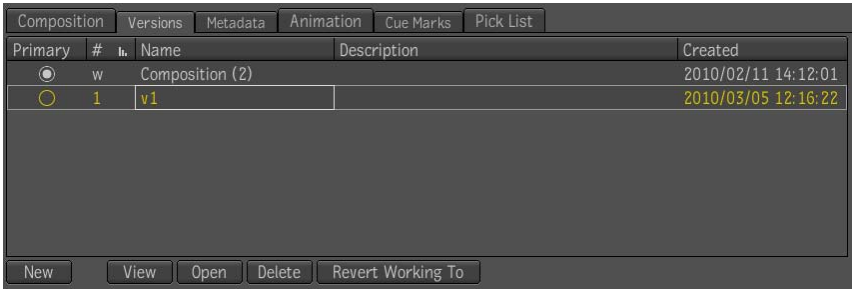
The intermediate results for all nodes of all versions of the composition are cleared.

Creating Versions of a Composition

You can create as many versions of your composition as you like, to facilitate creative decisions, to preserve a composition in a given state for later reference, or to track changes you have made.

To create a new version of a composition:

1. In the Tool Details area, click the Versions tab.
2. Click the New button, located in the bottom-left corner of the tab. You can then see the new version in Versions tab window.



3. Compositions always contain a working version and a primary version. The working version is the one that is edited by default. The primary version is other one that is linked to by default, for compositions that link to this one.

   Select any version in the Version browser to display its outputs and their rendered status in the Output browser at the right of the Version browser. You can also open, view, delete, and revert the selected version from the

Version browser. See the Viewing Rendered Versions of a Composition on page 230subsection.

When you revert to a previous version of a composition, the contents of that version are copied over to the working version.

Rendering Results over a Network

The Render Actions tab in the Project Preferences dialog allows you to edit existing render actions, and define new ones. To define a new render action, right-click in the render actions browser at left, and select "New". Select an existing render action to edit it. Render actions use variables, or tokens, that are replaced with actual values when the action is run.

Variable	Description
<InstallDir>	Composite root installation folder.
<BgTaskDir>	Points to the \resources\bgTasks folder of the Composite installation.
<PythonExec>	Points to the Python executable used by Composite.
<SharedDataDir>	Folder on the network where shared data files are stored.
<DestinationPath>	The destination folder specified in the current project's Rendered settings.
<CurrentUser>	The current user.
<CurrentProject>	The current project.
<CompositionPath>	The path to the current composition.
<ProjectFolder>	The path of the current project.
<Name>	The current composition name.
<Date>	The current date.
<Time>	The current time.
<FilenamePattern>	The file name pattern specified in the current project's Rendered settings.

Variable	Description
<Custom>	The custom string associated with the current composition.
<FrameNo>	The current frame number (only available in the file name Rendered setting).
<NbFrames>	The number of frames in the current composition.
<StartFrame>	The start frame index for the current composition.
<EndFrame>	The end frame index for the current composition.
<RenderedResult>	The current rendered result name.

For network rendering:

1. Ensure Composite is installed in the same path on every workstation that is part of the render farm, for example: *C:\Program Files\Autodesk\Autodesk Composite 2011\Composite*
2. Ensure the rendering application is installed on every workstation that is part of the render farm.

NOTE If you are using the Autodesk Backburner network rendering application, ensure that one Backburner Manager is running on one workstation and one Backburner Server is running on each workstation that is part of the render farm. In addition, each Backburner Server should be connected to the Backburner Manager.

Viewing Rendered Versions of a Composition

NOTE This procedure is aimed at advanced Composite users.

In the Versions tab, you can view the list of all versions for a composition when you have that composition open. Once you have rendered versions of a composition, you can open and view any of them, as well as revert and render any unrendered compositions you may have.

To view different versions of a composition:

1. Open a composition—see Opening and Viewing Compositions on page 220.

2 In the tool UI, select the Versions tab.

The Versions tab displays all existing versions of the composition. The working version of a composition is always listed first, followed by the most recently created versions.

3 To view a version, double-click any entry in the list.

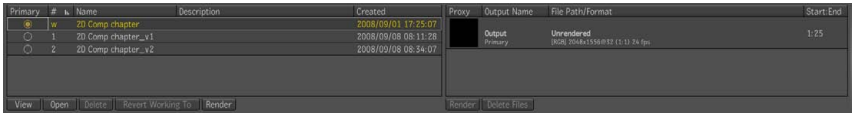
The read-only version is opened. If the version of the composition is already open, then Composite switches to display it. If you double-clicked the working version, it is opened in read/write mode.

To view the list of all rendered results for a composition to which you are linking:

1 In the Schematic view, right-click the Link Image node for the composition you are linking to, and select Open. The linked composition is opened.

2 In the tool UI, select the Versions tab.

The left side contains the list of all versions for the composition. The right side shows the result of the selected version.



To open a version:

➤ In the Versions tab, select a version to open and do one of the following:

- Click the Open button.
- Double-click the selected version.

To delete a version:

➤ On the left side of the Versions tab, select a version to delete and do one of the following:

- Click the Delete button.
- Right-click and select Delete.
- Press Delete on the keyboard.

Reverting a Composition

Reverting to a previous version of a composition lets you replace the working composition by the previous one.

To revert the working version:

➤ In the Versions tab, select a version to revert and do one of the following:

- Click the Revert Working To button.
- Right-click and select Revert Working To.

In the message box that appears, click OK to confirm the reversion.

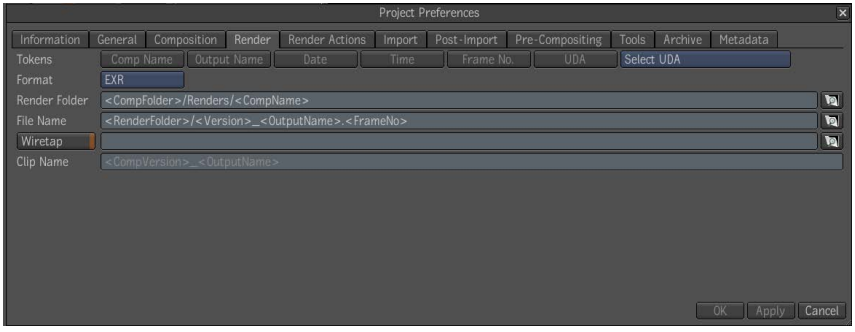
Rendering to Wiretap

The Wiretap protocol allows some degree of data exchange between Autodesk Composite and Autodesk editing and effects products, such as Autodesk Smoke®, and Autodesk Flame®. In a facility where Composite is used with one of these Wiretap-compatible products, rendering to Wiretap is an easy way for you to quickly make available the rendered result of a composition to the Flame or Smoke user—see Navigating and Browsing on page 40.

Rendering to Wiretap

To set the Wiretap mode for new compositions:

1. In the menu bar, select Edit > Project Preferences.
2. In the Project Preferences window, select the Render tab.



3. Click the Wiretap button.

4 In the text field next to the Wiretap button, do one of the following:

- Type in the path.
- Using the File Browser, navigate to a Wiretap folder, right-click the item in the right pane, and select Copy Path. Paste the contents in the Wiretap field (**Ctrl + V** for Windows and Linux or **Cmd + V** for Mac OS).
- Using the File Browser, navigate to a Wiretap folder, select the path in the browser path text field, copy it (**Ctrl + C** for Windows and Linux or **Cmd + C** for Mac OS), and then paste it in the Wiretap field (**Ctrl + V** for Windows and Linux or **Cmd + V** for Mac OS). Either one of the following forms are acceptable:
- Wiretap:<servername>/…
  or
- <servername>/…

The server specified corresponds generally to an IFFFS server. In the text field, the prefix "Wiretap:" is stripped off if present, because it is implicit. For IFFFS, the path must specify a writable folder, which is either a library or a reel. For example:
Wiretap:/belgium/stonefs/myProject/myLibrary/myReel/. The trailing slash is optional.

You can also set Wiretap rendering on existing compositions, for each output. Select an output node in the composition, and click on the Render tab in the UI. The same Wiretap controls as described above are available for the output.

Troubleshooting Wiretap

If the clip is not rendering to Wiretap, it may be for one of the following reasons:

- The Wiretap server on the destination host is down.
- The destination folder is not a library or a reel.
- The library is already opened by a Smoke or Flame user on the remote machine which will only allow the read only mode.
- A composition name contains parentheses.

Linking Compositions

By linking other compositions with your working version, you can create a more complex composition. You can link to as many compositions as you need. You can link to a specific output of a composition, as well as specify the mark in, mark out and offset times, and the repeat mode.

When you link to a composition, you are by default linking to the primary version which is read-only. The Link Image tab in the tool UI displays all the outputs associated with the version to which you are linking. When you attempt to link to a composition that is outdated, the link node turns red to indicate that it is out of date.

There are many advantages to linking compositions:

- Keeps the tool pipelines of individual compositions separate from each other. You see only your own tool pipeline. This eliminates the risk of inadvertently deleting or modifying tools added to the pipeline by another composition and can also make it easier to identify the different segments in the production pipeline.
- Makes it easy to review different possibilities for a shot. You can link to a composition and then choose the output you want to see.
- Linking makes dependencies easy and efficient. If your work depends on the result of another composition, you can link to that composition and work on the composition that contains the link. However, you may be working on multiple compositions independently and each one will stay up-to-date or will detect if a more recent version is available. This is controlled by the Link type list in the link node UI which lets you specify: primary, latest, working and explicit. For example, when linking to the Latest version, the link node will realize if it is outdated with respect to this selection. How the link node reacts to this out-of-date condition is controlled by the On Load button. When it's on, the link node will update to the proper version at load time (when the composition is open in read-only). When On Load is off, it is up to you to visit the link node and click the Update To button to link to the proper version.
- Makes it easy to apply your tool pipeline to a different rendered result. You just select the result.

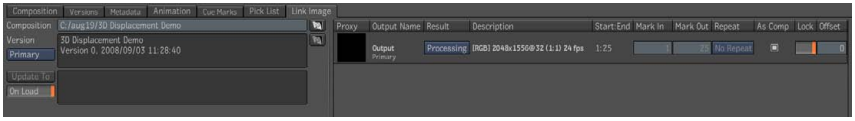
To link to a composition:

1. In a File Browser (**Ctrl** + **O** for Windows and Linux or **Cmd** + **O** for Mac OS), select a composition and drag it into the Schematic view.

2 In the gate that appears, swipe south through the Link option.

The linked composition appears in the Schematic. If this is the first time you are linking to a composition from this composition, the Link Image tab appears in the tool UI. If a Link Image tab already exists for this composition, its contents update to reflect those of the composition to which you just linked.

3 Select the Link Image tab.



4 Each version may have multiple outputs. The link node will have as many output sockets as there are output nodes in the currently linked version. If necessary, change the version to which you are linking, by selecting one from the Version list:

Primary Link to the primary version of the composition.

Latest Link to the most recently created version of the composition. This is a common choice at the beginning of a project when compositors are experimenting with ideas and nothing is locked down yet; you want to see the latest idea for a shot.

Working Link to the working version of the linked composition.

Explicit Link to any result. Click the Browse button beside the text field. In the Version browser that opens, select the output to which you want to link.

5 The Update To button is highlighted to indicate a discrepancy between the currently linked version and the Link mode being used. This option is also available when you use the "Primary" link mode and the Primary version is changed (i.e. version 2 becomes primary, whereas version 1 was primary beforehand). This option is not available if you are using an explicit version.

6 When using the Primary or Latest mode, click the On Load button to automatically update the link node when the composition that contains

the link node is opened. The update will not occur if you are viewing a version as viewed versions are read-only.

7 Set the mark in, mark out, offset, and repeat times by editing the respective fields. Or you can select the As Comp option to use the linked composition's times.

NOTE If there are further updates that occur while the composition is open, you must update them manually.

Things to Remember

- The Player Display preference determines what the Player displays (tool input, rendered result, etc.).

Determining the Status of a Linked Composition

You can determine whether the composition to which you are linking is up-to-date by checking the linked node's tool UI (Link Image tab). The Link Image tab is common to all linked compositions; you must click the node of a linked composition to see its information in the Link Image tab.

Linking to a Composition

NOTE This procedure is aimed at advanced Composite users.

When linking to a composition, you are actually linking to the primary version of the composition. You can use the result of any output of any version of the linked composition—see Linking Compositions on page 234.

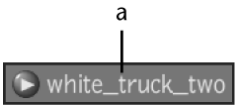
Closing a Composition

When you're finished working on a single composition, you can close it.

NOTE If there are multiple versions of the same composition open, there is no way to close them all in one operation.

To close a composition:

1. If the composition is not selected, in the taskbar, click the name of the current composition (to the right of the composition icon) and then select the composition you want to close.



(a) Name of the current composition in the taskbar

Alternatively, in a File Browser, click the composition you want to close.

2. Select File > Close or press **Ctrl + W** (for Windows or Linux) or **Cmd + W** (for Mac OS).

Things to Remember

- All open compositions are automatically closed when you exit. You cannot keep a composition open between sessions.

Multilayer Compositing and 3D Effects

12

Topics in this chapter:

Reaction Compositing and Effects

The Reaction super tool provides a complete multilayer compositing and 3D effects environment. You can use Reaction to quickly composite and output numerous images that are already processed using other tools, as well as cross over into a 3D environment where you can apply lighting, spatial, visualization, and rendering effects to your scene. You can then use your Reaction output to render your final composition, or use it as an input to other tools in your dependency graph, including another Reaction.

What is a Reaction Super Tool?

A Reaction tool is considered a "super tool" because it contains a set of tools that you can use to create 3D compositing effects on your composition. All individual tools contained in Reaction can be accessed by opening the Group Schematic. Reaction is used like any other tool, but can also be used in tandem with the Layer Editor to build and edit compositions interactively. You can use as little or as much of the available functionality as needed to get your job done. You may want to simply create a few layers where one layer is repositioned to place a character in a scene, or create a 3D scene with lighting and texture effects. Regardless, the Reaction tool behaves like any other tool in Composite; you can delete and connect a Reaction tool to other output nodes in the same manner. As always, the universal Tool Options panel is located at the right of the Tool UI—see The Tool UI on page 23.

Reaction Concepts

The following terms and concepts are used when working in Reaction.

Object

An object is any element in a 3D scene. Lights, cameras, and surfaces are typically referred to as *objects* when working in Reaction.

Source

A source is an input node to the Reaction tool. You can assign a source composition or image generator to a layer in your composition, or use the source as an input to a texture channel in a material node or image channel in a light node. When you add a Reaction tool to your dependency graph, its initial input is designated as the background for your composition. You can then add as many source nodes and layers to the Reaction tool as needed.

Layer

When performing multilayer compositing using Reaction, you typically set a background, although you are not required to, then assign source inputs to layers. A layer is composed of three separate components: a surface (3D geometry) object and local axis, a multi-channel material object, and a layer element. When viewing the Reaction node in the Group Schematic, you can see each component—see Working with Layers on page 251.

Material

A material defines how a layer interacts with lights to define how a scene is rendered. Each layer must have a material associated with it, but many layers can share a single material—see Working with Materials on page 257.

Surface

A surface is a geometric object onto which a source is mapped to create a layer for your composition. Available geometric types are bicubic, bilinear, frustum, box, geometry, plane, and sphere. Each surface has a local axis that you can use to transform the layer—see Working with Geometric Surfaces on page 254.

Camera

Each Reaction node includes a camera for viewing and rendering the scene. By default, the camera is set to perspective, and positioned, so that you can view your layers in 2D and immediately perform standard multilayer compositing, although you may want to set the camera to orthographic. You can add several cameras to a scene, but you can only choose one to render, called the render camera—see Working with Cameras on page 274.

Axis

Each layer has its own axis, or center, that you can use to perform transformations locally. You can also add axes to your scene to apply transformations to several objects at once, or to create more complex transformation effects. When you add an axis object to a Reaction tool, you can then parent it to one or more objects or layers—see Using Parenting Axes on page 256.

Light

Lights are used to illuminate your scene and apply lighting effects to surfaces. You can choose omni, directional, or spotlights and set lighting color, intensity, and decay, as well as create shadows and apply transformations to the light itself. Lights also have a Projector Image channel that you can connect to a

source to project an illuminated image onto a layer or the scene—see Working with Lights on page 268.

Shader

Shaders are programs that help define the look of objects in your scene. They can be considered a part of the rendering pipeline.

Compositing Workflow

The Reaction tool lets you create 2D multilayer compositions and perform 3D compositing with applied transformations and other effects.

NOTE You can also use the 2D Compositor tool to create multilayer 2D compositions, but without integrated transformations—see 2D Compositor on page 339.

Basic and Advanced Workflows

The following shows examples of a typical workflow you might follow when working with Reaction to perform multilayer compositing, or to create and integrate compelling 3D effects into your compositions.

Basic Multilayer Compositing in Reaction

1. Add a Reaction tool to your composition and set the composition background (optional).
2. Add sources and create layers.
3. Set layer priority and modify their properties.

3D Compositing in Reaction

1. Add a Reaction tool to your composition and set the composition background, which is optional. An empty background is set by default.
2. Add sources and create layers.
3. Modify or enhance the scene by:
   - Adding objects, such as lights, cameras, and material nodes, and edit their properties.

- Selecting and manipulating objects. You can transform (scale, rotate, and translate) any surfaces, cameras, and lights in your scene using the 3D manipulators and icons. You can also organize objects into hierarchies for easy manipulation.
- Working with cameras. You can add cameras, transform cameras, and define other properties, such as depth of field and field of view.
- Working with lights. You can add different types of lights and place them in your scene. Then you can set the color and falloff, and define different types of shadows.
- Animating properties. Move a camera, dim a light, or change the color of a material over time. All properties in Composite can be animated.
- Setting visibility properties. All objects, geometric objects, lights, and cameras are defined by their visibility and rendering properties. For example, you can determine whether a geometric object is visible, whether it casts shadows, and whether its reflection is visible.
- Working with materials and shaders. You can use the hardware renderer for fast results, or use software shaders to create special rendering effects, such as flares, fog, and flames. A shader modifies the behavior of the renderer while rendering an object in a scene.

Basic Compositing in Reaction

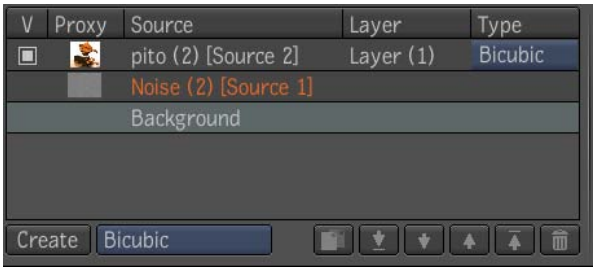
Whether you want to quickly perform straightforward 2D compositing or create 3D effects, the first thing you'll do is set your composition background to define the rendering plane, then you'll create layers in Reaction.

NOTE You do not have to connect an image generator to set the background; you can work on an empty background. You'll see that when you build a composition using the Layer Editor; the background appears automatically when you add a Reaction tool to your composition.

Building a Composition Using the Layer Editor

The Layer Editor is designed to work with Reaction and lets you quickly build and edit your composition while maintaining a clear view of the layers in your Reaction composition. As in the Schematic view, you can work with the Gate UI to create layers and perform other tasks. Then you can set the layer

priority, create, and access tools that affect the selected layer—see Layer Editor on page 150.



Setting the Player Settings

Before you do anything in Reaction, be sure to verify that your Player settings are set properly.

To set the Player display settings for Reaction:

1 With the cursor over the Player, press the tilde key (~) or middle-click to display the Gate UI. Swipe south to display the Player Options.

2 Under Display, select Tool Output to view the results of your Reaction composition in the Player.

NOTE When you are working with a default Compositing task group preset, Tool Output is enabled.

TIP You can set a context point for the Player to display the Reaction output. This is useful when you are working with more than one player, or you are editing a composition with Reaction using the Layer Editor and you always want to see your Reaction output—see Setting Context Points on page 191.

---

3 Under Guides, select Manipulators to view transformation and object icons. Manipulators is enabled by default.

4 Under Reaction, select a plane (XY, XZ, or YZ) for the grid, adjust the size of the grid and the distance between grid lines. The grid helps you place elements in 3D space.

NOTE To work with grids, you must select the Interactivity option (under Reaction).

5 Under Reaction, select a Highlight option for displaying objects:

- Coverage: Display the pixels of selected layers that contribute to the final composition. Adjust the transparency of the pixels by entering a value in the Threshold field.
- Object: Displays an object's geometry with a colored outline.

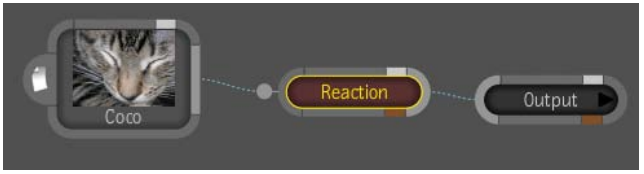
6 Under Reaction, select Interactivity to use the hardware renderer and achieve dynamic results while you work. The results you see when working with the hardware renderer may differ slightly from the results achieved using the software renderer.

NOTE If you are planning to quickly composite a series of 2D layers without manipulating or transforming layers or adding 3D effects, you do not need to enable Interactivity.

NOTE To work with lights and cameras, you must also select Icons (under Rendering) in the Render tab of the Reaction tool.

Adding a Reaction Tool and Setting the Composition Background

When you perform multilayer compositing in Composite, you typically create a new composition, then add a Reaction tool to start building from scratch. However, you can also connect Reaction to any composition or tool output in an existing dependency graph in the Schematic view. This automatically sets the background for the composition.

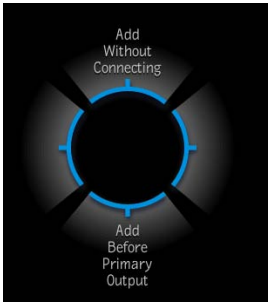


NOTE You can add, connect, and delete a Reaction tool as you would any other tool.

To add Reaction to a new composition:

➤ In the Tools tab, drag the Reaction tool to the Player, Layer Editor, or Schematic.

If you dropped Reaction on a Player or an empty Layer Editor view, the composition drop gate appears. You can link Reaction to the composition Output node, or add it to the current composition without connecting it to anything.



To add Reaction to the current composition:

➤ In the Tools tab, drag the Reaction tool to Schematic.

The background for the Reaction node is set.

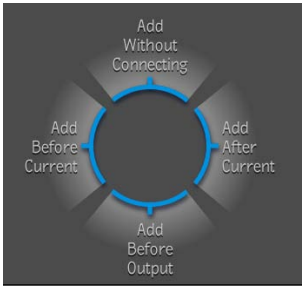
Setting the Background

Before you composite or add effects to a composition in Reaction, you typically set the background. The background is the rendering plane for your composition and sets the format. If you connect a Reaction tool to an image or to a tool output in an existing dependency graph, the background is set automatically. You can change the background at any time.

To set the background automatically in the Layer Editor:

1 In the Tools tab, drag the Reaction tool to the Layer Editor.

The composition drop gate appears. Depending on your dependency graph, you will see some or all of the following choices:



2 Decide where to add Reaction in your dependency graph.

The background is set accordingly, and appears in the Layer Editor.

To set the background automatically in the Schematic view:

➤ Drop the Reaction tool onto an existing tool or image output connection in the Schematic view.

To set the background manually in the Schematic view:

➤ Connect a media or tool node output to the background input connector.

To set or replace the background using the drop gate:

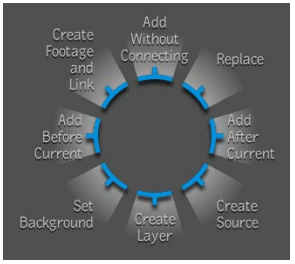
1 Select the desired Reaction tool in the Schematic or Layer Editor.

In the Schematic view, the Reaction tool is highlighted.

2 Select the composition to use as the background and drag it to the Schematic, Layer Editor, or Player.

The Reaction drop gate appears.

3 Swipe through Set Background.

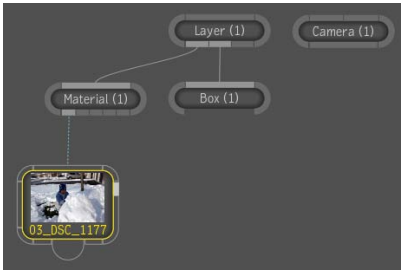


NOTE You cannot delete a background input connector.

Creating Sources and Layers

You can composite as many layers as you want using Reaction, but first you must create the layers. A layer in Reaction is composed of a source input that you assign to a geometrical support, or 3D geometry, and a material node that is automatically generated—see Working with Geometric Surfaces on page 254.

NOTE To view the Material and geometrical support nodes that are generated with a layer, you must be in the Group Schematic.



If you want to perform basic multilayer compositing, where you work with bilinear geometric surfaces only, you can quickly create a layer in a single step in the Layer Editor or Schematic by using the Reaction drop gate. To substitute a 3D geometric surface as your layer support, you can then simply choose a different geometric surface. After creating the sources and layers, you can publish a 2D composition if you are satisfied with the results.

To create one or more sources and layers in a single step:

1. In the Schematic view, select the Reaction node.
2. Select images or footage from the File Browser or desktop, and drop (or stick) them onto the Schematic, Layer Editor, or Player.

   The composition drop gate appears.
3. Swipe through Create Layer.

   The sources and layers are added in the order in which you selected the images/footage (from first to last) and labeled accordingly in the Layer Editor.

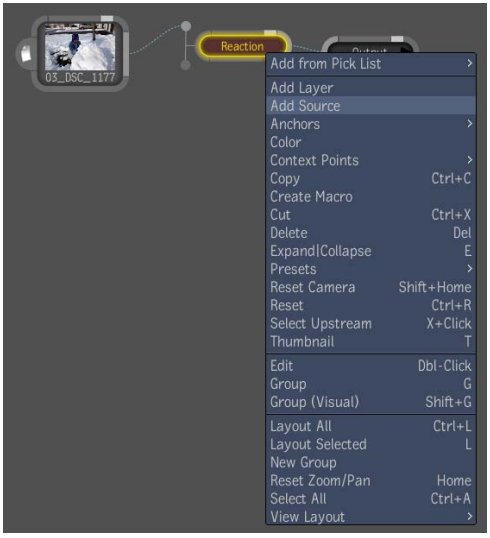
To add a layer:

➤ Do one of the following:

- At the bottom of the Layer Editor, select a surface geometry type and click Create.
  A layer with the surface you chose and a source are created.
- In the Schematic view, right-click the Reaction node and select Add Layer.
  A layer and a source are created. By default, the Bilinear surface is assigned to the layer.

To add a source:

1 In the Schematic view, right-click the Reaction tool and select Add Source.



2 A source input connector appears in the Schematic view. You can connect media or a tool node to it.

To add a layer and link to its source:

➤ In the Layer Editor, right-click a layer and select Add Linked Layer.
A layer is created and linked with its source.

To add one or more source compositions to Reaction in a single step:

1. Select the Reaction tool in the Schematic view or select the desired Reaction in the Layer Editor.
2. Select one or more compositions from the File Browser or Desktop and drop (or stick) them onto the Layer Editor, Schematic, or Player.

   The drop gate appears.
3. Swipe through Create Source.

   The sources are added in the order in which you selected the compositions (from first to last) and are labeled accordingly in the Layer Editor.

To create a layer using a source:

➤ In the Layer Editor, right-click a source and select Create Layer Using Source.

A layer is created in the Layer Editor.

NOTE You can immediately change the surface geometry of the layer using options in the Type column of the Layer Editor.

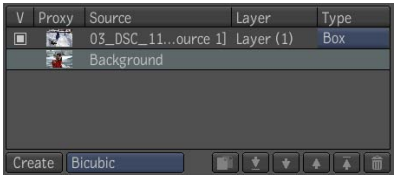
NOTE You can add more than one layer to a source if you want to create multiple layers using the same source image. If you have applied effects to the source image, they will be propagated to the layers to which they are linked.

Working with Maya Pre-Comps

Once created, you can automatically generate and update compositions based on Maya render layer and pass setups. This accelerates and streamlines the 3D to 2D workflow by allowing the 3D artist to pre-visualize a scene in the form of the final composite while continuing to iterate on and refine only required elements. The format supports different passes per layer and per camera; passes from pairs or sets of cameras can be imported side-by-side within a composition, allowing their dependency graphs to be manipulated in tandem, and making it easy to select pairs of nodes for viewing as stereoscopic output. For more information, see FBX Import Limitations on page 329.

Working with Layers

When you create a layer, it is automatically added to the top of the Layer Editor. The source name is displayed along with the name of the node to which it is connected. Also, each layer is numbered, so you can easily identify it. After creating layers, you can use the arrow buttons in the Layer Editor to quickly reorder layers. From the tool UI, you can interactively apply 2D and 3D transformations, as well as set global layer properties and visibility options. And because layers are discrete objects, you can also parent layers to axis objects in the Group Schematic.



Renaming Sources and Layers

Renaming sources and layers lets you quickly identify a layer and its source in the Group Schematic. You can rename sources and layers in the Layer Editor or the Tool Options at the right of the tool UI.

To rename a source or layer:

1. In the Layer Editor, click the name of a source or layer.
2. Press **F2** and type in a new name.

Selecting Layers

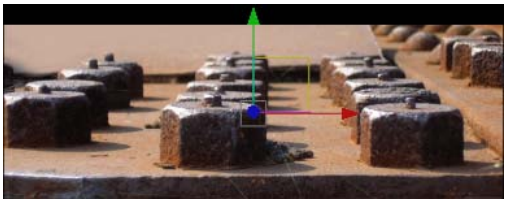
You can select a layer in several ways: through the Layer Editor, from the Group Schematic, or by clicking a layer in the Player.

To select a layer:

➤ Do one of the following:

- In the Layer Editor, click to select a layer.
- In the Schematic view, double-click the Reaction node to open the Group Schematic. Click a Layer node.
- In the Player, select a layer.

The selected layer is shown with the manipulator icon that represents the three axes.



NOTE To view the manipulator icon, make sure you have selected Manipulators in the Player Options.

Duplicating Layers

Duplicating a layer gives you an identical layer that includes the source, material, and surface type.

To duplicate a layer:

➤ Select a layer from the Layer Editor and do one of the following:

- At the bottom of the Layer Editor, click the Duplicate Layer button.
- Right-click a layer and select Duplicate Layer.

Reordering Layers

You can reorder layers interactively using the arrow buttons in the Layer Editor. Layers are composited in the order in which they appear in the list.

To reorder layers:

1. Select a layer from the Layer Editor.
2. Use the arrow buttons at the bottom of the Layer Editor to reorder the layer.

Modifying Motion Blur on Individual Layers

You can modify the motion blur applied to your composition on individual layers, so that each layer appears to be moving at a different speed than other layers—see Adding Motion Blur and Depth of Field to a Scene on page 292.

To modify the motion blur on a layer:

1. Select a layer from the Layer Editor.
2. Enter a value in the Time Dilation field. A higher value speeds up the blur effect for the layer; a lower value slows it down.

Setting Layer Visibility Properties

You can set layer visibility properties to specify how the layer is rendered or "seen" by the camera.

To set the visibility options:

➤ Select any of the following in the Reaction tab.

Thumbnails:	Size (in pixels):
Visible to Camera	The layer is visible to the camera. If you hide the layer from the camera, the layer may still cast a shadow and can be seen by a light.
Cast Shadow	The layer can cast shadows.
Receive Shadow	The layer receives shadows cast by itself and by other layers in the composition.
Front Face	The front faces of the layer will be rendered.

Thumbnails:	Size (in pixels):
Back Face	The back faces of the layer will be rendered.
Time Dilation	Modifies the motion blur that affects layers animated in the scene. You can give the layer the appearance of moving at a different speed than is set for Motion Blur in the Render tab.

Working with Geometric Surfaces

When you create a layer, you are assigning a source input to a geometric object that defines the surface on which the source is mapped. You can replace the assigned surface object with a different one, as well as disconnect a surface object in the Schematic and replace it with another. By default, when you create a layer, the surface type is determined by the last geometric setting chosen in the Layer Editor.

NOTE When you create a layer in the Schematic view, the default surface is Bilinear.

There are seven surface types available in Reaction: bicubic, bilinear, box, frustum, geometry, plane, and sphere. The bilinear surface is a planar surface with four vertices, one at each corner. The vertices are joined by straight line segments using linear interpolation. The box, frustum, and sphere surfaces are 3D primitives meshes with no control vertices.

Modifying Surface Shape and Geometry

You can change a surface's shape by scaling it using the transformation tools. You can also modify a surface's base geometry from the Surfaces tab in the Reaction tool UI. You can increase or decrease the number of segments to change the number and shape of the surface's polygons, which may affect its shape, how it reacts to lights in the scene, and how texture maps are displayed.

TIP You can toggle the display of the icons that represent the camera, lights, axis, and transformation by selecting Icons on the Render tab.

To modify a surface's geometry:

1. Select the Surfaces tab.
2. Select a layer from the Layer Editor.

3 To change the surface, select a surface type under the Type column.

4 In the Surfaces tab, adjust the properties to modify the surface's geometry.

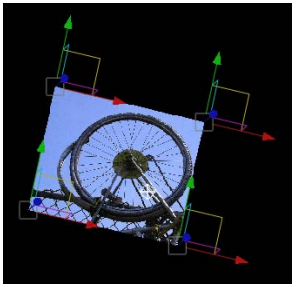
Surface Type	Description
Bicubic	
Bilinear	You can change the number of segments to increase or decrease the number of polygons, and change their shape. You can also move corner points in X, Y, and Z to create a variety of 4-sided shapes. Because you can also move corners in Z, you can create warped effects.
Box	You can change the number and size of segments to increase or decrease the number of polygons, and change their shape.
Frustum	You can change the geometry of the top, bottom, and height, as well as the number of segments to increase or decrease the number of polygons.
Geometry	
Plane	You can change the number of segments to increase or decrease the number of polygons, and change their shape.
Sphere	You can change the number of segments to increase or decrease the number of polygons, and modify the smoothness of the sphere, or change its shape. When you reduce the number of segments, you will create a more faceted appearing sphere that will also affect how it reacts to lights. You can also modify the radius to change the size of the sphere.

To modify a bilinear surface interactively:

1 In the Layer Editor, select a layer with a Bilinear surface.

2 Select the Surfaces tab.

In the Player, manipulators appear at each corner of the surface.



3 Adjust the manipulators to modify the surface.

TIP You can also select a bilinear surface by opening the Group Schematic.

Using Parenting Axes

You can add axes to a 3D scene and parent them to cameras, lights, layers, and other axes in the Group Schematic. This lets you control the movement of one or more child objects by transforming the parent axis.

You can transform axes interactively using the transformation buttons in the Axes tab, where you can also set values for the axis position, rotation, scale, and pivot. You can animate and set expressions for all transformation values, as well as assign them to a tracker. The axis is represented by a white cross in the Player, which you can select and interactively move with the manipulator.

NOTE The axis icon is visible only if you have selected Icons in the Render tab of Reaction.

To add an axis from the Axes tab:

1 In the Schematic view, select the Reaction node and then select the Axes tab.

2 From the Axes list, click Add.

An axis is added to the Axes list.

3 In the Schematic view, double-click the Reaction node to display the Group Schematic.

4 Parent the axis to a layer, camera, or light.

To add an axis from the Tools tab:

1. In the Schematic view, double-click the Reaction node to display the Group Schematic.
2. From the Tools tab, drag the Axis tool from the Reaction folder to the Group Schematic.
3. In the Schematic view, parent the axis to a layer, camera, or light.

To set the visibility of an axis:

1. In the Schematic view, select the Reaction node.
2. Select the Axes tab.
3. From the Axes list, click the white square beside an axis.

Working with Materials

When you create a layer, a separate material node is automatically generated and linked to the layer. Every layer must be associated with a material, but many layers can share a single material. Because the material node is separate from the layer and surface, you can easily propagate material properties to several layers from a single source, instead of duplicating effort by having to assign the same material attributes to several surfaces. You can quickly connect materials to layers from the Reaction Group Schematic and see which layers share materials.

About Materials, Shaders, and Textures

A material defines how the layer and its associated surface object appears. It defines how it interacts with lights, how it reflects light back to the view, and how it blends in with the rest of the scene. You can choose one of four shader types to set the basic material properties for a given layer and choose from a wide selection of blending modes to define how the current layer is blended with the scene element lying behind it (with respect to the view).

In addition, each material node has five input tabs, or channels, that you can use to assign one or more sources to apply textures to an object. The main material channel defines a layer's base material and texture mapping, and is always used. You can optionally connect sources to the other inputs on the

material node to define a bump map, as well as radiosity, reflection, or refraction environment maps, depending on the shader type selected.

Textures are 2D images that can be wrapped around an object's surface, much like a piece of paper wrapped around an object. The information displayed on the object's surface depends on the type of texture map used. In bump maps or normals maps, RGB or luminance values are used to give the surface the illusion of peaks and valleys. In radiosity maps, the object appears to blend in the source image's radiosity. This is useful for creating realistic lighting effects without the high rendering cost associated with raytracing or radiosity.

Setting Material Properties

When you create a layer, a material is associated with it automatically. By default, the material displays the source image on the layer surface using the Standard shader, which gives you immediate access to the full range of parameters in the Materials tab.

To access the Materials UI:

➤ Do one of the following:

- In the Layer Editor, select a layer and then select the Materials tab.
- In the Schematic view, right-click the Reaction node and select Edit Group to display the Group Schematic, and then click a material node to select it.
  The Materials UI displays the properties for the selected material node.

Materials UI

The Materials UI contains a comprehensive set of options and controls for setting the shader type, blending mode, shading parameters, color factors, and texture mapping and blending.

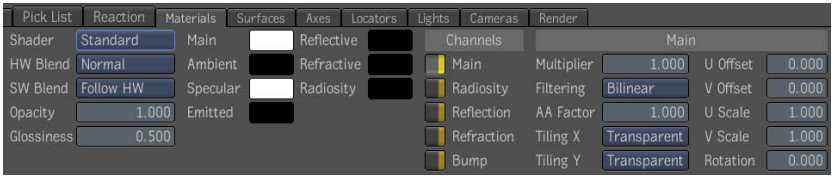
Shader Types

There are four shader types: Standard, Simple, Shadow Matte, and No Shade.

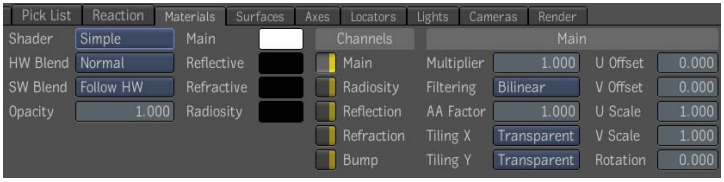
To select a shader type:

➤ In the Materials UI, select a shader:

- **Standard** The full range of parameters for creating diverse 3D effects using lights and cameras.

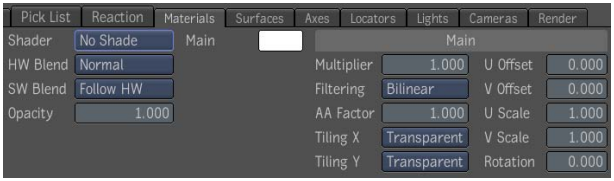


- **Simple** A subset of Standard shader parameters. It does not consider lights in the scene, but you can create lighting effects using environment maps.



- **Shadow Matte** Used to blend shadows into a scene. By adding objects to a scene, you can create realistic shadow effects.

- **No Shade** Lets you set parameters and texture mapping for the main material channel only. You rely on the lighting effects in the source image to provide illumination.



Blending Modes

Blending modes are available with all shader types. A blending mode defines how the color of one layer is blended with the color of the layer or part of the scene lying behind it (relative to the Player). You can animate and apply expressions to blending modes—see Animation on page 659 and Expressions on page 719.

There are two categories of blending modes:

Software Lets you select the blending mode used by the software renderer. If you selected Interactivity in the Player Options to enable the hardware renderer for fast results while you work, you can set the Software blending mode to Follow HW. This ensures that the software and hardware renderer produce almost exactly the same results. The Follow HW option is selected by default.

NOTE Because the hardware and software renderer use different methods to calculate results, it is recommended that you perform all transformations using the hardware renderer, but verify your results using the software renderer.

Hardware Lets you select the blending modes used by the hardware renderer, which can only reproduce a subset of the blending mode effects available to the software renderer.

Blending Mode	Description	SW	HW
Follow HW	Set results to follow the hardware renderer.	X	
Normal	Maintains the normal colors for the layer.	X	X
Premultiplied	Multiplies the colors of the current layer with the alpha of the layer behind it.	X	X
Dissolve	Randomly dissolves pixels of the current layer over the layers behind it.	X	
Add	Adds the Red, Green, and Blue values of the current layer's pixels to the layers behind it. The resulting composite can make the layer's pixels appear very bright, especially over a bright background.	X	X
Subtract	Subtracts the Red, Green, and Blue values of the current layer's pixels from the layers behind it. The resulting composite can make the layer's pixels appear very dark.	X	X

Blending Mode	Description	SW	HW
Multiply	Multiplies the pixel values of the current layer with the pixels in the background, and clips all RGB values at 255. The overall effect is similar to drawing with a colored marker over an image: it darkens and colorizes at the same time using the current layer imagery.	X	X
Spotlight	Shines a spotlight uniformly on the current layer.	X	X
Spotlight Blend	Shines a spotlight on the layer and blends with the layer behind it.	X	X
Screen	Combines the pixels in the current layer with the pixels in the background so that the current layer is composited over the layers in the background with lighter pixels than before. The effect is similar to the photographic technique of combining two slides in a slide "sandwich" and then reshooting them. Screen mode is the inverse of Multiply mode.	X	X
Overlay	Displays the image through a gel of the current layer. It combines the colors of the current layer with those of the layers behind it to create new tints based on these results. It boosts contrast and color saturation at the same time.	X	
Soft Light	Shines a soft, diffuse light associated with the current layer onto the layers behind it. It reduces the contrast levels in the image.	X	
Hard Light	Shines a harsh light associated with the current layer onto the layers behind it. It primarily affects areas of detail, and greatly reduces the contrast levels in the image.	X	
Darken	Composites only the pixels of the current layer that are darker than the pixels of the layers behind it.	X	X
Lighten	Composites only the pixels of the current layer that are lighter than the pixels of the layers behind it.	X	X
Difference	Displays the difference between the pixels in the current layer and the pixels of the layers behind it. When a brighter pixel is subtracted from a darker pixel, the	X	

Blending Mode	Description	SW	HW
	positive value of the color is used and results in bright color shifts. The layer order is not significant, as both layers contribute nearly equally to the result.		
Exclusion	Creates an effect similar to Difference, but lower in contrast and resulting in a grayer image. The layer order is not significant, as both layers contribute nearly equally to the result.	X	X
Hue	Changes the hue of the current layer to the hue of the layers behind it. This effectively makes the current layer take on the "tint" of the elements in the background of the composite.	X	
Saturation	Changes the saturation of the current layer to the saturation values of the layers behind it. (This differs from Saturate, which uses the saturation of the current layer to increase that of the layers behind it.)	X	
Color	Changes the hue and saturation of the current layer to the hue and saturation values of the layers behind it.	X	
Luminance	Changes the luminance of the layers in the background of the composite to the luminance value of the current layer.	X	

Basic Shading Parameters

The basic shading parameters let you set a layer's opacity. Depending on the shader type selected, you can also set the refractive index and surface glossiness.

NOTE Basic shading parameters do not apply to the Shadow Matte shader—see Using the Shadow Matte Shader on page 268.

To set basic shading parameters:

➤ Enter a value for the object's opacity, refractive index, and glossiness.

Opacity Sets the transparency for the layer. Values are measured in percent. A value of 100 is opaque, a value of 0 is transparent. If you want to create a transparent surface, such as glass, a low value (10) will yield a more convincing result than 0.

NOTE Transparency affects the visibility of a layer's other attributes.

Glossiness Sets the level of glossiness for the layer, which affects the specular highlights created when a light shines directly on a layer. A glossier, more reflective surface yields a smaller, sharper highlight. A less glossy surface has a highlight that is more diffuse or blurry. Glossiness is available with the Standard shader only. Values range from 0 to 1, with a value of 1 being completely reflective. In this case, you would not be able to see any of the objects other surface properties, so glossiness should be set to less than 1.

Setting Material Colors

You can set color factors for each texture channel available for the type of shader you select. If you are working with a Standard shader, you can also set color factors for ambient and specular lighting effects. The colors you set are multiplied with the colors of the input source image connected to that channel. The Shadow Matte shader has no color factors.

NOTE You can choose to add a layer without a source to your composition. For example, if you want to add a colored ball to your scene, you can simply create a layer and set its color in the Materials tab. In this case, the main color would be the color you set.

Color	Description
Main	Sets a color to modulate the main material color. The Main color is a diffuse color that the light scatters in all directions, so that the layer surface appears to have the same brightness from all viewing angles. Available for all Shaders except for Shadow Matte.
Ambient	Sets a color to modulate the color of areas of the object that are shielded from direct light sources, but are still visible, due to non-directional ambient lighting that exists in the scene. Available in the Standard shader only.
Specular	Sets the color of shiny highlights on the layer surface. It is usually set to white or a brighter shade of the main color. Available in the Standard shader only.
Emitted	Sets a color to modulate the color emitted from an object in the scene. Available in the Standard shader only.

Color	Description
Reflective	Sets a color to modulate the color of layer surfaces with reflective properties and/or reflection maps applied. You typically set a grayscale value, with white being reflective, and black having no reflectivity.
Refractive	Sets a color to modulate the subtle changes in color that occur as light passes through an object where transparency or translucency is incorporated. Black has no refractive properties.
Radiosity	Sets a color to modulate radiosity effects applied to your scene. Radiosity effects add a high degree of realism to images as it considers all light in an entire environment and simulates what happens when rays of light hit an object. Some stick (where a surface is opaque and absorbs the light), and others are reflected and refracted. These rays then go on to illuminate other surfaces before reaching the human eye. This yields indirect lighting and color bleeding effects ideal for global illumination.

To set material color values:

➤ Click the color pot of the channel that you want to set a color.

The Color Picker is displayed, so you can set color values—see Color Picker on page 154.

Using Texture or Environment Maps to Control Surface Attributes

The Main texture channel defines the surface's general appearance, but if your main texture is not enough to give you the look you want, you can add more texture maps. Each Material node has five texture channels that you can use to apply texture and environment maps to refine the appearance of an object's surface. You can set parameters for each channel.

The Main and Bump texture channels let you wrap and position a texture on the surface. By default, the image is clamped to the surface. You can then use placement parameters to position the source image on the surface and apply tiling effects.

The Radiosity, Reflection, and Refraction channels let you apply environment maps to your surface object. The environment maps use their source image to surround the surface with a virtual sphere to simulate an environment, which shows up as reflections on the surfaces of objects with reflective properties. The environment map always covers the sphere exactly once. You can transform your object and see the effects of the environment map change

dynamically. Using environment maps is a fast and inexpensive way to achieve highly realistic lighting effects.

NOTE Material nodes in the Group Schematic always show all five input tabs. However, the number of active input tabs depends on the channels available for the selected Shader type.

To connect a source image to a texture or environment map channel:

1. Do one of the following to display the Group Schematic:
   - In the Schematic view, double-click the Reaction tool.
   - In the Schematic view, right-click the Reaction node and select Edit Group.
2. Connect a source image node to a channel tab on a Material node. To identify the nodes, pause over a material node to see a tooltip with the node name.

Material Texture Channel	Description
Main	The main channel is used to apply a source image as a texture map on a selected object's surface. You can position the material on the surface using the UV placement controls, and tile the image in U and V.
Reflection	A reflection map uses the source image to create realistic reflections. You can use the color factor to set areas and the degree of reflectivity for the material. You typically set a grayscale color value, with white being completely reflective, and black having no reflectivity. You can, however, achieve tinted reflections by set setting a color.
Refraction	A refraction map is an environment map that can be used to simulate how light traveling through a refractive, transparent material is distorted.
Radiosity	The radiosity map uses the source's color and luminance values to add radiosity effects to the layer surface.
Bump	A bump map is usually a different image that you specify as a source for a bumpy or textured appearance on a surface. It is sensitive to light sources. Its RGB (when using an RGB image, it is a normals map) or luminance values are used to

Material Texture Channel	Description
	simulate relief on objects by perturbing the surface shading according to an image map. The geometry is not affected.

3 To exit the Group Schematic, double-click anywhere in the Schematic.

NOTE When using an RGB image as an input to the Bump channel, the RGB values map to XYZ when calculating the effects of light on the surface normals.

TIP Black and white or grayscale images are easy to work with when adding a bump map, as you can quickly identify how the bump map will appear on the target surface. Black areas create ridges, and white areas create indentations.

Things to remember

- Only the Standard Shader considers lights in the scene.
- Using environment maps adds lighting effects to the associated objects only.

To set texture and environment map parameters:

1 In the Materials UI, select a layer from the Layer Editor.

2 Select the Simple or Standard shader.

3 Under Channels, select a channel: Main, Radiosity, Reflection, Refraction, or Bump.

4 Adjust the parameters to control image brightness and texture quality, and in the case of the Main and Bump textures, specify how the texture is positioned on the object.

The following parameters are available depending on the channel you selected:

Property	Description
Multiplier	Uniformly scales the brightness of the image texture up or down. This is useful for setting the amount that a given texture affects the object's overall look. The default value is 1, but you can overdrive the brightness to achieve specific effects.

Property	Description
Filtering	Sets the quality of the texture image. When a texture is stretched onto an object, it may, depending on the camera position, lose some of its crispness. The rendered can compensate for this, but it may slow the process. Filtering options include: ■ **Nearest** A box filter and the fastest way to resample an image since it only samples a single pixel of the input image to determine the value of a given pixel in the result image. Produces significant amount of aliasing. ■ **Bilinear** A separable triangular filter that takes into account more area when resampling. ■ **Mitchell** Considered one of the best magnification filters for images; has a good balance between ringing and sharpness. ■ **Gaussian** Good magnification and magnification filter with no ringing, but introduces noticeable softness to the result image. ■ **Jinc 2** Offers better sharpness than the Gaussian filter, but at the expense of ringing. Similar to the sinc filter, but with better isotropic qualities, less ringing, and same sharpening and anti-aliasing. ■ **Jinc 3** Offers better sharpness than the Gaussian and Jinc 2 filters but at the expense of even more ringing. Similar to the Lanczos filter but with better isotropic qualities, less ringing, and same sharpening and anti-aliasing.
AA Factor	Lets you adjust the trade-off between anti-aliasing and blurriness.
Refract Inx	Sets the amount of refraction for the current layer when using a refractive texture. The higher the value, the greater the amount of light The higher the value, the greater the amount of light dispersion (splitting), which increases the object's brilliance.
Tiling X, Tiling Y	Set the repeat mode. Choose from; Transparent, Edge, Repeat, and Mirror.
U Offset,V Offset	Translates the texture in U and/or V.
U Scale, V Scale	Lets you specify the number of times a texture is repeated over a surface. You can use a non-integer value, such as 2.35.

Property	Description
Rotation	Rotates the texture over a surface.

Using the Shadow Matte Shader

The Shadow Matte shader lets you capture shadows created by using 3D objects as proxies. This creates the illusion of shadows cast from objects or image elements onto 2D images where none exist. The resulting output using this shader is a black and white image that shows the shadows in black, and everything else in white.

For example, if you have an image of a cityscape, and you want to have a red balloon fly overhead, casting a shadow as it travels, you could create your balloon using a sphere, add a light source to illuminate it, and then add a planar surface parallel to the ground to receive the shadow. You would then select the Shadow Matte shader to use on the plane, and set the Blending Mode to Multiply to mask out the plane, except for the shadow. You can then animate the sphere, and the shadow will follow it.

To set Shadow Matte parameters:

1. Select the Materials tab.
2. Select a layer from the Layer Editor, and then select the Shadow Matte shader.
3. Set parameters to control how light or dark the shadow should look, and to invert the shadow.

Parameter	Description
Multiplier	Uniformly scales the lightness or darkness of the shadow up or down.
Invert Shadow	Flips the shadow inside out to reveal the background in the shadow regions and mask the illuminated areas.

Working with Lights

Light is a basic geometric 3D scene element. Each light in a scene contributes to how a scene is illuminated. Lights affect the way all object's surfaces appear

in the rendered scene. You can dramatically change the nature and mood of your compositions by modifying lights and their properties.

NOTE You can animate lighting parameters using keyframes and by applying expressions—see Animation on page 659 and Expressions on page 719.

Adding Lights to a Scene

You can add one or more lights to your scene and set their properties to achieve a variety of lighting effects. There are two ways to add lights to a scene: the Lights tab in the Reaction node or the Tools tab.

When you add a light to Reaction, you must be in the Reaction Group Schematic to view the light(s) in your scene. The light you add is positioned at 0, 0, 0, so you may need to zoom out to view the light icon.

You can also set the light type and parameters, as well as parent a light to an axis, and animate its properties.

NOTE The light icon is visible only if you have selected Icons in the Render tab of Reaction.

To add a light from the Lights tab:

1. Select the Lights tab.
2. Under the Light list, click Add.
   A light is added to the Group Schematic and the Lights list.
3. To view the light, double-click the Reaction node in the Schematic view.
   The Group Schematic appears.

To add a light from the Tools tab:

1. Do one of the following to display the Group Schematic:
   - Double-click the Reaction node in the Schematic view.
   - Right-click the Reaction node and select Edit Group.
2. From the Tools tab, select the Light tool from the Reaction folder and drag it to Schematic.

Choosing the Type of Light

When you add a light to your scene, an Omni light is added by default. You can change the light type and its properties, and add as many lights to your scene as needed.

You can define physical properties for your light, depending on the type of light you selected. There are three types of lights available.

Omni Simulates a point light source that casts rays in all directions from the position of the light. Similar to a light bulb, where the light rays emanate in all directions from the bulb.

Spotlight Casts rays in a cone, or four-sided pyramid, simulating a real spotlight. This is useful for lighting a specific object or area and creating sharp-edged shadows.

Directional Casts rays in a single direction, from a geometric area and is useful for creating soft-edged shadows with both an umbra (a full shadow where an object blocks all rays from a light) and a penumbra (a partial shadow where an object blocks some of the rays).

To set attributes for a Spotlight:

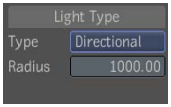
1. Select the Lights tab.
2. Under Light Type, select the Spotlight light type.

3. Set the Spotlight properties.

Properties	Description
Shape	The shape of the light can be a circle or rectangle.
Hot Spot Angle	The angle where the solid cone of full intensity light ends and a gradual fading begins.
Cutoff Angle	The angle at which the light ends.
Aspect	The aspect ratio of the spotlight shape.

To set properties for a directional light:

1. Select the Lights tab.
2. Under Light Type, select the Directional light type.



3. Adjust the Radius field to set the radius of the light. This value is also used to delimit the size of the shadow map.

Setting Basic Lighting Properties

You can set basic lighting properties for all light types.

To set basic lighting properties:

➤ On the Lights tab, under Basics, enter values to define the light.

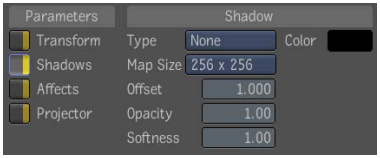
Properties	Description
Color	Sets the color for the light. Click the color pot to display the color picker, choose a color, and click Set—see Working with the Color Picker on page 155.
Intensity	Sets the amount of light emitted by the source light.
Decay	Sets the light's intensity to diminish gradually using a linear or quadratic function.
1/2 Distance	Represents the distance at which the light's intensity is reduced by half.
Time Dilation	Modifies the motion blur that affects lights animated in your scene. You can set the Time Dilation factor to give the light the appearance of moving at a different speed than is set for Motion Blur in the Render tab.

Setting Shadow Properties

You can enable shadows and set their properties and parameters to achieve the results that you want.

To enable shadows and set their properties:

➤ On the Lights tab, under Parameters, select Shadows.



Property	Description
Type	Sets the type of shadow created. There are four types: ■ Normal: The default shadow. ■ Mid-Depth: Uses a mid-depth z-buffer to compute a shadow that usually yields fewer artifacts. ■ Deep: Allows a transparent object to color light as it passes through. ■ None: The light does not cast a shadow.
Map Size	Select the map size. The default is 256 x 256.
Offset	Experiment with self-shadowing effects. The default value is 1.00 and is intended to reduce self-shadowing artifacts.
Opacity	Set the transparency of the shadows. By default shadows are opaque.
Softness	Set the softness for the shadow.
Color	Click the color pot to display the color picker, choose a color, and click Set—see Working with the Color Picker on page 155.

Determining How a Light Affects Shading

You can choose the shading properties a light affects. By default, a light affects the ambient, diffuse, and specular components of shading.

To choose the shading properties that a light affects:

1. On the Lights tab, under Parameters, select the Affects option.
2. Under Light Affects, select the shading properties you want the light to affect.

Adding a Projector Texture to a Light

A projector texture is created when you connect an image to a light node in the Reaction Group Schematic. The Projector Texture is applied in the same way as a material texture.

To add a projector texture to a light:

➤ On the Lights tab, under Parameters, select Projector.

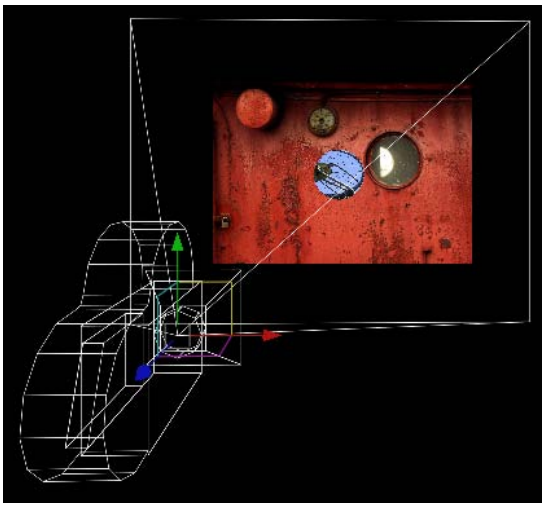
Property	Description
Multiplier	Uniformly scales the brightness of the image texture up or down. This is useful for setting the amount that a given texture affects the object's overall look. The default value is 1, but you can overdrive the brightness to achieve specific effects.
Filtering	Sets the quality of the texture image. When a texture is stretched onto an object, it may, depending on the camera position, lose some of its crispness. The rendered can compensate for this, but it may slow the process. Filtering options include: ■ **Nearest** A box filter and the fastest way to resample an image since it only samples a single pixel of the input image to determine the value of a given pixel in the result image. Produces significant amount of aliasing.

Property	Description
	■ **Bilinear** A separable triangular filter that takes into account more area when resampling. ■ **Mitchell** Considered one of the best magnification filters for images; has a good balance between ringing and sharpness. ■ **Gaussian** Good magnification and magnification filter with no ringing, but introduces noticeable softness to the result image. ■ **Jinc 2** Offers better sharpness than the Gaussian filter, but at the expense of ringing. Similar to the sinc filter, but with better isotropic qualities, less ringing, and same sharpening and anti-aliasing. ■ **Jinc 3** Offers better sharpness than the Gaussian and Jinc 2 filters but at the expense of even more ringing. Similar to the Lanczos filter but with better isotropic qualities, less ringing, and same sharpening and anti-aliasing.
AA Factor	Lets you adjust the trade-off between anti-aliasing and blurriness.
Tiling X, Tiling Y	Set the repeat mode. Choose from; Transparent, Edge, Repeat, and Mirror.
U Offset,V Offset	Translates the texture in U and/or V.
U Scale, V Scale	Lets you specify the number of times a texture is repeated over a surface. You can use a non-integer value, such as 2.35.
Rotation	Rotates the texture over a surface.

Working with Cameras

The camera in Reaction is analogous to a physical camera in the real world. Each Reaction node has a camera that outputs a result; this is the render camera. The camera through which you are currently viewing through is displayed in the upper-left corner of the Player.

NOTE When using the orthographic camera, you cannot set certain basic camera properties.



Adding a Camera to Your Scene

You can add as many cameras to a scene as you like, and you can view and render your scene from any camera. There are two ways to add cameras to a scene: through the Cameras tab in a Reaction node or through the Tools tab.

Once you've added cameras, you can indicate which camera(s) to use as the render, or active, camera(s). You can designate any number of cameras as active. Active cameras contribute to the composition's output, and is indicated by the white square under the “A” (active) column of the Cameras tab. You can also set the visibility of each camera.

NOTE The camera icon is visible only if you have selected Icons in the Render tab of Reaction.

To add a camera from the Cameras tab:

1. In the Schematic view, select the Reaction node.
2. Select the Cameras tab.
3. At the bottom of the Cameras list, click Add.

A camera is added to the Cameras list.

To add a camera from the Tools tab:

1. In the Schematic view, display the Group Schematic by double-clicking the Reaction node or right-clicking the Reaction node and selecting Edit Group.
2. From the Tools tab, select Camera from the Reaction folder and drag it to the Group Schematic.

   A camera is added to the Group Schematic and the Cameras list.

NOTE In the Group Schematic, you can parent a camera to an axis, animate its properties using keyframes or expressions, as well as apply tracking and stabilization.

To set the render camera(s):

1. In the Schematic view, select the Reaction node.
2. Select the Cameras tab.
3. From the Cameras list, click the white square under the A (active) column beside the camera(s) you want to designate as the render camera(s).

To set the visibility of a camera:

1. In the Schematic view, select the Reaction node.
2. Select the Cameras tab.
3. From the Cameras list, click the square beside the camera and under the V (visible) column.

Setting Camera Properties

Once you've selected a camera, you can set the properties for each camera in your scene.

To set a camera's properties:

1. In the Schematic view, select the Reaction node.
2. Select the Cameras tab.

3 From the Cameras list, select a camera.

4 In the Cameras tab, you can set the properties for the selected camera.

Parameter	Description
Type	■ **Perspective (default)** This projection simulates a real physical camera. ■ **Orthographic** With this projection, all camera rays are parallel, and objects do not appear to change size as they change distance from the camera. Field of view settings and depth of field settings available from the Render tab have no effect with this projection.
Focal Length	Sets the distance for the camera's focal point. As the focal distance increases, the field of view decreases, and vice versa.
Near Plane	Sets the minimum viewable distance from the camera. By default, the near clipping plane is close to the camera. Setting the near plane farther back will hide objects very close to the camera.
Far Plane	Sets the maximum distance from the camera. By default, the far clipping plane is very far away, so that you can see the entire scene. Setting the far plane closer to the camera will hide objects farther away.
Time Dilation	Modifies the motion blur that affects layers animated in the scene. You can give the layer the appearance of moving at a different speed than is set for Motion Blur in the Render tab.
Angle of View	Lets you frame the scene, making objects appear larger or smaller in the frame.
Input Stream	The input stream selector controls which streams of the Reaction sources are used when rendering through the given camera. You can specify an explicit stream index (Manual mode) or let the ordering of the camera in the camera list control the selection of the stream (Match Out).
Film Back	■ **Film Gate** The format of the film gate. ■ **Horizontal/Vertical Aperture** The height and width of the camera's aperture or film back, measured in inches. The Camera Aperture attribute determines the relationship between the Focal Length attribute and the Angle of View attribute. The default values are 1.417 and 0.945.

Parameter	Description
	■ **Film Aspect Ratio** When the aspect ratio of the film is modified, the horizontal camera aperture is modified. X = Y multiplied by the aspect ratio. The ratio of the camera aperture width divided by the camera aperture height. When changed, it dynamically adjusts the camera aperture width. ■ **Lens Squeeze Ratio** The amount the camera's lens compresses the image horizontally. Most cameras do not compress the image they record, and their Lens Squeeze Ratio is 1. Some cameras (for example, anamorphic cameras), however, compress the image horizontally to record a large aspect ratio (wide) image onto a square area on film. The default value is 1. ■ **Fit Resolution Gate** Controls the size of the resolution gate relative to the film gate. If the resolution gate and the film gate have the same aspect ratio, then the Film Fit setting has no effect. ■ **Film Fit Offset** Offsets the resolution gate relative to the film gate either vertically (if Film Fit is Horizontal) or horizontally (if Film Fit is Vertical). Film Fit Offset has no effect if Film Fit is Fill or Overscan. Film Fit Offset is measured in inches. The default setting is 0. ■ **X & Y Film Offset** Vertically and horizontally offsets the resolution gate and the film gate relative to the scene. Changing the X and Y Film Offset produces a two-dimensional track. Film Offset is measured in inches. The default setting is 0.
Position	Translates the selected camera about the X, Y, or Z axis.
Rotation	Rotates the selected camera about the selected axis and changes its orientation. Angles of rotation are measured in degrees.
Pivot	Moves the selected camera's center along the X, Y, or Z axis.

Accessing Cameras and Camera Views

You can see your scene through different cameras and from different points of view. To see different views, you can use the predefined orthographic views or the perspective view. You can also dolly, truck, or zoom those views without affecting of the contents in the scene, including the render camera.

NOTE When using any camera, other than the render camera, zooming and panning in the Player is equivalent to dollying and trucking the camera.

The View From menu lets you access each camera and camera view, and is available from both the hardware and software renderer.

To access the camera(s):

➤ Right-click the Player and select View From > Camera (**Shift + 1**). If there is more than one camera, each one is listed. To cycle through each camera, press **Shift + 1**.

To access the camera views:

➤ Right-click the Player, select View From and one of the following: Perspective (**Shift + 2**), Front (**Shift + 3**), Back, Left (**Shift + 4**), Right, Top (**Shift + 5**), or Bottom.

Transforming Cameras

You can transform cameras interactively in the Player or set values in the transformation fields in the Cameras tab—see Transforming Objects on page 288.

NOTE The camera is initially positioned in *Z* according to the composition's format.

NOTE Interactive transformations are applied using the camera's local axis, but its position is recorded using global coordinates.

Dollying, Trucking, and Orbiting the Camera

You can dolly, truck, and orbit any camera to view your scene from different angles.

NOTE Camera movements are not the same as zooming and panning the Player. For example, if you need to view the pixel output of Reaction up close, you should zoom in. But if you want to move the camera itself, you should dolly.

To dolly the camera:

➤ Hold down the **E** key and drag up to move the camera into the scene. Drag down to move the camera out of the scene.

To truck the camera:

➤ Hold down the **W** key and drag up, down, left, or right to position the camera.

To orbit the camera:

➤ Hold down the **Q** key and drag in any direction to position the camera.

Modifying Motion Blur for the Render Camera(s)

You can modify the motion blur applied to the render camera(s) in your composition. This gives the appearance of movement in the scene relative to the camera—see Adding Motion Blur and Depth of Field to a Scene on page 292.

To modify the motion blur:

1. Select the Cameras tab.
2. From the Cameras list, select the render camera(s).
3. Under Motion Blur, modify the Time Dilation. A higher value increases the blur effect giving the impression of faster motion, and vice versa.

Resetting the Camera

You can easily reset the camera to its default.

To reset the camera:

1. In the Schematic view, select the Reaction node.
2. Do one of the following:
   - Press **Shift** + **Home** to reset the camera in the current Player.
   - Right-click the Player and select Reset Camera.

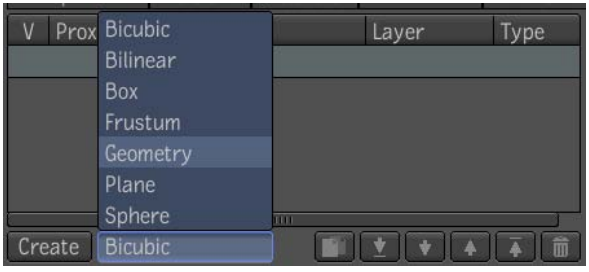
Camera Mapping

Camera mapping lets you add depth and perspective to a matte painting or rendered image by allowing it to be projected onto geometry from the point of view of the camera. In this way, you can add the illusion of 3D movement to a 2D scene. This facilitates the creation of virtual set extensions, and is also used as part of the process of converting 2D material to stereoscopic imagery.

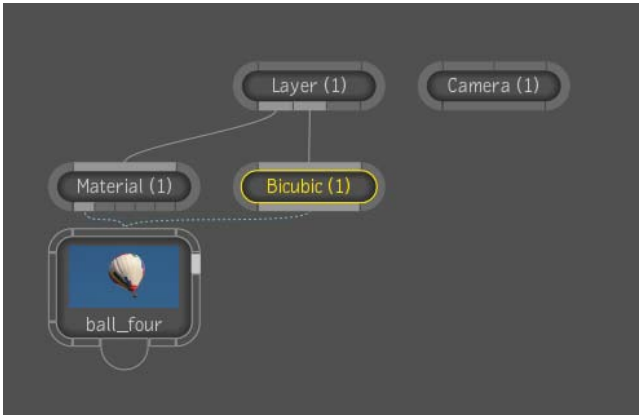
Using the Camera Mapping tool, which you can place between the geometry generator and the Layer node, you can generate UV coordinates. UV coordinates are 2D coordinates that you can apply to an object; they are generated by the camera connected to the Camera Mapping tool. UV coordinates are affected by the camera's parameters, such as the field of view, distance, and so on.

To use the Camera Mapping tool:

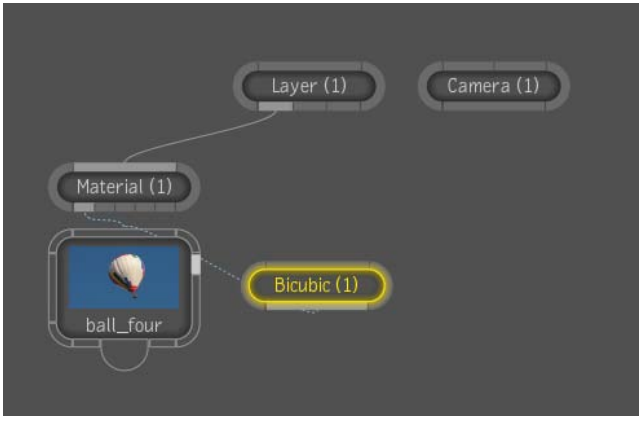
1. Create a new composition.
2. From the Tools tab, select Reaction and drag it to the Schematic view
3. Select a layer type from the Layer type menu and click Create to add a new layer.



4. Open the File browser and drag the image you want projected into Schematic and attach it to the new layer you created.
5. Display the Group Schematic by double-clicking the Reaction node or right-clicking the Reaction node and selecting Edit Group.



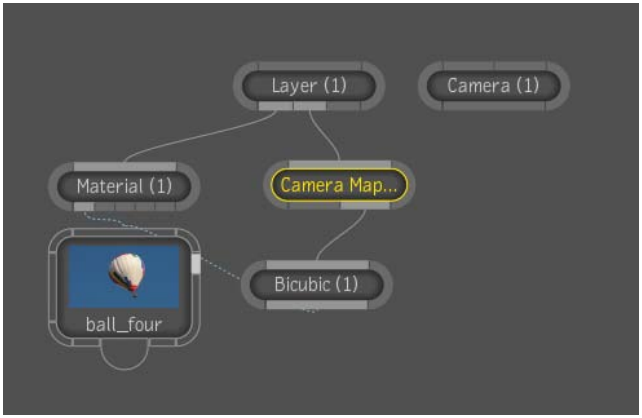
6 Temporarily disconnect the layer type (bicubic in this case) node from the sub-graph.



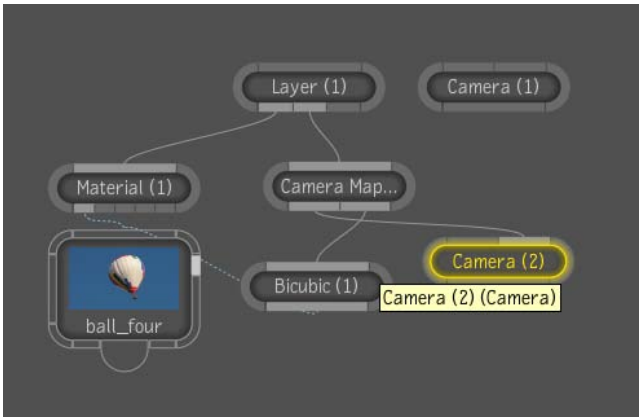
7 From the Tools tab, select Camera Mapping from the Reaction folder and drag it to the Group Schematic.

8 Connect the Layer Geometry output to the Camera Mapping input.

9 Connect the Camera Mapping Geometry output to the bicubic node.



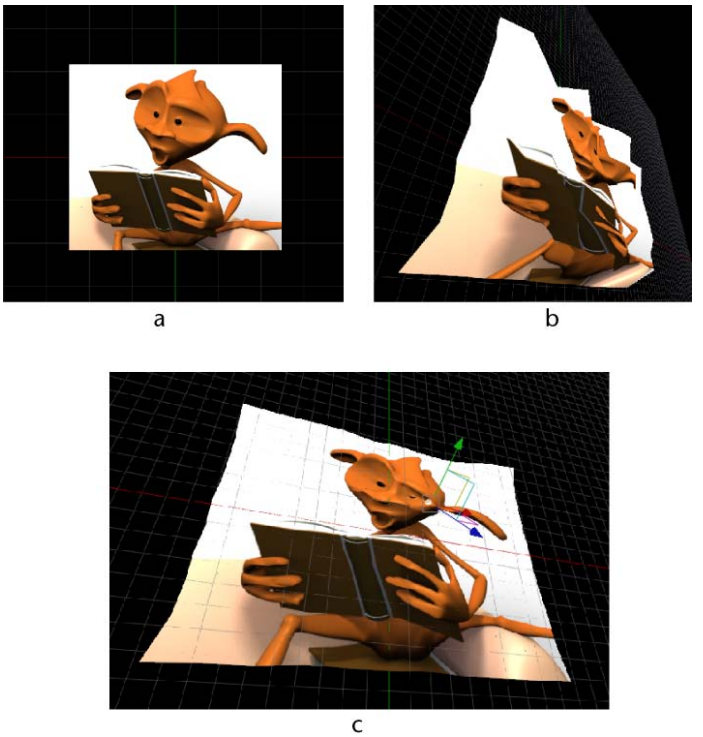
10 From the reaction folder, drag a second camera to the sub graph and connect its camera output to the Camera Mapping node's Camera output.



11 Adjust the Camera's Film Back, Position, Rotation and Pivot parameters.

3D Displacement

The 3D Displacement tool lets you add depth to meshes created in or imported into Composite. The pixel values in an image are used to displace vertices. This allows the illusion of perspective to be maintained as the camera is moved in 3D space. You can also create a displacement that converges on an object, such as a light. Once created, you can blur the displacement, as well as set the tiling.



(a) Image before displacement. (b) Image after displacement. (c) Displacement targeted towards and converging on an object. In this case, a light.

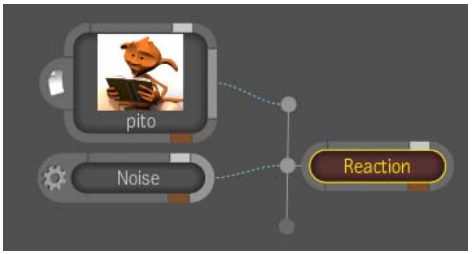
NOTE The 3D Displacement tool can only perform displacement on images with sufficient geometry. If the image you are using does not contain sufficient geometry, you can increase it by adjusting the width and height of segments in the Surfaces tab of Reaction.

To use the 3D Displacement tool:

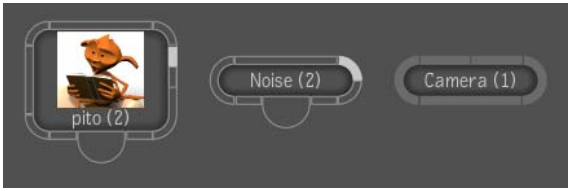
1. Select File > New or press **Ctrl + N** (for Windows and Linux) or **Cmd + N** (Mac OS) to start a new composition.
2. From the Tools tab, select Camera from the Reaction folder and drag it to the Schematic view.
3. You will need two sources, one for the image to be displaced and another for the image that will do the displacing. Right-click the Reaction node and select Add Source. Repeat to add another source.

4 Select the Reaction node. In the tool UI, select the Render tab and click Z-Buffer—see Enabling Z-Buffer Effects in Your Scene on page 292.

5 Add the two images you want to use to the Schematic view, connecting one to each source. In the following example, the Noise image generator is used to displace the character.

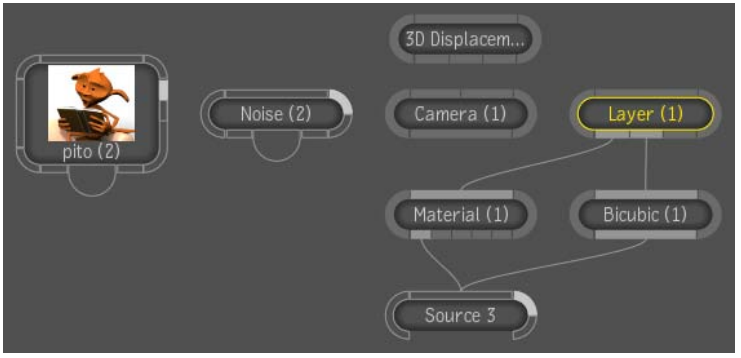


6 Display the Group Schematic by double-clicking the Reaction node or right-clicking the Reaction node and selecting Edit Group.

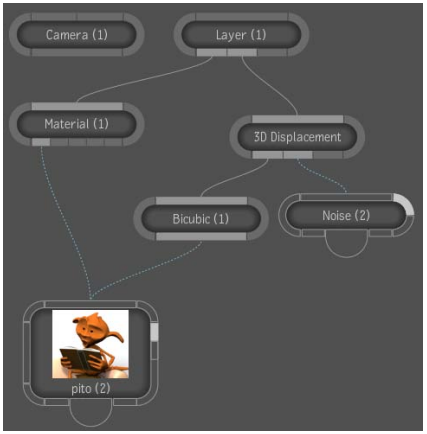


7 From the Tools tab, select 3D Displacement from the Reaction folder and drag it to the Group Schematic.

8 In the Layer Editor, click Create to add a new layer.



9 Delete the Source node and arrange the nodes as follows:



10 Select the 3D Displacement node. In the tool UI, you can adjust the following parameters:

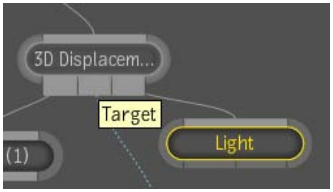
Parameter	Description
Amount	Amount of displacement in the image. Positive values make the displacement protrude and negative values invert the displacement. **NOTE** The displacement occurs in Reaction units (Amount times the displacement image values minus the Offset).
Offset	The value that is subtracted from the displacement image values before it is interpreted as a displacement. It is used to determine the value of the displacement image that yields no displacement.
Type	■ **Surface** Orients the displacement towards the surface normal. ■ **Toward Target** Orients the displacement towards a target which can be a light, locator, axis, camera, layer, or 3D object. ■ **RGB > XYZ** Specifies the displacement by using the explicit XYZ components of the displacement in the RGB channels of the image.

Parameter	Description
Channel	Set the channel for displacement: Luma, Red, Green, Blue, Alpha.
Convergence	Used with the Toward Target type. The displacement converges or is parallel to the target.
X Radius, Y Radius	Set the amount of blur to apply in the X or Y direction.
Link	Couple the X and Y Radius so that when you change the X Radius or Y Radius, the other changes in the same proportion.
Tiling X, Tiling Y	Set the repeat mode: Transparent, Edge, Repeat, and Mirror.

11 To make the displacement point towards a target, set the Type to Toward Target. Set Convergence parameter; the displacement can converge on the target or be parallel to it.

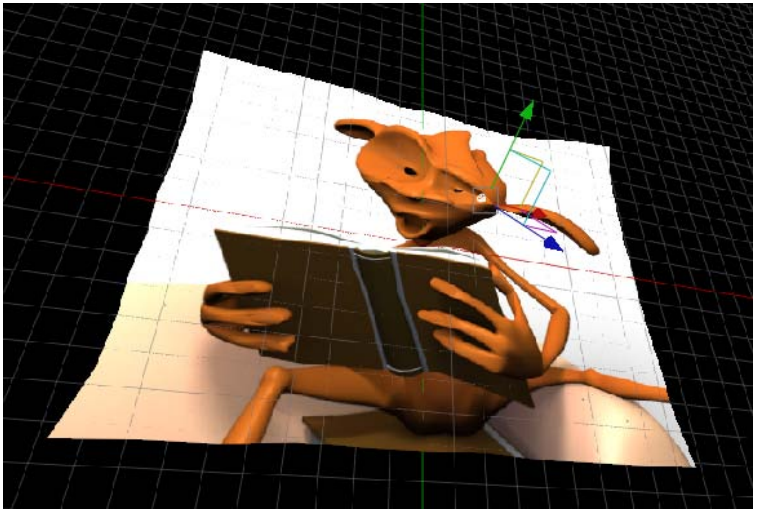
12 From the Tools tab, select the Reaction folder and drag one of the following to the Group Schematic: Axis, Camera, Layer, Light, or Locator.

13 Connect the target to the Target tab of the 3D Displacement node. In the following example, a light was added as the target.



14 Select the Light node. In the Lights tab, adjust the light to your liking—see Working with Lights on page 268.

15 Select the 3D Displacement node and adjust the Amount until you have the desired result.



The image is starting to displace towards the light object.

Transforming Objects

You can transform an individual layer, axis, light, or camera along a specific axis to achieve the effects you want. You can transform objects interactively in the Player, or by setting values in the transformation fields in the Reaction tool UI.

NOTE When transforming lights, you may need to zoom out to view a light's icon. Also, interactive transformations are applied using a light's local axis, but its position is recorded using global coordinates.

To transform objects interactively:

1. In the Schematic view, display the Group Schematic by double-clicking the Reaction node or right-clicking the Reaction node and selecting Edit Group.
2. Select a layer, axis, light, or camera by doing one of the following:
   - Click an object in the Group Schematic.
   - In the Reaction tool UI, select the appropriate tab and select an item from the Layer Editor, Axes, Lights, or Cameras list.

   The Reaction tool UI displays the tab associated with the selected object.

3 Click a transform button.

Click:	To:
	Translate an object along the selected axis. When you move an object, its coordinates are displayed and update dynamically.
	Rotate an object about the selected axis and change its orientation. Angles of rotation are measured in degrees.
	Scale an object in X, Y, or Z according to the selected axis. An object is scaled from its center. Scaling uses a multiplication factor. Negative values yield an inverted object. **NOTE** When a layer is created, its scale is automatically set to 1.0. **NOTE** Cameras cannot be scaled.

To transform objects using the tabs:

1 In the Reaction tool UI, select the tab of the object you want to transform, and then select an item from the Layer Editor, Axes, Lights, or Cameras list.

NOTE If you selected a light, select Transform in the Lights tab (under Parameters) to display the fields.

2 Transform objects by adjusting the Position, Rotation, and Scale values (X, Y, and Z).

NOTE To scale a layer, light, or axis uniformly, click the Link button below the Scale fields.

NOTE Cameras cannot be scaled.

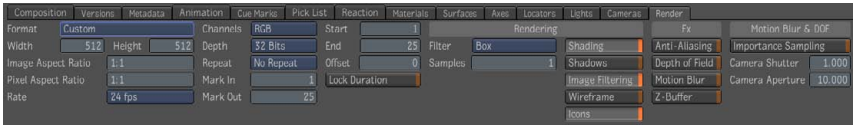
3 Move the object's center by adjusting the Pivot values (X, Y, and Z).

Reaction Rendering Effects and Output Results

Because Reaction provides a 3D environment, it has its own rendering and output options that you can use to specify an output format, set certain rendering features to quickly render draft results, or set rendering effects such as anti-aliasing or motion blur and depth of field.

Setting the Reaction Composition Format

The Reaction Render tab contains the same composition format options available from the Composition tool UI.



By default, the Reaction tool inherits the format settings specified for the project. You can set them as desired for your Reaction output.

Setting Rendering Options

Depending on whether you are using the hardware or software renderer, you can set some rendering options and effects. When you use the software renderer, you can set the Depth of Field and Motion Blur effects, and specify the number of samples to take, and select a filter for anti-aliasing.

To set Rendering and Effects Options:

1. With the cursor over the Player, press the tilde key (~) or middle-click to display the Gate UI. Swipe through Player Options.
2. Under Reaction, select Interactivity to use the hardware renderer to preview results in Reaction. Deselect Interactivity if you want to use the software renderer.

   NOTE The hardware preview in Reaction does not affect the Reaction output to any tools downstream, and does not affect the published results.

3. In the Reaction UI, select the Render tab.

4 Under Rendering, set rendering options.

Parameter	Description
Filter	Applies filter effects on your composition. Choices are listed in order of speed and quality: Box, Conic, Max, Quadratic, Narrow Gaussian, Medium Gaussian, Wide Gaussian, Cubic, Cook, and Mit-Net Noch. The Box filter is ultra-fast, but low quality. The Gaussian filters give more blur. The choice you make also depends on the number of samples used.
Samples	Sets the number of samples. A larger sample gives higher quality but slower speed. This is used with the software renderer only.
Shading	Turns on/off shading. You may want to disable shading for generating a draft result.
Shadows	Turns on/off shadows.
Image Filtering	Applies image filtering.
Wireframe	Enables surfaces to render in wireframe.
Icons	Displays light, camera, and axis icons. You must also enable Interactivity in the Player options to view the icons.

5 Under Fx, set the effects options.

NOTE You must have software rendering enabled. To enable software rendering, deselect Interactivity in the Player options.

Parameter	Description
Anti-aliasing	Smooths the edges of rounded surfaces and diagonal lines.
Depth of Field	Includes DOF in your scene. This effect simulates a plane of maximum sharpness and blurs objects close to or beyond this plane.
Motion Blur	Adds motion blur to your scene.
Z-Buffer	Uses the Z-buffer in your composite. The Z-buffer works with hardware and software rendering.

Adding Motion Blur and Depth of Field to a Scene

When the Motion Blur and Depth of Field (DOF) effects options are enabled, you can set parameters to define how they will appear in your scene.

NOTE The software renderer must be enabled to apply Motion Blur and Depth of Field effects.

To add Motion Blur to your scene:

1. Under Fx, select Motion Blur.
2. Under Rendering, enter the desired number of samples. A greater number of samples gives a more realistic effect, but requires more processing time.
3. Set a value for Camera Shutter.

To add depth-of field to your scene:

1. Under Fx, select Depth of Field.
2. Set a value for Camera Aperture.

Property:	Description:
Importance Sampling	Used with motion blur. The software calculates which sample should be considered most important and makes it appear brighter.
Camera Shutter	Used with motion blur. Sets the length of time the shutter is open. A larger number yields a slower shutter speed and a greater amount of motion blur.
Camera Aperture	Used with DOF. Sets the size of the camera "lens" opening. Increasing the size of the aperture increases the zone of sharpness in front of and behind the plane that the lens is focused on.

Enabling Z-Buffer Effects in Your Scene

You can enable the Z-buffer for your scene to create realistic spatial depth effects. The Z-buffer calculates the spatial depth for each pixel in an image to define which pixels are hidden by others.

To enable the Z-buffer:

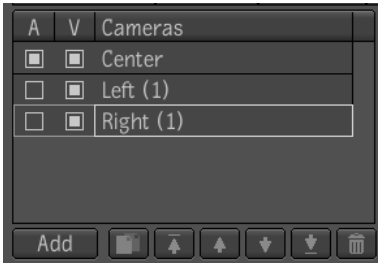
➤ Under Fx, select Z-buffer.

Setting up a Stereo Camera Rig

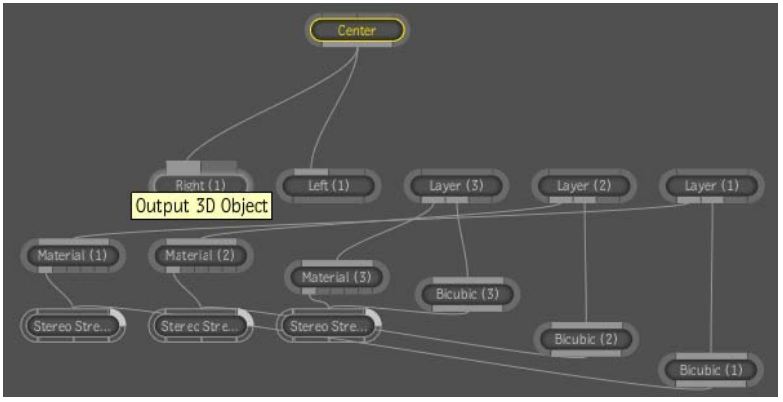
You can use Reaction to render a stereo scene from two cameras, or vastly different viewpoints from an arbitrary number of cameras.

To set up a stereo camera rig in Reaction:

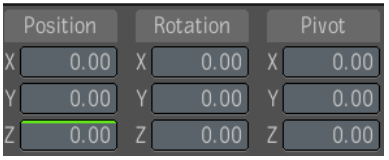
1. Select the Reaction node in the Schematic view and then select the Cameras tab.
2. Create three cameras and Rename them: Center, Left, and Right.



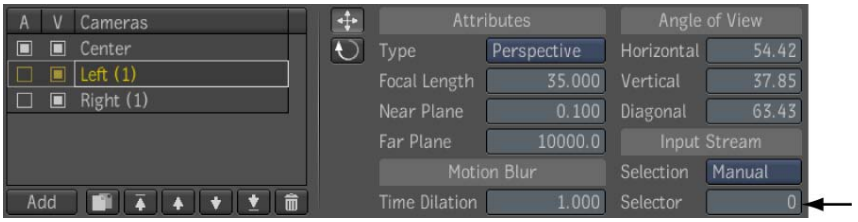
3. Make the Center camera parent of the Left & Right cameras by double-clicking the Reaction node in the schematic to access the Reaction sub-schematic then connect the Center camera 3D Object input to the Left and Right cameras Output 3D object output.



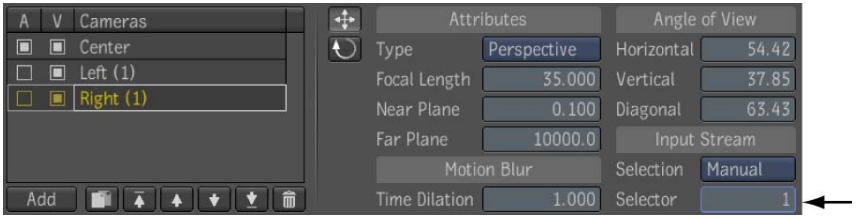
4 From the Tool UI, set the Left & Right cameras Translation Z parameter to 0.



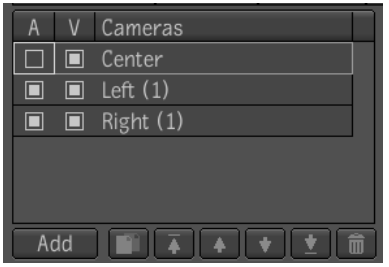
5 From the Tool UI, set the Left camera Input Stream parameter to 0.



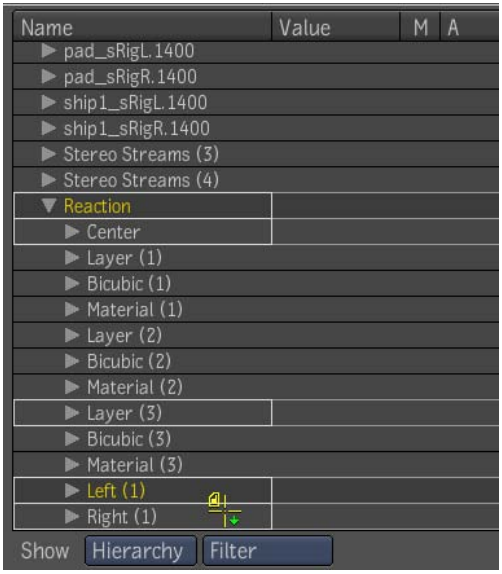
6 From the Tool UI, set the Right camera Input Stream parameter to 1.



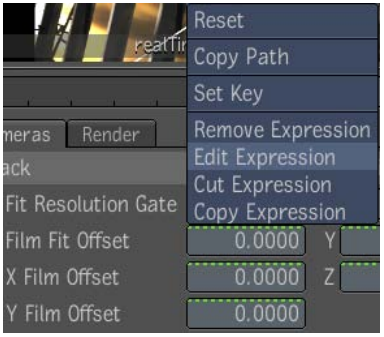
7 From the Reaction Cameras list, set the Left & Right cameras to active by enabling the square button under the A column. Set the Center camera as inactive by disabling the square button under the A column.



8 From the Composition Browser, drag & drop the Left camera folder on the Right camera folder to link them together with expressions.



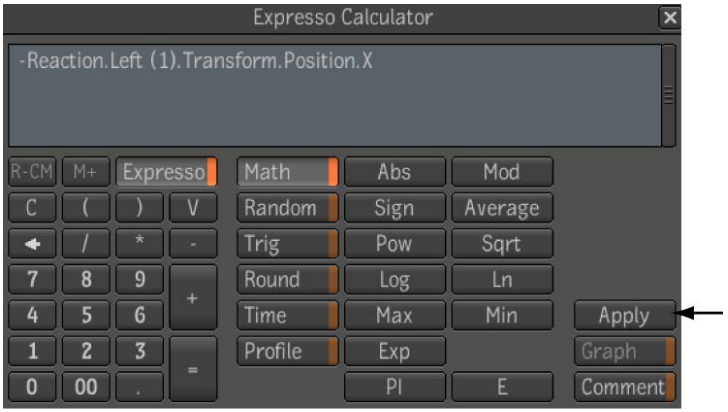
9 From the Tool UI, right-click on the Right camera X Film Offset parameter and choose Edit Expression option.



10 In the text field, type a minus sign in front of the expression and press Apply.

11 From the Tool UI, right-click on the Right camera X Translation parameter and choose Edit Expression option.

12 In the text field, type a minus sign in front of the expression and press Apply.



The stereo rig is ready to use. You can adjust left and right eye convergence.

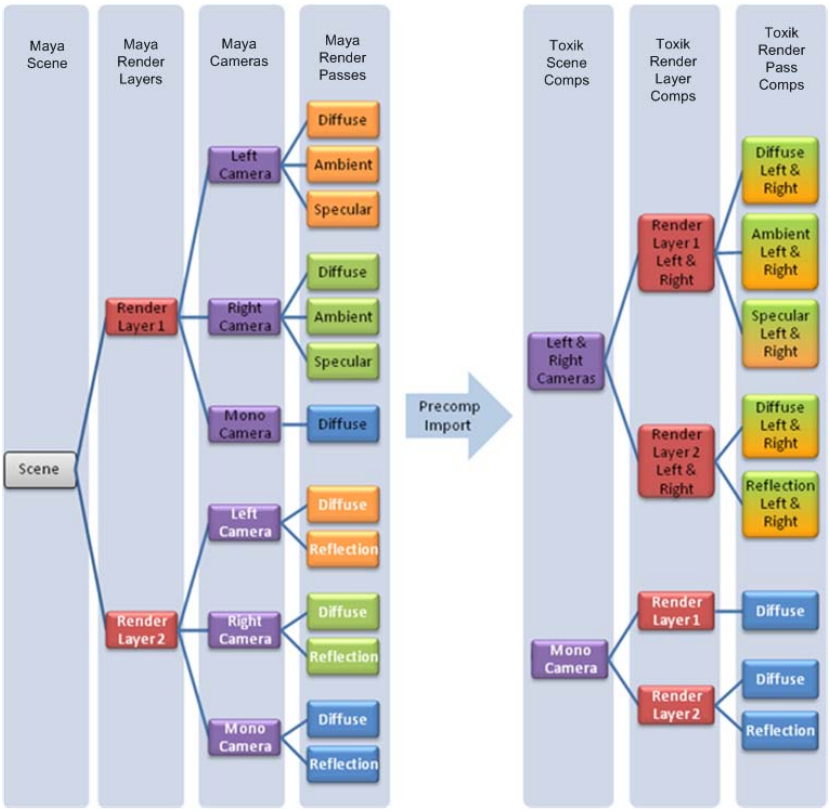
Pre-Compositing

13

Topics in this chapter:

About Pre-Compositing

Composite 2011 allows you to import a render layers and passes setup exported as a pre-comp (*.precomp*) file by 3D applications that support this file format, such as Autodesk Maya and Autodesk 3ds Max. Once imported into Composite, a set of compositions is automatically created, representing the render layer setup. For example, the left side of the following illustration shows the structure of a Maya scene and its render layers, cameras, and render passes. The right side shows the corresponding compositions that are created after it has been imported into Composite.



3D applications are render layer centric because it suits 3D artists. However, once you import a pre-comp file into Composite, it becomes camera centric because it is more suitable for the 2D artist. Now you can perform any compositing work needed on the pre-comp file, and go back and forth between Composite and the 3D application.

The Composite pre-compositing interoperability supports having a different set of render passes for each render layer. In our example, Render Layer 1 has more passes than Render Layer 2. Render Layer 1 only has an Ambient pass and Render Layer 2 only has a Reflection pass.

For a given render layer, the set of render passes produced by each camera can be different in a pre-compositing setup. For example, Maya cameras can decide

whether to contribute to a pass or not (compare the Mono camera against the Right and Left cameras on Render Layer 1 in our example).

- A render pass composition imports render pass file sequences; it's a footage composition.

- A render layer composition assembles the render passes for a given render layer. A render layer composition is basically a representation of the 3D application shading/rendering network that produced the set of passes for a layer.

- A scene composition assembles (stacks) the render layers in the scene for a specific camera view point.

Sets or pairs of cameras are also supported by the pre-compositing workflow (for stereoscopy purposes).

- Render passes produced by cameras of a set (Right and Left cameras in our example) are imported in the same render pass composition in Composite.

- Dependency graphs of render passes produced by camera pairs are also created in the same render layer composition.

- Finally, render layers produced by camera pairs are stacked in the same scene composition.

This classification facilitates stereoscopic compositing once in Composite. The dependency graph of the left camera can be manipulated next to the dependency graph of the right camera in the same composition, allowing expressions to be set between the two dependency graphs. Or, if you import in Multi-Stream mode, images from the left and right camera views can be merged and processed by a multi-stream dependency graph—see Working with Stereoscopic Compositions on page 319.

TIP Pre-comp files are actually Python modules that you can read and edit with any text editor. Just be sure to conform to the Python syntax if you choose to edit your pre-comp file. Otherwise, Composite will encounter an error and display a dialog box highlighting the problem.

Pre-Compositing Import

You can either import render passes from a pre-comp file generated by any 3D application or import Maya passes directly from the Maya (*.ma* or *.mb*) file.

A Maya pre-comp file contains the same information as the corresponding *.ma* or *.mb* file of the 3D scene. However, pre-comp files are faster to import into Composite and more stable to use. An advantage of using pre-comp files is that they are self-contained. That is, you don't need to have Maya installed and licensed on your system like you do for importing Maya files. Before importing a Maya file, see To Import an FBX File on page 324.

To import a pre-comp or Maya file:

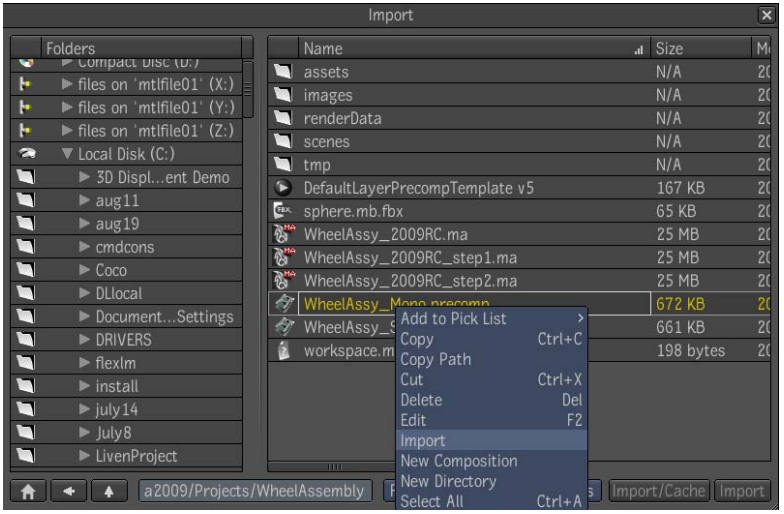
1. Select File > Import or press **Ctrl** + **I**(for Windows and Linux) or **Cmd** + **I**(for Mac OS).

   The File Browser is displayed.

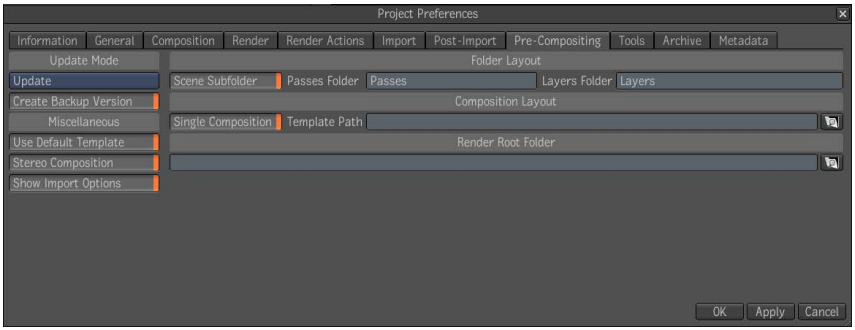
2. Locate the folder that contains the pre-comp file.

3. Right-click the pre-comp (*.precomp*) file and select Import or press the Import button.

NOTE Pre-comp and Maya files can only be imported in Composite one at a time.



The Script Output floating window displays the progress of the import, and the Pre-Compositing Import Options dialog box displays. See Viewing Output and Error Messages on page 871.

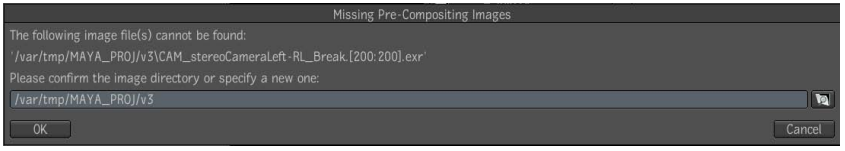


4 Set parameters for the pre-comp file—see The Pre-Compositing Import Options on page 306.

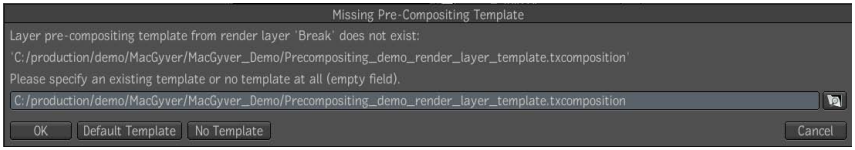
If images are missing, you are prompted to locate the folder that contains the images.

NOTE The Browse button does not work when importing Maya files, because the import is performed in a background process.

NOTE For Maya files, by default, Maya expects the render files to be in the default location for the Maya project. Place them there to avoid having to input the location or add the correct path in the dialog box. On Windows, the location is C:/My Documents/maya/projects/default. You will need to locate the My Documents folder as it could at a different location, depending on where it was installed. On Linux and Mac OS, the location is: ${HOME}/maya/projects/default.



If Composite cannot find a pre-compositing template referenced by the pre-comp file, the following dialog box is displayed:



5 Do one of the following:

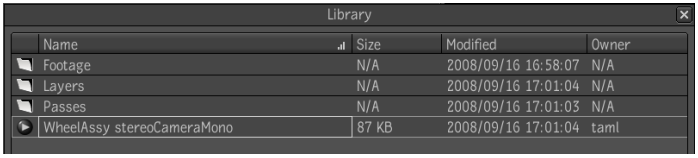
- If you have a template you want to use, locate it.
- If you want to use the default template provided by Composite, click Default Template.

  NOTE The Browse button does not work when importing Maya files, because the import is performed in a background process.

- If you do not want to use a template, click No Template.

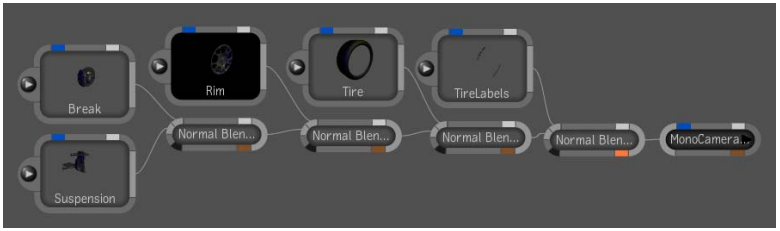
For more information, see Using Pre-Compositing Templates on page 310.

The render layers and passes are imported. The scene composition, named after the scene anchor and camera(s) anchor, is created in the folder you specified during import. This folder contains a Layers folder and a Passes folder, unless you specified a different name during import. Render layer compositions are named after the scene anchor, the camera(s) anchor, and the layer anchor. Render pass compositions are named after the scene anchor, the camera(s) anchor, the layer anchor, and the pass anchor.



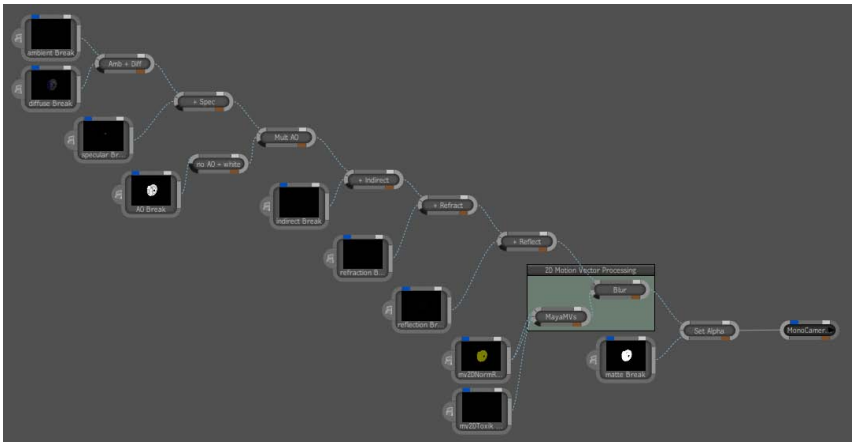
6 When the imported composition has opened.

The composition and associated render layer compositions are opened, and the chain of Blend & Comp nodes in the scene composition matches the order and blend modes in the render layers.

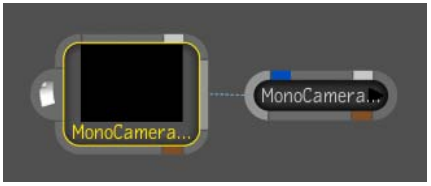


7 To view a layer composition, do one of the following:

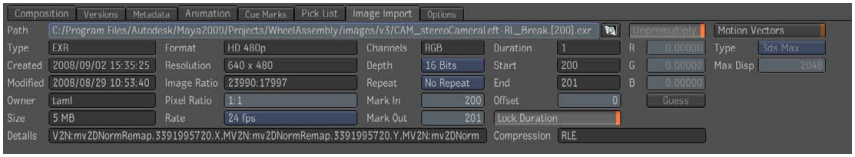
- Double-click a link node.
- Right-click a link node and select Open.
- In the File Browser, open it from the Layers folder.



8 To view a render pass composition, open it from the Passes folder in the File Browser. Click the Import Image node.



In the tool UI, select the Import Image tab.



Notice that the Path field displays path of the image sequence that is imported for that render pass. And, the image format, pixel format, rate, frame range and pre-multiplication import options are properly set.

The Pre-Compositing Import Options

When you import pre-comp files and render layers and passes, you can set the following options in the Pre-Compositing Import Options dialog box. You can also set these options in your project preferences—see Setting Project Preferences on page 97.

Parameter	Description
Destination Folder	The folder in which imported Composite compositions are created or updated.
Update Mode	Specify how to handle existing compositions during import. You can update or recreate the working version, overwrite all versions, skip existing compositions (no update), or asked to be prompted for each composition.
Backup Working Version	Back up the working version of the composition when an update is required.
Default Template	Whether the default composition template should be used when no template is specified.
Multi-Stream	Specifies how to create stereoscopic compositions—see Working with Stereoscopic Compositions on page 319.
Verbose	Increases pre-compositing import script verbosity.
Show Options	Determines whether the pre-compositing options dialog is shown on import.
Scene Subfolder	Adds a subfolder (named after the pre-compositing anchor) to the destination folder.

Parameter	Description
Passes Folder	The name of the subfolder in which the render pass compositions are stored.
Layers Folder	The name of the sub folder in which the render layer compositions are stored.
Merge	Merges the content of all render layer compositions into the scene composition—see Merging Layers on page 321.
Template Path	The location of the scene composition template to use for merging compositions—see Using Pre-Compositing Templates on page 310.
Render Directory	The root folder of the rendered image files hierarchy. It is used to locate the rendered image files.
Save to Project Settings	Saves the current pre-compositing import options in your project preferences—see Setting Project Preferences on page 97.

About Anchors

In a scene, there are unique anchors for the following elements: render layers, render passes, and cameras. Anchors identify these elements in the resulting Composite composition that is created after importing a 3D scene.

One or more anchors can be associated with any node in a dependency graph. You can add, remove, and edit anchors. You can also view a node's list of anchors. When a node has an associated anchor, the anchor tab at the upper-left of the node turns blue.

NOTE An anchor name does not have to be unique within the dependency graph.

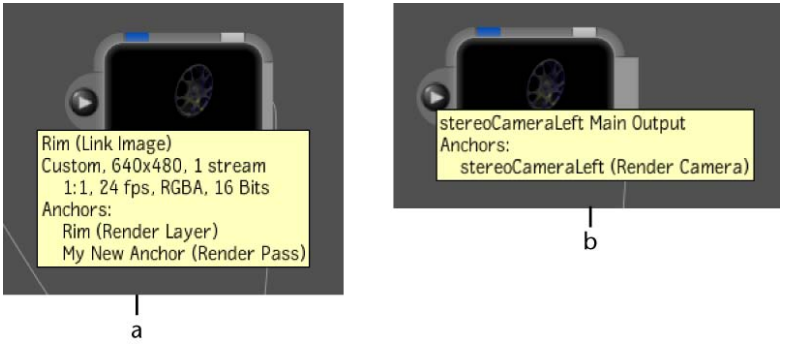
Pre-Compositing Anchors

In a pre-compositing scene, each render pass is uniquely identified by a render pass anchor. The same applies to render layers/render pass anchors and cameras/camera anchors.The various anchor names in the 3D application are used by Composite during the import of pre-comp files. The anchors are used to locate the entry points for each type of 3D asset in the resulting Composite compositions.

Camera anchors are specifically set on composition output nodes and on the output socket of link nodes identify each camera view of a stereo setup.Also, anchors are crucial to the update process, so they should not be changed. It is also important to have properly defined anchors in the composition templates—see Using Pre-Compositing Templates on page 310.

To view a node's anchor:

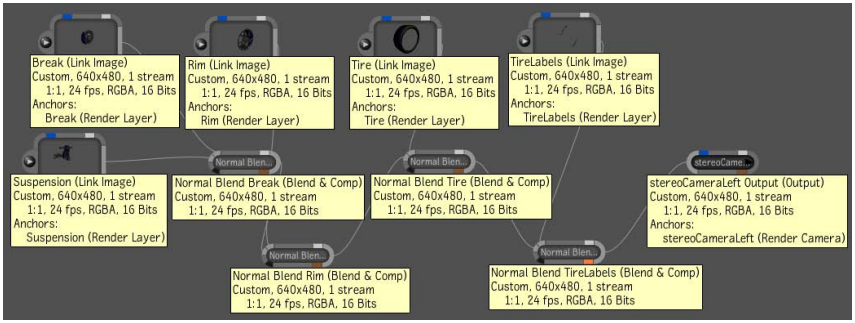
1. Place the cursor over a node or an output socket that has an anchor, which is indicated by blue anchor tab, and press the d (lowercase) key to turn on the Details mode. Now you can place your cursor over any node, as well as its output socket to view its details. Press d again to turn off the Details mode or if you have tooltips turned on, simply hover over the node.



a) Details of a link node with anchors.

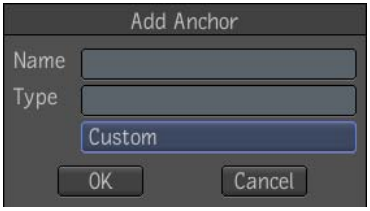
b) Details of the output socket of a link node.

2. To view the details of all nodes in the dependency graph, press **Shift + D**. Press **Shift + D** again to turn off the Details mode.

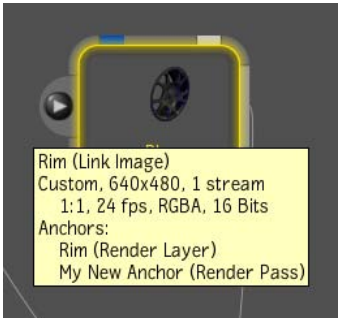


To add an anchor to a node:

1. Right-click the node on which you want to add an anchor and select Add.
2. In the Add Anchor dialog box, type in a name for the anchor.



3. Do one of the following to specify the type of anchor:
   - In the Type field, type in the kind of anchor you are adding. It can be anything you want.
   - Click the button below the Type field and select an anchor type: Custom, Render Pass, Render Layer, or Render Camera.
4. Click OK.
5. To view the anchor, place your cursor over the node and press the d (lowercase) key.



The new anchor was added to the bottom of the list of anchors. The type of anchor is indicated in parentheses.

To remove anchors:

➤ Right-click the node on which you want to remove an anchor and select Anchors and one of the following:

- Remove All: To remove all the anchors associated with the node.
- Remove: To select a specific anchor to remove from the list.

To edit an anchor:

1 Right-click the node on which you want to edit an anchor and select Anchors > Edit Anchor and select the anchor you want to edit.

The Edit Anchor dialog box displays.

2 Edit the anchor name and type and click OK.

Using Pre-Compositing Templates

A pre-compositing template is a Composite composition, a *.txcomposition* file. There are three types of templates: layer, scene, and merged. A template

contains an arbitrary number of anchor link nodes that source an arbitrary dependency graph.

The anchor link nodes in a template can be unconnected or linked to existing compositions; the link is simply replaced when the template is used to create a composition.

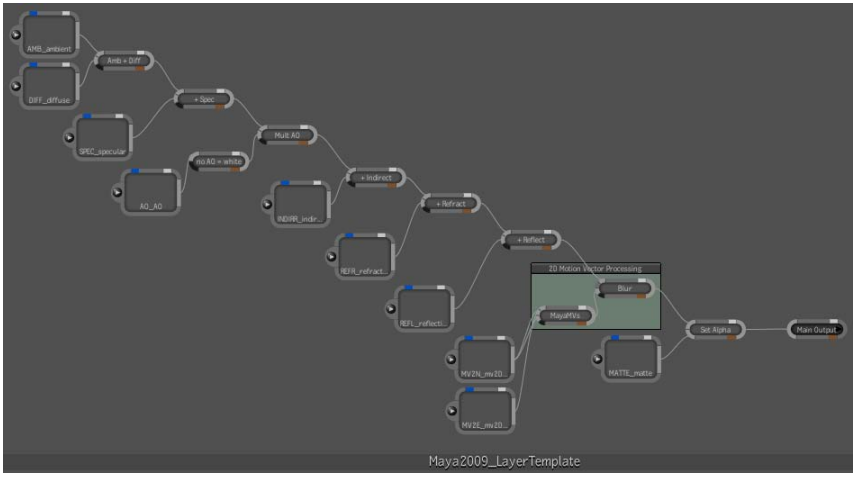
A template has at least one output (the primary) which is typically the final look of the render layer or scene. However, you can add an arbitrary number of secondary outputs to the template.

Template Types

There are three types of templates that you specify in the Pre-Compositing Import Options dialog box during import: layer, scene, and merge.

Layer Template

The layer template assembles passes from a single layer. This is the default template provided with Composite. For example, the following shows the default Maya 2011 layer template provided with Composite. Its link nodes have render pass anchors that match the default name of the standard render passes in Maya 2011.



Because the layer template assembles passes from a single layer, the render layer template composition must contain link nodes with a render pass anchor on them—see Pre-Compositing Anchors on page 308. On render layer templates, the render passes are assembled to reflect the shaders network in the 3D

application. The render layer template can be different on each layer. If no render layer template is specified, the default layer template for the given 3D application is used.

NOTE In Maya, the path to the render layer template composition can be specified on the render layer node.

The name of the default template is based on the application name and its version, such as *Maya2011_LayerTemplate.txcomposition*. The script uses the template file that is closest to the version that the application used to generate the pre-comp file.

Default pre-compositing templates are located in:

- Windows: C:\Program Files\Autodesk\Autodesk Composite 2011\resources\precomp\templates
- Linux: /usr/autodesk/Autodesk_Composite_2011/resources/precomp/templates.
- Mac OS: /Applications/Autodesk/Composite/2011/AutodeskComposite.app/Contents/Resources/precomp/templates.

Scene Template

The scene template stacks the layers using Blend & Comp nodes and assembles the layers in the same order as in the 3D scene. This template assembles layers in the scene for a given camera view, so the scene template composition must contain link nodes with a render layer anchor on them—see Pre-Compositing Anchors on page 308.The scene template can be different for each camera view. If no scene template is specified, the layers are stacked using Blend & Comp nodes and the layers are assembled in the same order and blend modes as in the 3D scene.

NOTE In Maya, the path to the scene template composition can be specified on camera node.

Merge Template

The Merge template assembles all passes of all layers, so the merged template composition must contain link nodes with both Render Pass and Render Layer anchors on them—see Pre-Compositing Anchors on page 308.

Creating a Pre-Compositing Template

You can create your own template for pre-compositing.

To create a template:

1. Import a pre-comp file without specifying a template—see Pre-Compositing Import on page 302.

   If you are creating a render layer template, the render layer composition contains a number of link nodes to render pass footage compositions. The render pass anchors are already set on the link nodes—see Render Layer Composition Creation on page 316.

   If you are creating a scene template, the scene composition contains a number of link nodes to render layer compositions. The render layer anchors are already set on the link nodes—see Scene Composition Creation on page 317.

2. Build the dependency graph.

3. If needed, edit the render pass or layer anchors—see Pre-Compositing Anchors on page 308.

4. (Option) Clean up the template by selecting Composition > Pre-Compositing and one of the clean-up options. When cleaning up the template, link nodes are reset, their name is changed to match the anchors, and unused anchors are removed from nodes and output sockets.

5. Provide the pre-comp file (*.txcomposition*) to the 3D artist as the render layer template.

Pre-Comp File Creation and Updates

The creation and update of pre-comp files always starts with the scene compositions, working its way down to the render layers and finally the render pass compositions.

The scene/layer/pass compositions are named accordingly: <scene anchor> <camera anchor> <layer anchor> <pass anchor>.

WARNING Renaming compositions will impair the update process.

All nodes, including output nodes, in each type of composition can be renamed, since node anchors are used to keep track of the topology during import.

Non-Destructive Updates

Pre-comp import updates are completely non-destructive. When an update is needed on a composition of any type (render layer, render pass, or scene), modifications are applied to a new version of the composition (if you turned on the Backup Working Version option in the Pre-Compositing Import Options dialog box or project preferences), which could be a clone of the working version or a new version recreated from scratch.

Moreover, when an update is applied on a clone of the working version, none of the existing nodes in it are deleted (only new content is added) and existing connections are preserved.

Partial updates are also supported. That is, only some cameras or some render layers and passes could be modified during a pre-composition update.

A scene, render layer, or render pass composition is not updated if it doesn't need to be. You can re-export the entire contents of the scene and only the content that needs to be updated will be.Changing the path or the content of a pre-compositing template will not trigger an update as the Composite application does not detect when the template has changed. In this case, you should use the Recreate Working Version update mode in the Pre-Compositing Import Options dialog box or project preferences.

Frame Rates

Since Composite does not support all frame rates, there may be cases where the time units in the 3D application, such as Maya, may correspond to an unsupported rate in Composite. During import of the pre-comp file, Composite finds and uses the closest supported rate. However, a warning is displayed which applies to all types of compositions (render layer, render pass, or scene).

Render Pass Compositions

Render Pass Composition Creation

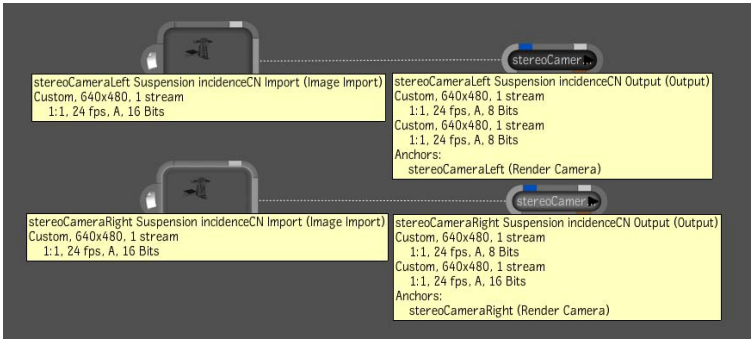
The render pass image files are imported in a new footage composition. If the files are in OpenEXR format, the appropriate channels are selected.

The render pass files don't need to be on disk for them to be imported by Composite in the correct image and pixel format and time range because all of this information can be queried by Composite through the pre-comp file. Also, pre-multiplication with the proper background color is handled correctly.

If the render pass file names do not have extensions or if the extension does not correspond to an image file format that Composite can import, a warning is issued, but the render pass composition is still created and its Import node holds the specified sequence path, however, it produces a red X frame.

The render pass footage can be anywhere on disk, organized in an arbitrary folder hierarchy. The files can be in any format that Composite supports during import—see Supported Media Formats on page 116. If rendered to OpenEXR, the render passes can be stored in any file channel. The pre-compositing import is totally independent from the way images are organized on disk.

If the render pass was rendered from many cameras of a camera set (stereoscopy), the file sequences of each render camera are imported in the same render pass composition, each with it own Import node and output node. The name of the output nodes is suffixed by its corresponding camera name. The camera anchor is set on the output node.



NOTE Unlike other footage compositions which have an Original Footage and a working version, a render pass composition initially has only one version, the working version.

Render Pass Composition Updates

A render pass composition update can be started when there is a new version of the rendered files on disk (includes changes to the rendered files path, the frame range, their image format, the OpenEXR channels the pass is stored in if any, etc.).

When updating a render pass composition, a new empty version of the render pass composition is created if you turned on the Backup Version option in the Pre-Compositing Import dialog box, and the new render pass file sequence

is imported the same way it is done in the creation process—see The Pre-Compositing Import Options on page 306.

Note that the primary version of a render pass composition is never changed by the pre-compositing update. It is up to you to decide which one is the primary—see Working with Compositions on page 217.

Render Layer Compositions

Render Layer Composition Creation

Before the render layer composition is created, render pass compositions for the given render layer are created or updated.

With a Template If a template is specified for the render layer, its content is copied into the render layer composition and the rest of the operations applied to it are quite similar to what happens during an update.

The dependency graph is reviewed to find all the render pass anchor link nodes which are updated to link to their associated render pass composition, which was previously created or updated.

If some render pass anchor link nodes are missing in the template, new unconnected render pass anchor link nodes are automatically added to the render layer composition (in the same way as when no template is specified).

Without a Template If no template is specified, the render layer composition is created with a single unconnected output node. For each render pass composition, there is an unconnected link node that points to it.

The render pass link nodes have their anchors automatically set to their corresponding render pass anchor name and the nodes are initially named after their linked render pass compositions (their names are not changed by further updates).

What Happens Next? By default, a render pass anchor link node points to the Latest version. This is because a new render pass composition version was added during a pre-comp file update is not automatically tagged as the primary version. Choosing the primary render pass version is up to you. If you want the render layer composition to be updated automatically when new Maya render passes are imported in Composite, the render pass anchor link nodes must point to their latest version by default—see Working with Compositions on page 217Working with Compositions on page 217.

The image format, pixel format and rate of the render layer composition are set according to the information contained in the pre-comp file.

The time range (start, end) of the render layer composition is then set to the union of its render pass compositions time ranges.

Render Layer Composition Update

A render layer composition is updated when a new pass is added to the scene. Deleting a pass will not trigger an update because it's non-destructive. Renaming a pass is equivalent to adding a new pass.

The working version of the render layer composition is backed up if you set the Backup Version option. Then, in Update mode, the working version is updated. otherwise it is recreated from scratch (Recreate mode).

Updating the render layer composition is then quite straightforward. Existing render pass anchor link nodes are left unchanged because they point to the latest version of the render pass composition by default. The ones that are missing are created (but are unconnected) and linked to their respective render pass compositions.

If an existing render pass is no longer in the render layer (which could be because it was removed from the 3D scene or because the pre-comp file update is partial), its anchor link node is not deleted; the render layer composition update is non-destructive—see Non-Destructive Updates on page 314.

The format and rate of a render layer composition is not modified on update. However, the start/end of the render layer composition is updated to be the union of its current time range and the time ranges of all its linked render pass compositions.

Scene Compositions

Scene Composition Creation

Before the scene composition is created, render layer compositions in the scene composition are created or updated.

With a Template If a template is specified for the scene, its content is copied into the scene composition and the rest of the operations applied to it are quite similar to what happens during an update of the scene.

The dependency graph is reviewed to find all the render layer anchor link nodes which are updated to link to their associated render layer composition, which was previously created or updated.

If some render layer anchor link nodes are missing in the scene template, new unconnected render layer link nodes are automatically added to the scene composition (in the same way as when no template is specified).

Without a Template If no scene template is specified, render layer anchor link nodes are created with their proper anchor name set and point to their respective render layer composition. Render layers are then stacked in a chain of Blend and Comp nodes, respecting the compositing order and blend modes used in the 3D scene.

What Happens Next? The render layer link nodes have their anchors automatically set to their corresponding render layer anchor name and the nodes are initially named after their linked render layer compositions (their names are not changed by further updates).

The image format, pixel format, and the rate of the scene composition are then set according to the default render settings of the 3D scene. The time range (start, end) of the scene composition is then set to the union of its render layer compositions time ranges.

Scene Composition Update

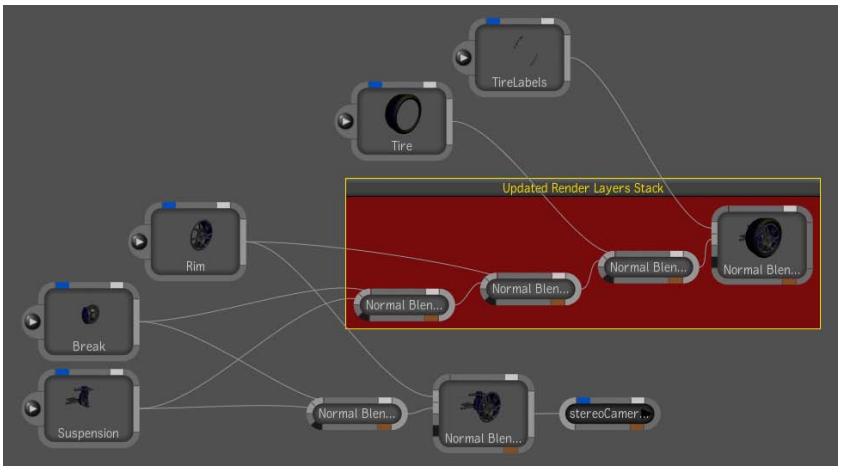
A scene composition is updated when a new layer is added to the scene. Deleting a layer will not trigger an update because it's non-destructive. Renaming a layer is equivalent to adding a new layer.

The working version of the scene composition is backed up if you set the Backup Version option. Then, in Update mode, the working version is updated. otherwise it is recreated from scratch (Recreate mode).

Existing render layer anchor link nodes are left unchanged because they point to the latest version of the render layer composition by default. The ones that are missing are created and linked to their respective render layer compositions.

If an existing render layer is no longer in the scene (because it was removed from the 3D scene or because the pre-comp file update is partial), its anchor link node is not deleted; the scene composition update is non-destructive—see Non-Destructive Updates on page 314.

If the layer ordering or the blend modes between layers have changed, a new stack of Blend & Comp nodes is created within a visual group next to the old stack. In the following example, the tire and tire label layers were added during update.

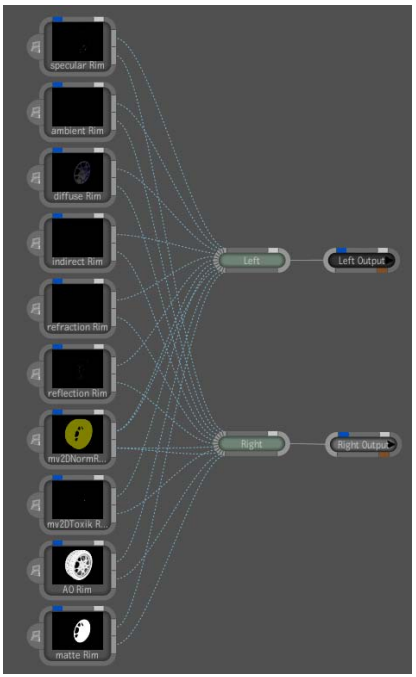


The format and rate of a scene composition is not modified on update. However, the start/end of the scene composition is updated to be the union of its current time range and the time ranges of all its linked render layer compositions.

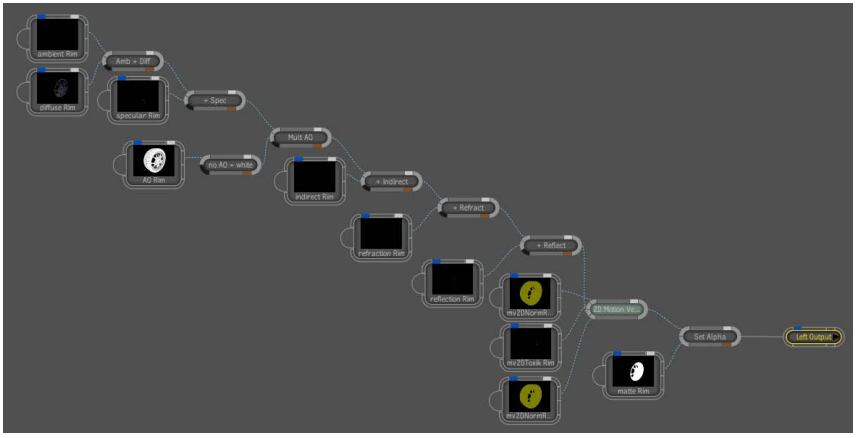
Working with Stereoscopic Compositions

In a stereoscopic scene, rendered files produced by each camera of the stereo rig are imported and composited in the same composition at all levels (pass, layer and scene). There are two modes in which you import a stereo scene, either in multi-stream mode or not—see The Pre-Compositing Import Options on page 306.

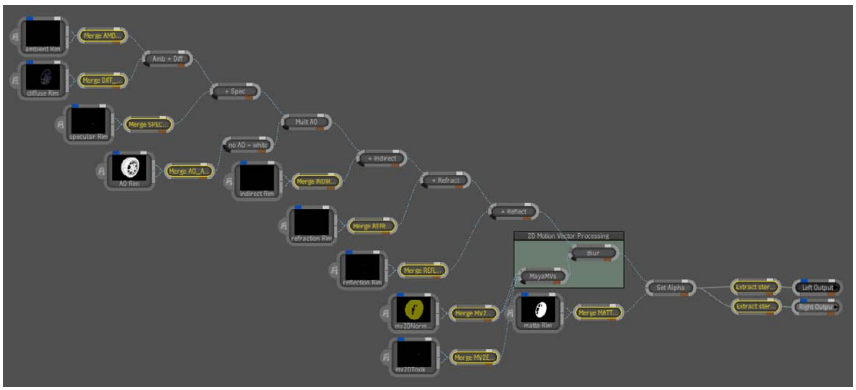
In single stream, the graph of the template is duplicated for each camera and placed within a group node to avoid name and expression clashes. The left and right of the output link nodes feed the corresponding dependency graph.



For example, if you double-click the left group, you'll see the default render layer template in the following dependency graph.



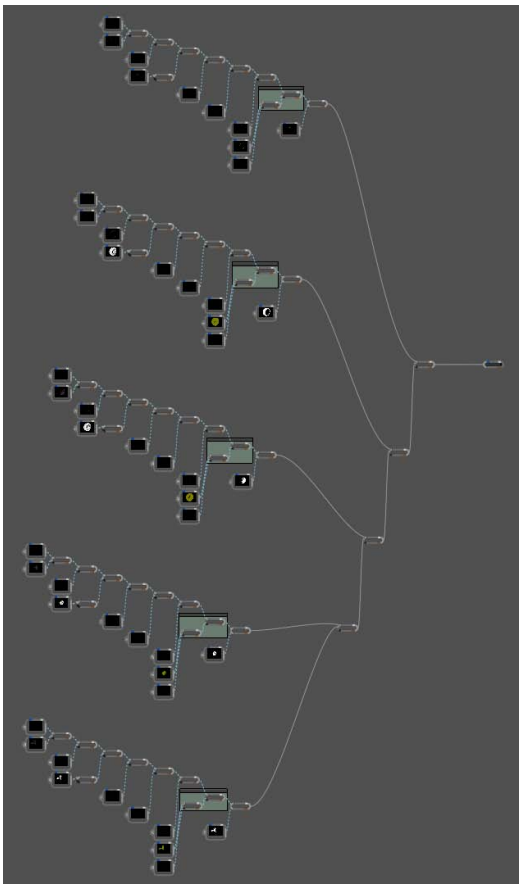
In Multi-Stream mode, the left and right outputs of each link node are merged together, feeding only one instance of the dependency graph. At the end, each stream is extracted—see Multi-stream Compositing on page 371.



Merging Layers

By default, each layer of a scene is imported into a different render layer composition. For greater compositing freedom and to set expressions, you must have all the dependency graphs of each layer in one scene composition.

To merge layers during import, click the Single Composition button in the Pre-Compositing Import Options dialog box. The following shows what a scene composition would look like after merging layers. Notice that each layer is represented by the same dependency graph because they all use the same default template.



You can also specify a template to handle all passes from all layers—see Using Pre-Compositing Templates on page 310.

You can replace a link node by the content of its linked composition.

To merge in a composition:

➤ In the Schematic view, right-click a Link Image node and select Merge and one of the following:

- Non-Recursive: Bring in one level of the linked composition.
- Recursive: Apply the merge recursively on all link nodes found in the linked composition version.
- Recursive Except Footage: Stop recursion at footage compositions.

Importing FBX Files

14

Topics in this chapter:

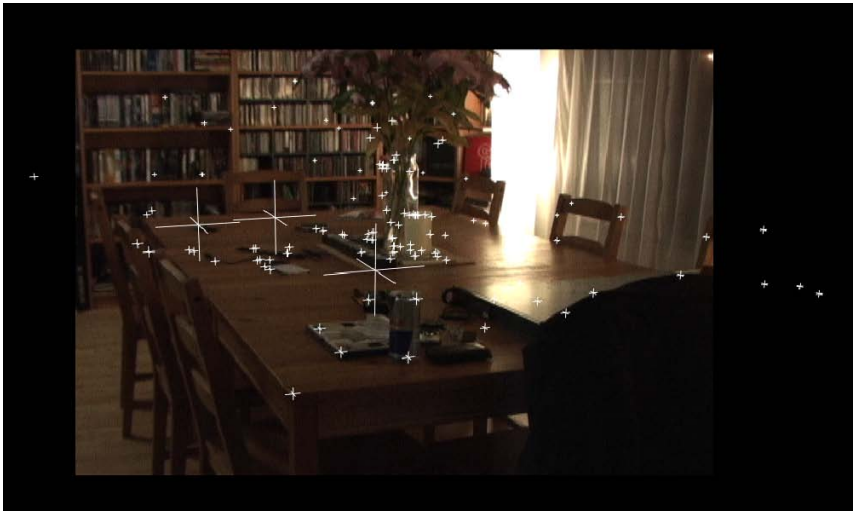
Importing FBX Files

You can import all renderable cameras, camera planes, lights, geometry and locators. When you import a scene, a new composition is created within a Reaction node with all 3D data.

The name of the created composition (the one that contains the Reaction node) is based on the name of the FBX file. However, 3D updates are not supported; you can only overwrite the existing composition. To update a composition, you can transfer objects from one Reaction node to another by copying and pasting.

NOTE All FBX Geometry is imported in a single layer.

To Import an FBX File



To import an FBX file into Composite:

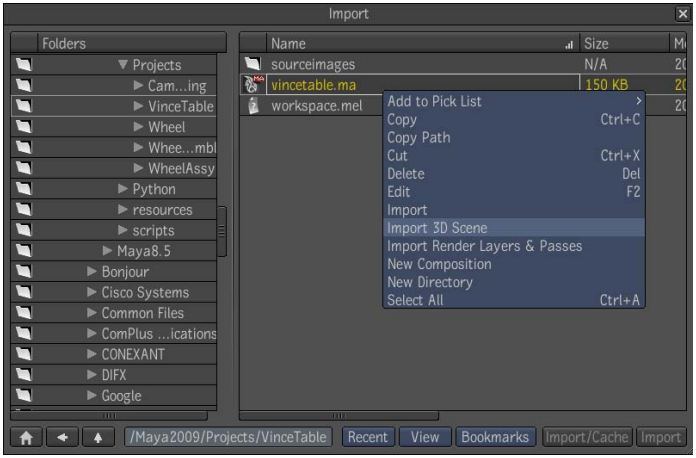
1. Select File > Import or press **Ctrl + I**(for Windows and Linux) or **Cmd + I**(for Mac OS).

   The File Browser is displayed.

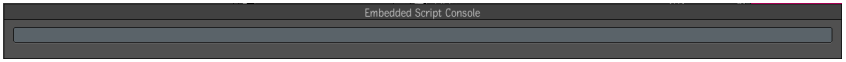
2. Locate the folder in which the FBX file (*.fbx*) resides.

3. Right-click import or click the Import button to import and select Import 3D Scene.

NOTE Multiple FBX files can be imported into Composite at the same time.

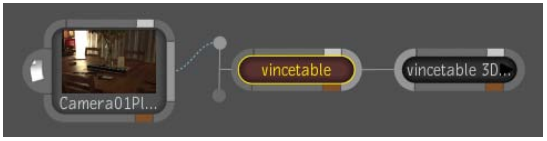


The Embedded Script Console displays the progress of the import.



4 Open the File Browser using the The Gate UI on page 12 or by pressing **Ctrl + O** (for Windows and Linux) or **Cmd + O** (for Mac OS).

5 Open the imported composition. The composition name is based on the FBX file name but without its extension.



6 Select the Reaction node to view the layers.

WARNING There is currently no mechanism in Composite that locks imported objects to their original values. If an object's values are changed by mistake, use the undo feature or **Ctrl** + **Z** (for Windows and Linux) or **Cmd** + **Z** (for Mac OS). Pressing Reset will reset the values to the default settings of the identity matrix, not to the original values of the object.

To view the imported objects in the Player:

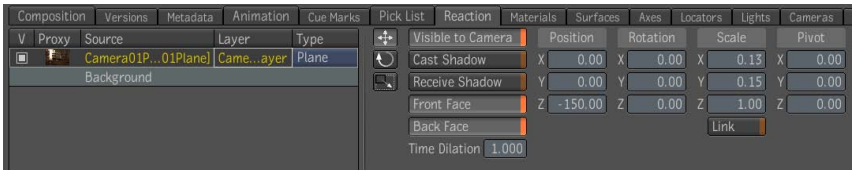
1. Right-click and swipe south to display the Player options.
2. Under the Reaction label, select Interactivity. Under the Display label, select Tool Output.

   Imported objects are now visible in the Player.

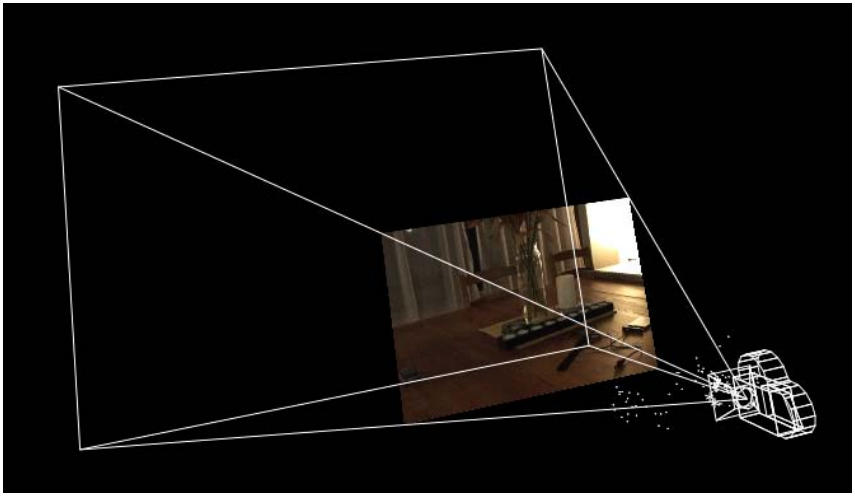
To view the camera image plane and its properties:

1 In the Layer Editor, select the camera image plane layer.

Its properties are displayed in the Reaction tab.

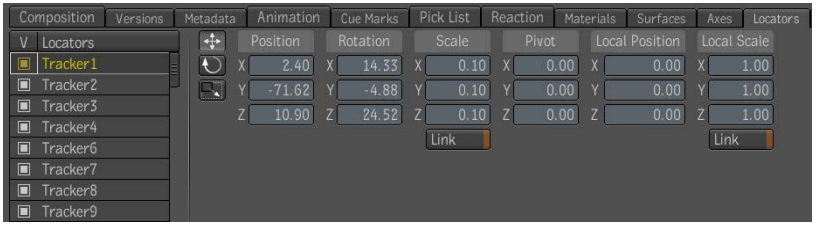


2 In the Player, right-click and select View From > Perspective or press **Shift + 2** to view from the perspective camera.



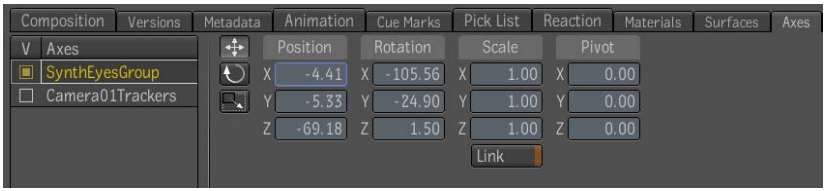
To view locators and their properties:

➤ Select the Reaction node in the Schematic view and then select the Locators tab.



To view the imported FBX transforms:

➤ In the Tool UI, select the Axes tab and select an axis to view.



The imported FBX transforms are displayed in the Player and their properties displayed in the Axes tab.

To view the camera(s):

1 In the Tool UI, select the Cameras tab.

2 From the Cameras list, select Camera01.

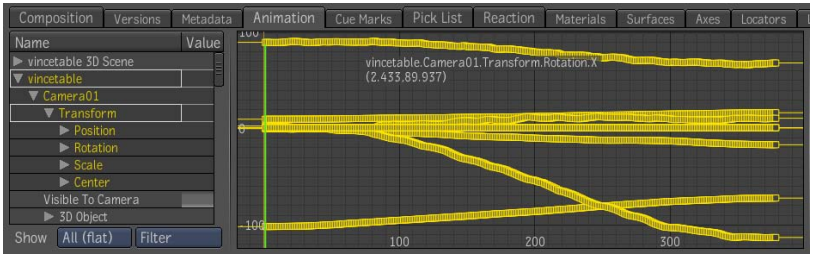
The camera is selected in the Player and its properties are displayed in the Cameras tab.

To view the camera animation:

1 In the Tool UI, select the Animation tab.

2 In the Composition Browser, select the Reaction node, which is named after the FBX file. Select Camera > Transform.

The Animation Editor displays the animation curves of the camera transformation. To accurately represent the animation that was created in the FBX file, a keyframe is set at each frame.



FBX Import Limitations

The Composite 3D workflow lets you import scenes into Composite. The following tables show which node attributes are brought over to Composite (indicated by an “S” for supported) and whether the attributes can be edited (indicated by a “E”) in Composite or whether it's recalculated into some other attributes or Composite paradigms.

Transformation Attributes

Group	Attributes	S	E	Comments
Transform	Translate	X	X	
	Rotate	X	X	
	Scale	X	X	
	Shear			
	Rotate Order	X		Recalculated in Composite transform. Expressions set on those will interfere with the recalculation process.
	Rotate Axis	X		
	Inherits Transform			
Pivots	Rotate Pivot	X		Recalculated in Composite transform. Expressions set on those will interfere with the recalculation process.

Group	Attributes	S	E	Comments
	Scale Pivot	X		
	Display Rotate Pivot			
	Display Scale Pivot			
Object Display	Visibility	X	X	In Composite, it affects only Axis visibility (not whole hierarchy).

Locator

Group	Attributes	S	E	Comments
Local Transform	Position	X	X	
	Scale	X	X	
Object Display	Visibility	X	X	

Camera

Group	Attributes	S	E	Comments
Camera Attributes	Controls (Aim, Aim-Up)	X		Only free cameras in Composite.
	Focal Length (Horizontal Angle of View)	X	X	
	Camera Scale			
	Auto Clip Planes			
	Near Clip Plane	X	X	
	Far Clip Plane	X	X	
Film Back	Film Gate Presets	X	X	
	Camera Aperture	X	X	

Group	Attributes	S	E	Comments
	Film Aspect Ratio	X	X	
	Lens Squeeze Ratio	X	X	
Film Fit On Resolution Gate	Film Fit Mode	X	X	
	Film Fit Offset	X	X	
	Film Offset	X	X	
Film Back 2D Transform	Pre Scale			
	Film Translate			
	Film Roll Pivot			
	Film Roll Value			
	Film Roll Order			
	Post Scale			
Output Settings	Renderable	X		Camera is imported if renderable.
	Mask	X		Mask of the "render" (current) camera translates to Alpha channel in Reaction output format. Affects all cameras.
	Depth			Image Z channel not supported in Composite.
Environment	Background Color			
Display Options	Display Film Gate			
	Display Resolution Gate		X	Player setting.
	Display Field Chart			

Group	Attributes	S	E	Comments
	Display Safe Action			
	Display Safe Title		X	Player setting.
	Display Film Pivot		X	
	Display Film Origin			
	Overscan			
Orthographicview	Orthographic Mode	X	X	
	Orthographic Width			Can not be set in Composite. Orthographic height is always 100 and width is set according to output pixel aspect ratio.
Object Display	Visibility	X	X	

Camera Image Plane

Once imported into Composite, an FBX image plane loses part of its relationship with its camera. It is imported as a normal plane layer parented to the camera transformation, but it does not rescale itself automatically when its depth or the camera view frustum changes.

Group	Attributes	S	E	Comments
Image Plane Attributes	Display Mode (RGB, RGBA, A, Luminance, etc.)			
	Color Gain			
	Color Offset			
	Alpha Gain			
	Attached to Camera vs. Fixed			Always attached to camera.

Group	Attributes	S	E	Comments
	Type			Always Image File (Texture not supported).
	Image Name	X	X	In Image Import node feeding the Reaction source.
	Use Image Sequence	X		Can be controlled through Image Import node mark in/out.
	Frame Offset	X	X	In Image Import node feeding the Reaction source.
Placement	Fit Mode			Always “To Size”.
	Size	X		Recalculated into plane layer transformation according to Depth and camera Focal Length.
	Offset			
	Depth	X		Recalculated into plane layer transformation.
	Rotate	X		
	Squeeze Correction	X		Recalculated into plane layer transformation (inverse of camera Lens Squeeze Ratio).
Placement Extras	Coverage Width			Could be translated into a chain of Composite Crop and Resize nodes at the Reaction source.
	Coverage Height			
	Coverage Origin			
	Center			Irrelevant since Fixed camera image plane is not supported.
	Width			
	Height			

Current Limitations

The following list outlines the current limitations for FBX import:

- Password protected FBX files cannot be imported in Composite.
- You cannot selectively import assets from the FBX file; Composite imports everything it recognizes.
- By default, the geometry import tool imports all meshes of the scene into a single layer. However, you can select which geometry to import.
- The geometry tool imports the global transformation of the geometry in Composite; its bakes the transformation of its parent hierarchy into the mesh.
- Composite doesn't support multiple UV coordinate on a vertex,; it only imports the first (index 0) diffuse set of coordinates.
- FBX animation curves are recalculated into a Composite animation with one keyframe at each frame.
- If the FBX scene contains more than one camera, the format of the resulting composition (and the Reaction node) is based on the render settings of the default camera.
- The camera image plane distance, its size on aperture, the camera focal length and the horizontal scale are all animatable values in FBX, but the transformation of the imported plane layer in Composite is not animated.
- The image plane offset values which controls how much the center of the image plane is offset from the centre of the viewing frustum of the camera are not supported by the FBX import.
- The camera image plane layer cannot be created correctly if Composite doesn't have access or cannot import the asociated image files, because it needs to know the image resolution to scale the plane layer in Reaction. Also, the Image Import tool cannot store the path of an image sequence that doesn't exist.
- Parameters that control the amount of the source image that is used in the camera image plane are not supported.
- Target cameras are not supported.
- Once the Maya camera image plane is imported in Composite, it is no longer a “camera” image plane (one that automatically rescales itself to fit

the cam¬era view), just a simple image plane parented by the camera and properly scaled for the current depth of the plane.

Premultiplication

15

Topics in this chapter:

Premultiplied Images

A premultiplied image is one where the RGB channels have been multiplied by the alpha channel. In other words, it is a four-channel image in which the red, green and blue channels have already been multiplied by the integrated matte channel.

Typically, images from a 3D renderer are premultiplied, which means that the transparent areas are black in both the RGB channels and the alpha channel. As such, in premultiplied images, the RGB channels never have a higher value than the alpha channel.

When to Premultiply

You should consider premultiplying an image if you have to modify a foreground element and composite over a background image. Furthermore, you should also

consider premultiplying two or more images whenever you have to perform a color correction or use filtering nodes.

When you premultiply an image by a matte, you should be aware that there is a very specific brightness relationship between the pixels in the color channels and the pixels in the matte. Since certain systems that will assume that you are working with unpremultiplied systems, will also assume this brightness relationship. As such, the brightness of any color channel cannot be modified without also impacting the alpha channel.

There are two general rules to remember when considering premultiplication:

1. Only color correct unpremultiplied images.
2. Only apply filter and transform nodes to premultiplied images.

Common Problems with Premultiplication

There are certain problems that can occur when the rules of premultiplication are ignored. These can range from problems with edges to raised global levels. Additionally, when the premultiplied state of an image is ignored, there may also be unwanted fringing around a masked object or the unwanted side effects that may occur when a node affects parts of an image that it should not.

A possible result of compositing with a premultiplied image would be slight lightening or darkening of the composite areas where the foreground matte is semitransparent.

Another common problem is when color correction has been applied to a foreground premultiplied image. This could result in the blacks in the scene being elevated above the value of 0, which means that even though the matte channel may still specify that the surrounding field is in black, the RGB channels may have some small, often visually undetectable value.

Premultiplication Tools

Composite includes two tools for the purpose of premultiplication functions: the Premultiply tool and the Unpremultiply tool. The Premultiply tool premultiples an input image (see Premultiply Tool on page 413). The Unpremultiply tool unpremultiplies an input image (see Unpremultiply Tool on page 414).

2D Compositing

16

Topics in this chapter:

About 2D Compositing

This chapter describes the 2D compositing tools available to you in Composite. They are described in the order in which they appear in the Tools tab.

2D Compositor

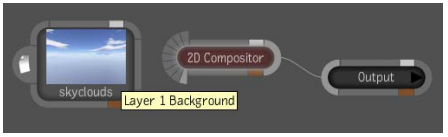
The 2D Compositor tool lets you create multilayer compositions. It allows you to stack four layers over a background. Each layer becomes the foundation for the subsequent layer, starting with the background, and moving up the stack. You must use a Matte image with an alpha channel between layers to expose what lies beneath the top layer.

NOTE If the top layer has an embedded alpha channel, it is shown as black. You can connect the same composition to the Matte channel to achieve the desired result.

The 2D compositor is a "super tool" because it contains a separate 2D Compositor Layer tool inside the tool. A "super tool" is identified in the Tools tab by a spherical icon—see Tools, Views & Pick List tabs on page 30.

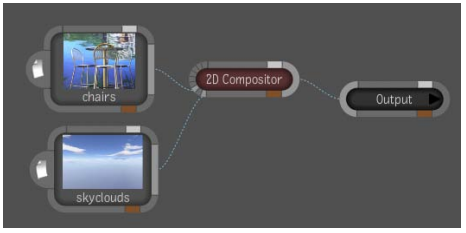
To composite two or more layers:

1 Drag the 2D Compositor tool to the dependency graph in the Schematic view and connect an image or tool output to the Layer 1 Back input tab. This is the background for the first layer.

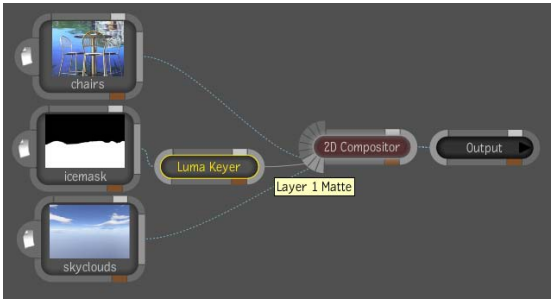


NOTE If you drop a 2D Compositor tool onto an existing connection between two nodes, the existing composition becomes the Layer 1 background input.

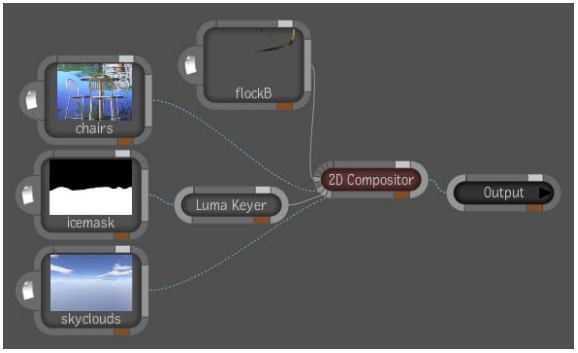
2 Connect a composition to the Layer 1 Front input. This is the image you want to have on top of the background.



3 Connect a composition to the Layer 1 Matte input. This image will determine what portions of the background are visible if the image does not have an embedded alpha channel.



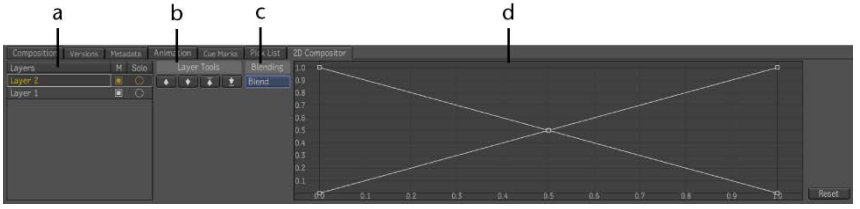
4 Add more layers as desired. If a composition has an embedded alpha channel, the alpha channel will appear black. You can connect the same image to the associated Matte input to achieve the desired result.



Adjusting the Matte's Luminance Curves

You can adjust the luminance curves of the front and back mattes of your composite by selecting the layer from the 2D Compositor tool.

The 2D Compositor tool UI is divided into four areas:



(a) Layer Browser (b) Layer tools (c) Blending modes (d) Luminance curve controls

To adjust the matte's luminance curves:

1. In the Layer Browser, select the composite layer to adjust. You can mute or solo the layer.
2. Use the Layer Tools to navigate through the 2D Compositor's layers.
3. Select mathematical blending operations from a context menu.
4. Once a layer is selected from the Layer Browser, click a curve to select it. Adjust the luminance curve by clicking a point to display its tangent handle and drag the handle to adjust the curve. Using hotkeys, you can add and delete control points, break or unify tangent handles—see Animation Editor on page 662.

Alpha Tools

The Alpha tools let you control the look of your alpha:

Tool	Description
Alpha Levels	For matte cleanup tasks during the keying process—see Alpha Levels on page 343.
Blend Alpha	Blends two mattes together under the optional control of a third matte—see Blend Alpha on page 344.
Blur Alpha	Softens the alpha channel of its primary input—see Blur Alpha on page 348.
Clamp Alpha	Brings the alpha channel of the primary input within a predetermined range—see Clamp Alpha on page 349.
Cleanup Alpha	Lets you remove gray details from white and/or black regions of the alpha channel—see Cleanup Alpha on page 349.
Comp Alpha	Composites two mattes together—see Comp Alpha Tool on page 350.
Control Edge	Allows edges in the matte to be moved, stretched, and softened—see Control Edge on page 352.
Detect Edge	Replaces the source alpha channel with an edge map of the source alpha—see Detect Edge on page 354.

Tool	Description
Drop Alpha	A simple matte modifier that drops the alpha channel from its primary input—see Drop Alpha on page 356.
Edge	Lets you refine the edges of a matte—see Edge Tool on page 356.
Extract Alpha	Extracts the alpha channel from its primary input—see Extract Alpha on page 357.
Fade	Lets you modify the transparency of an image—see Fade Tool on page 357.
Invert Alpha	Inverts the alpha channel of its primary input—see Invert Alpha on page 358.
Median Alpha	Excellent for removing impulse noise—see Median Alpha on page 358.
Remap Alpha	Remaps the alpha—see Remap Alpha on page 358.
Set Alpha	Create an alpha channel for an image based on a component from a second image—see Set Alpha Tool on page 360

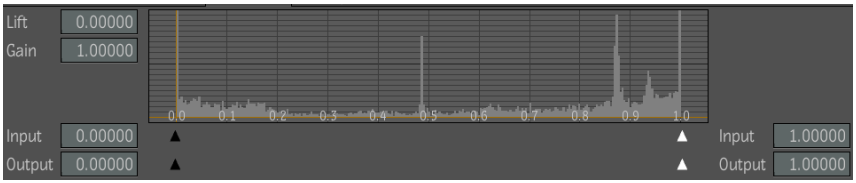
Alpha Levels

The Alpha Levels tool is one of the tools used for matte cleanup tasks during the keying process. When used as a tool defined by the keyer tool's dependency graph, the Alpha Levels tool applies its effect to the alpha component of the keyer's output. You can clean up the matte by manipulating the input/output luma remapping curve, by adjusting the input and output luma fields or sliders, or by adjusting the lift and gain. The Alpha Levels tool also allows you to invert the matte. The Alpha Levels tool is located in the Alpha tools folder in the Tools tab.

The histogram indicates the distribution of luma values in the input source alpha channel with gray vertical bars along the X-axis. The curve illustrates the resulting luma value. For example, if you drag the part of the curve that represents the darker input grays upwards, the curve over that region becomes lighter.

Since the Alpha Levels tool affects the alpha and is typically applied to the output of a keyer tool, such as the Luma Keyer or the Diamond Keyer, your dependency graph should include, as a minimum, one source and a keyer.

The Alpha Levels tool UI has the following parameters.



(a) Lift/Gain fields (b) Minimum Output slider and field (c) Minimum Input slider and field (d) Luma Remapping Curve (e) Maximum Input slider and field (f) Maximum Output slider and field

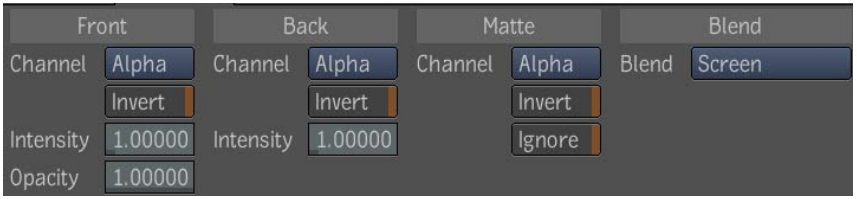
- **Lift** Adjust the Lift to add an overall offset to the matte.
- **Gain** Adjust the Gain to adjust a scaling factor for the matte. Lift, Gain, Input and Output are animatable attributes—see Marking Attributes for Keyframing on page 674.
- **Minimum Output slider** Drag to remap input blacks to dark gray.
- **Minimum Input slider** Drag to the right to remap dark grays as black.
- **Luma remapping curve** View the changes you make in this curve.
- **Maximum Input slider** Drag to the left to remap light grays as white.
- **Maximum Output slider** Drag to the left to remap output whites to light gray.

Blend Alpha

The Blend Alpha tool is used to blend two mattes together under the optional control of a third matte. It has front, back, and matte inputs. It extracts a matte from the front image and composites it over the alpha channel of the back input using a choice of blend modes. The coverage of the front can be controlled by the matte input. The back is the primary input; the output inherits the format of the back input; this tool only affects alpha; if the back is an RGBA image, the color part is simply copied to the output.

NOTE The alpha output of this tool is always clamped to the [0,1] interval.

The Blend Alpha tool has the following parameters:



- **Front Channel** Selects which channel to use for the front. Channel selections include Luma, Red, Green, Blue, and Alpha (default is Alpha).
- **Front Invert** Inverts the front before using it (default is off).
- **Front Intensity** Specifies the intensity of the front layer (default is 100%; range is [0,10]).
- **Front Opacity** Controls the opacity of the front in the blending. If a matte image is also used to control the blending, the two are multiplied together. This parameter is never ignored (default is 100%; range is [0,1]).
- **Back Channel** Selects which channel to use for the back. Channel selections include Luma, Red, Green, Blue, and Alpha (default is Alpha).
- **Back Invert** Inverts the back before using it (default is off).
- **Back Intensity** Specifies the intensity of the back layer (default is 100%; range is [0,10]).
- **Matte Channel** Selects which channel to use for the matte. Channel selections include Luma, Red, Green, Blue, and Alpha (default is Alpha).
- **Matte Invert** Inverts the matte before using it (default is off).
- **Matte Ignore** Determines whether or not the matte input is used to modulate the blend. The default is false (meaning that the matte input will be used in the blending equations). Note that if the Matte Input is not chain connected, it will be automatically ignored (no feedback needs to be provided in the UI).
- **Blend Mode** Determines which blend mode will be used (the default is Normal).
  Click the Blend button to view other available modes.

Blend Modes

The following tables (grouped by type) list the available blend modes and describe the resulting blend effect.

NOTE The Blend Alpha tool has four fewer modes than the Blend & Comp and Glow tools, because the Blend Alpha tool affects only the alpha channel.

Basic blend modes

Mode	Blend Result
Normal	This is the default setting and displays the front input
Average	Adds the front and back, then divides by 2.
Add	Adds the front and back.
Subtract	Subtracts the front from the back.

Blend modes that darken

Mode	Blend Result
Darken	Compares the pixels of the front and back inputs at a given location and selects the darker of the two.
Multiply	Multiplies the color values of the back input pixels and the front input pixels, producing a darker color.
Color Burn	Colorizes darker back input pixels with the front input color.
Linear Burn	Same as Color Burn but with less contrast.

Blend modes that lighten

Mode	Blend Result
Lighten	Compares the pixels of the front and back inputs at a given location and selects the lighter of the two.
Screen	Makes the light areas much lighter, darker areas somewhat lighter.
Color Dodge	Colorizes lighter pixels with the front input color.
Linear Dodge	Same as Color Dodge but with lower contrast.

Modes based on lighting

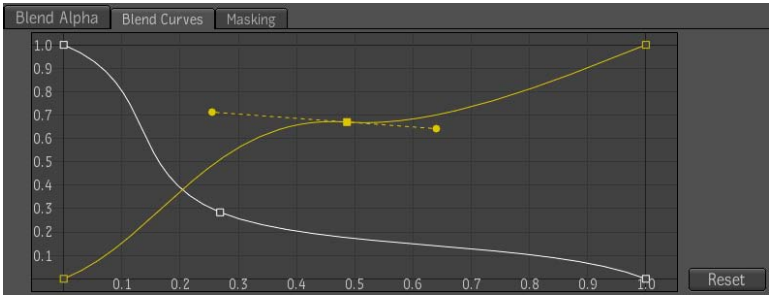
Mode	Blend Result
Spotlight	Like Multiply but with 2 X the brightness.
Spotlight Blend	Same as Spotlight but also adds ambient illumination to the back input.
Overlay	Darkens or lightens the pixels depending on the back color.
Soft Light	If the front color is lighter than mid-gray, the image is lightened. If the front color is darker than mid-gray, the image is darkened.
Hardlight	If the front colors are lighter than mid-gray, screen mode is applied. If the front colors are darker than mid-gray, multiply mode is applied.
Pin Light	Replaces the back colors depending on the brightness of the front color. If the front color is lighter than mid-gray, back colors darker than the front color are replaced. And vice versa; if the front color is darker than mid-gray, back colors lighter than the front color are replaced.
Hard Mix	Produces either white or black depending on similarities between front and back.

Difference Modes

Mode	Blend Result
Difference	Looks at the front and back inputs, and subtracts the less bright from the brighter one.
Exclusion	Similar to Difference but with less contrast.

Blending Curves

Blend curves are used to adjust the blend of the front and back elements by adjusting the amount of fractional opacity (pixel area) for the front, and one minus front. The latter gives more importance to the back. Blend curves is useful for controlling the blend along the edges of a matte by adjusting fractional values, not 0 and 1 values. Because mattes have fractional values along the edges, the blend curves affect the edge blending.



To blend curves:

➤ Select and drag a control point on the curve.

To add a control point to the curve:

➤ With the cursor in the curve window, press + (plus sign) and click the curve at the location in which you want to add a control point.

To delete a control point from the curve:

➤ With the cursor in the curve window, press - (minus sign) and click the control point you want to delete from the curve.

To remove the Add or Delete control point mode:

➤ Press **Esc**.

To reset a single curve:

➤ Select the curve and click Reset.

To reset both curves at the same time:

➤ Hold down the **Ctrl** key and select both curves then select Reset.

NOTE Blend Curves can be animated.

Blur Alpha

The Blur Alpha tool lets you soften the alpha channel of its primary input.

The Blur Alpha tool has the following parameters:

- **X Radius** Change this value to blur horizontal pixels.
- **Y Radius** Change this value to blur vertical pixels.
- **Link** Select to link X and Y values.

NOTE X and Y Radius are animatable attributes—see Marking Attributes for Keyframing on page 674.

Clamp Alpha

The Clamp Alpha tool is used to bring the alpha channel of the primary input within a predetermined range. You can use clamp alpha values outside of the [0,1] range in order to prepare the alpha channel for use in compositing operators. This is necessary because Composite does not force alpha values to be in the [0,1] range.

The Clamp Alpha tool contains the following parameters:

- **Minimum Alpha** Set Largest negative float point. By default, Min is 0.
- **Maximum Alpha** Set Largest positive float point. By default Max is 1.0.

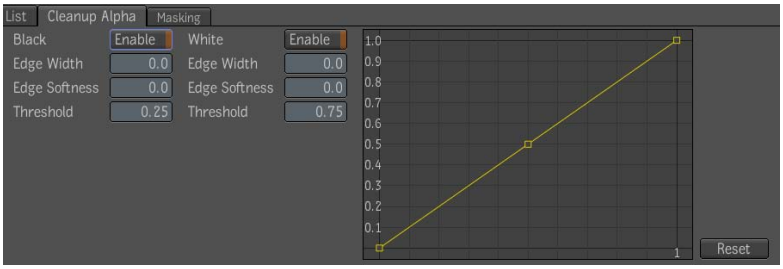
NOTE Min and Max Alpha are animatable attributes—see Marking Attributes for Keyframing on page 674.

Cleanup Alpha

The Cleanup Alpha tool lets you remove gray details from white and/or black regions of the alpha channel. You can choose to remove holes in black regions,

white regions, or both. You can specify what range of values is black and what range is white. In addition, you can control the maximum number of pixels of edges to preserve and the softness controlling the blend between the preserved edge region and the interior filled-in region. The specified edge preservation width is considered to be a maximum limit; for instance, when removing holes in white regions, all pixels that are connected to black regions and within the specified distance from black regions will be considered part of the edge and preserved. Essentially the black region fills into the edges until it hits the white region or its maximum width limit, and then this is used as a mask to preserve input values in the filled region.

The Cleanup Alpha tool has the following parameters:



Use:	To:
Enable	Enable the black or white (or both) clean up controls.
Edge Width	Control the number of pixels of edges to preserve.
Edge Softness	Control the blend between the preserved edge region and the interior filled-in region.
White/Black Threshold	Specify black or white (or both) thresholds.
Curve	Gesturally manipulate the alpha remapping.
Reset	Reset the remapping curve.

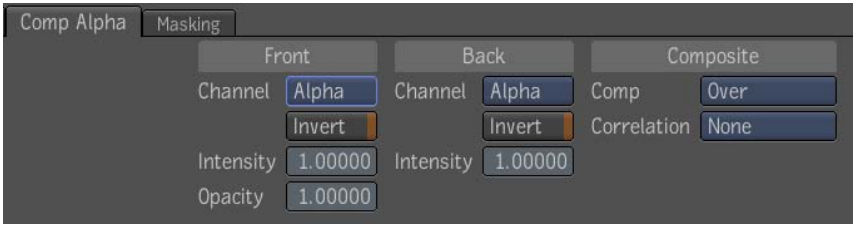
Comp Alpha Tool

The Comp Alpha tool lets you composite two mattes together. It has front and back inputs, and works by extracting a matte from the front image and compositing it over the alpha channel of the back input using a choice of compositing operators. The back is the primary input and the output inherits the format of the back input.

This tool only affects alpha. If the back is an RGBA image, the color part is simply copied to the output.

NOTE The alpha output of this tool is always clamped to the [0,1] interval.

The Comp Alpha tool has the following parameters:



- **Front Channel** Selects which channel to use for the front (default is alpha).
- **Front Invert** Inverts the front before o using it (default is off).
- **Front Intensity** Specifies the intensity of the front layer. Default is 100% and range is [0,1].
- **Front Opacity** Controls the opacity of the front in the compositing. If the opacity is less then one, the front will get more transparent and you will start seeing the back through it. Default is 100%; range is [0,1].
- **Back Channel** Selects which channel to use for the back (default is alpha).
- **Back Invert** Inverts the back before using it (default is off).
- **Back Intensity** Specifies the intensity of the back layer. Default is 100%; range is [0,1].
- **Comp Mode** Determines which compositing mode will be used (default is Over)—see Compositing Operators on page 352.
- **Correlation** Specifies how the two input mattes are correlated. This can be used to improve the quality of the composite in special cases. For example, if you composite two mattes that share a good portion of their outline, you should indicate if they are Adjacent or Superposed. By default, the correlation mode is None, assuming that normally, the input mattes are not correlated.

Compositing Operators

The following table lists the compositing operators applicable between front and back input images:

Operator	Result:
Over	Composites the front over the back. The output will cover any area covered by either the front or the back. Where the front and back overlap, the output will show the front.
Replace	Completely replaces the back image with the front image.
Atop	Similar to the Over operator, but the output will cover the same area covered by the back.
Inside	Composites the front over the back, but the output will cover only the area covered by both front and back.
Outside	The output will be equal to the front except that the part of the front overlapping the back image will be missing. No part of the back will be visible in the output in any case.
Cutout	The output will be equal to the back except that the part of the back covered by the front image will be missing. No part of the front will be visible in the output in any case. Basically this operator is equal to the Outside operator with the front and back roles reversed.
Xor	Both front and back are copied to the output except for the part where they overlap.

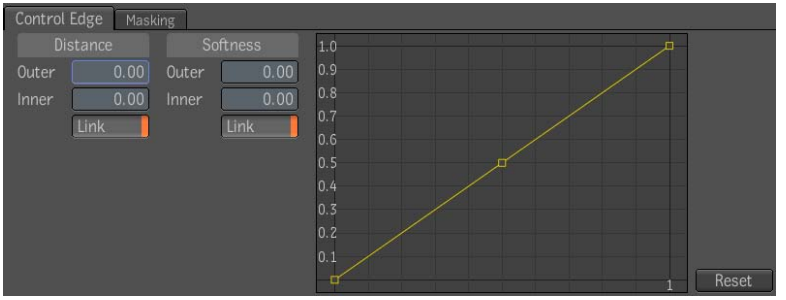
Control Edge

The Control Edge tool lets you move, stretch, and soften the edges of a matte. You can widen an edge, offset it toward the interior or exterior, and blur its inner and outer extremities separately.

(a) Original matte (b) Modified with Control Edge (c) Alpha Remapped

NOTE This tool can be masked and animated.

The Control Edge tool has the following parameters:



Use:	To:
Outer Distance	Control the width and offset of the outer edges.
Inner Distance	Control the width and offset of the inner edges.
Link Distance	Link the Outer and Inner Distance sliders so that the alpha edges are thickened by an amount of twice the slider value, centered about their original position in the input image. By default, the Link button is enabled.
Outer softness	Add blur to the outside extremities of edges.
Inner Softness	Add blur to the inside extremities of edges.

Use:	To:
Link Softness	Link the Outer and Inner Softness sliders. By default, the Link button is enabled.
Remap Alpha Curve	Remap the alpha as the last step in the internal pipeline.
Reset	Reset the alpha remapping curve.

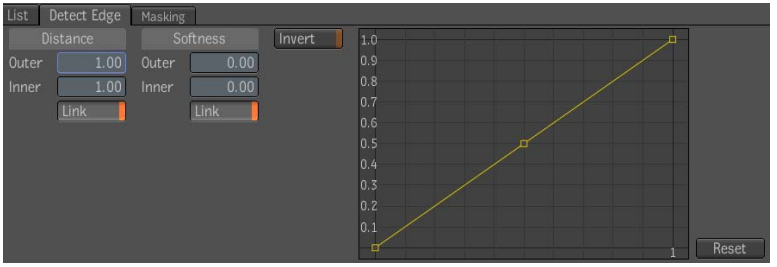
Detect Edge

The Detect Edge tool lets you create an image composed of the edges in an image. The resulting grayscale image can be used as a matte or to produce special effects. You can apply the Detect Edge tool to a color or monochrome image. This tool has many applications, the most useful include:

- Delimiting a portion of the matte to perform color corrections on the front image.
- Creating special effects by using it directly on the front or back image.

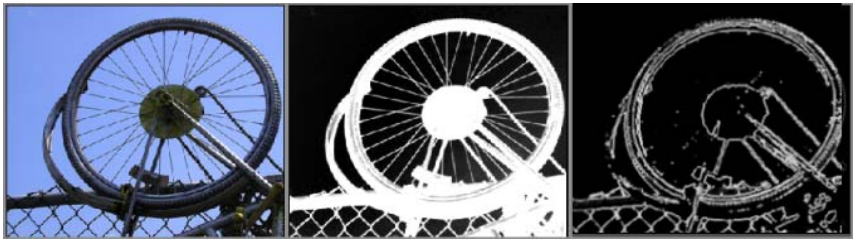
The Detect Edge tool replaces the source alpha channel with an edge map of the source alpha.

The Detect Edge tool has the following parameters:



Use:	To:
Outer Distance	Control the width and offset of the outer edges.
Inner Distance	Control the width and offset of the inner edges.

Use:	To:
Outer softness	Add blur to the outside extremities of edges.
Inner Softness	Add blur to the inside extremities of edges.
Link Distance	Link the Outer and Inner Distance sliders so that the alpha edges are thickened by an amount of twice the slider value, centered about their original position in the input image. By default, the Link button is enabled.
Link Softness	Link the Outer and Inner Softness sliders. By default, the Link button is enabled.
Invert	Create negatives of Edge Mattes, by simply inverting the output alpha.
Remap Alpha Curve	Remap the alpha as the last step in the internal pipeline.
Reset	Reset the alpha remapping curve.



(a) Front input Image (b) Alpha (c) Detect Edge applied

(a) Outer edge distance increased (b) Inner and outer softness added (c) Alpha remapped towards 1.0

Drop Alpha

The Drop Alpha tool is a simple matte modifier that drops the alpha channel from its primary input. This tool has no control parameters but always outputs an RGB image.

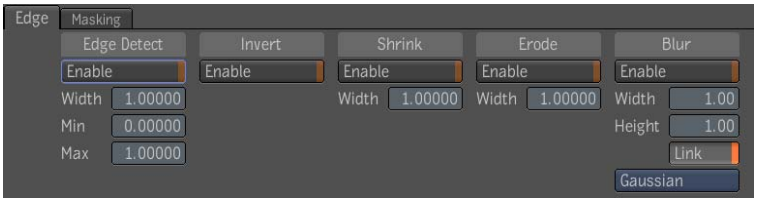
NOTE This implies that if the input is an alpha-only image, the output will be a black RGB image.

Edge Tool

The Edge tool is one of the tools used for matte cleanup tasks during the keying process. It lets you shrink, erode, or blur the edge of the matte, as well as isolate the edge of the matte, or invert the matte.

Because the Edge tool affects the alpha and is applied to the output of a keyer tool, such as the Luma Keyer or the Diamond Keyer, your dependency graph should include, as a minimum, a source and a keyer.

The Edge tool contains the following parameters:



Use:	To:
Edge Detect	Isolates the edge of the key. The numeric field takes values representing the width of the detected edge based on a 0% to 100% float. Edge width can depend on image size.
Invert	Invert the matte.
Shrink	Reduce the edge width. The numeric field takes values representing the pixel width of the shrink filter (from 0.000 to 30.000 float).
Erode	Blend (soften) the light and dark edges. The numeric field takes values representing the pixel width of the erode filter (from 0.000 to 30.000 float).

Use:	To:
Blur	Reduce the edge width. The numeric fields take values representing the pixel width of the blur filter along the X-axis and the Y-axis (from 0.000 to 30.000 float). ■ Link: Blurs the width and height proportionally. Default is on. ■ Gaussian: Default is on, but can be changed to a box blur filter.

Extract Alpha

The Extract Alpha tool lets you extract the alpha channel from its primary input and has no control parameters. This tool always outputs an alpha-only image, implying that if the input is an RGB-only image, the output will be a constant 1.0 alpha image.

NOTE The Extract tool will extract a selected channel into an alpha-only image. The Extract Alpha tool is just a more convenient and readable tool to use when the selected channel is always the input alpha channel.

Fade Tool

The Fade tool provides a convenient way to modify the transparency of an image. The usual case will be to fade an image without having to do it at the same time as compositing. This tool fades an image by reducing the value of its alpha channel, making it transparent.

The Opacity parameter controls the fading effect; at 100% (the default value) the image is unchanged; smaller values fade more; at 0% the image is completely transparent.

This tool in an image modifier; it restricts processing to the masked region and propagates it to the output. The Fade tool can be muted, as well as animated and masked; it affects the alpha channel only.

NOTE If the input image has no alpha channel, one will be added. This is consistent with Composite's general behavior with respect to missing channels. A missing alpha channel is interpreted as a virtual solid opaque alpha channel.

Invert Alpha

The Invert Alpha tool inverts the alpha channel of its primary input and has no control parameters. The Invert tool is already capable of inverting the alpha channel of its primary input. However, Invert Alpha can achieve this without the need to select the proper affect target and makes the Schematic easier to understand.

Median Alpha

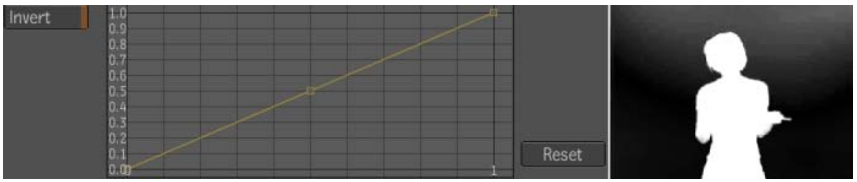
The Median Alpha tool is almost identical to the Median Tool (see Median Tool on page 406) and is excellent for removing impulse noise, but is designed to operate on the alpha channel of the input image.

It has all the parameters of the Median tool except for the Criteria parameter, which is not needed because this tool always computes the median value by ranking the alpha value of each pixel.

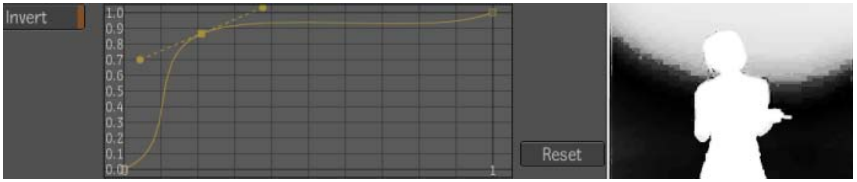
NOTE This tool affects only the alpha channel.

Remap Alpha

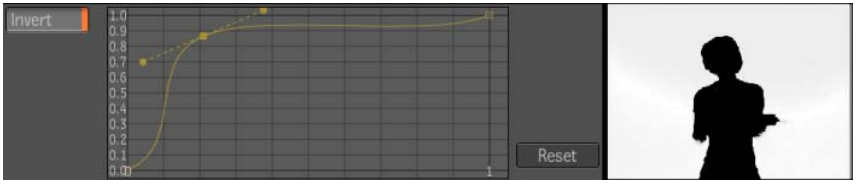
The Remap Alpha tool lets you remap the alpha using a single curve manipulation interface similar to Blend Curves. The curve defaults to the identity mapping.



Original matte



Alpha remapped



Alpha inverted

An invert button provides inversion of the alpha after the curve mapping. This tool affects alpha only; the RGB channels, if present, are passed through unchanged. The pixel format of the output is the same as the pixel format of the image input. If the image has no alpha, then the output will have an alpha channel, properly modified by the curve remapping.

For more detailed curve manipulation, control points can be added to, or deleted from the curve.

To add a control point to the curve:

➤ With the cursor in the curve window, press + (plus sign) to enter a mode where each click on the curve creates a control vertex at that location.

To delete a control point from the curve:

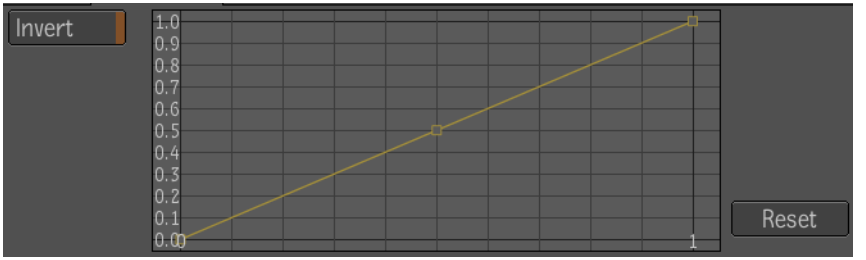
➤ With the cursor in the curve window, press - to enter a mode where each click on a control vertex deletes the control vertex at that location.

To exit the Add or Delete control point mode:

➤ Press **Esc**.

NOTE This tool can be masked.

The Remap Alpha tool has the following parameters:



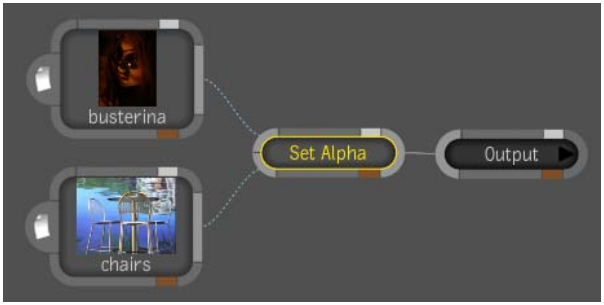
Use:	To:
Invert	Invert the alpha after the curve mapping.
Curve	Gesturally manipulate the alpha remapping.
Reset	Reset the remapping curve.

Set Alpha Tool

The Set Alpha tool lets you create (or replace, or add to) an alpha channel for an image (image A) input based on a component from a second image (image B). The selected component from image B may also be blended with the alpha component from image A.

To set a new, or replace an existing, alpha channel in an image:

1. To view the results of this operation, set the Channels selection to RGBA in the Composition tab of the Tool UI, or in Player Display Options, select Tool Output in the Display field and Channels: RGB in the View field.
2. Drag the Set Alpha tool from the Tools tab onto the dependency graph in the Schematic view.
3. Connect the images to the node (image A and image B).



4 Select the component from image B that you want to add to the alpha component from image A.

5 Blend the components until you are satisfied with the result.

NOTE The Set Alpha tool's Blend attribute is animatable—see Animation Concepts on page 660.

Channel Processing Tools

You can modify the individual color channels of an image using channel processing tools. These tools are in the Channel folder.

There are five channel processing tools:

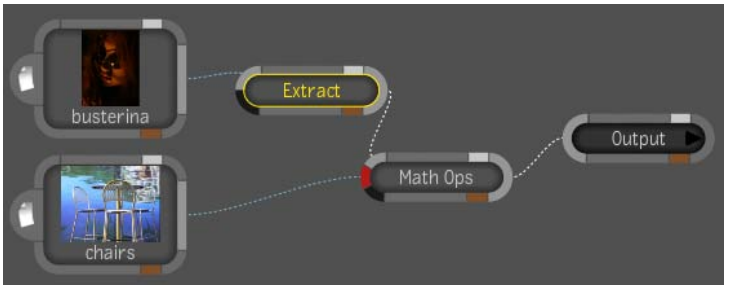
Tool:	Description:
Channel Extract	Remove the red, blue, green, or alpha channel from an image—see Channel Extract Tool on page 362.
Out of Range	Analyze the channels of an image based on a specified thresholds—see Out of Range Tool on page 362.
Channel Replace	Remap the channels in one image by those in another image—see Channel Replace Tool on page 363.
Channel Rewire	Copy component values of an image to component values of an output image—see Channel Rewire Tool on page 364.

Channel Extract Tool

The Channel Extract tool lets you to remove a channel from an image. You can remove the red, green, blue, or alpha channel.

To extract a channel from an image:

1 Drag the Extract tool from the Channel folder on the Tools tab onto the dependency graph in the Schematic view.



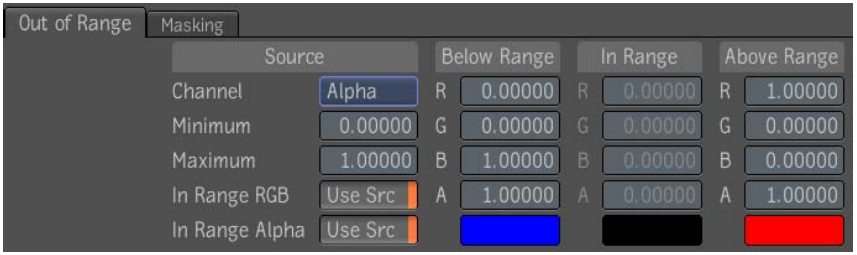
2 Click the channel list button and select the channel you want to extract from the image.

NOTE The Channel Extract tool has a masking input—see Pixel Masking on page 560.

Out of Range Tool

The Out of Range tool uses pseudo colors to display pixels above or below user specified threshold. It is also capable of outputting a mask of the out-of-range pixels or its inverse, so that corrective action can be taken by other tools downstream. The Out of Range tool is in the Channel folder in the Tools tab.

The Out of Range tool has the following parameters:



- **Channel** Select the source image channel to analyze.
- **Minimum** Set the minimum value threshold of the source image channel.
- **Maximum** Set the maximum value threshold of the source image channel.
- **In Range RGB** Select the source image channel to display as the "In Range" color or deselect to use arbitrary color.
- **In Range Alpha** Select the source image alpha channel to display as the "In Range" alpha or deselect to use arbitrary alpha value.
- **Below Range** Select color to display pixels below the specified threshold.
- **In Range** Select color to display pixels within the specified threshold.
- **Above Range** Select color to display pixels above the specified threshold.

Examples

The following examples show you how the Out of Range tool can be used.

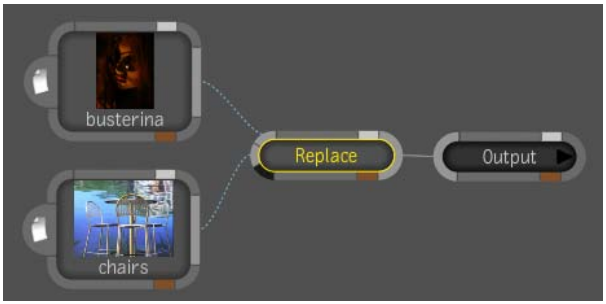
- **Viewing out of range alpha in a player** Set the display modifier as a range display tool. An alpha below range will show up as opaque blue, an alpha above range will show up as opaque red, and all others will have their original alpha value.
- **Generating a matte from out of range alpha** Using the range display tool, take the matte from the output image. Out of range pixels will have alpha of 1.0, all others will have alpha of 0.0.

Channel Replace Tool

The Channel Replace tool lets you remap the red, green, blue, or alpha channel values in one image by those of another image.

To remap the channel values of one image to a different image:

1 Drag the Replace tool from the Channel folder in the Tools tab to the dependency graph in the Schematic view and connect the images to the tool node (one image for input A and one image for input B).



2 In the Tool UI, click the channel you want to replace and select values; A or B for Red, Green and Blue or A, B, or None for the Alpha channel.

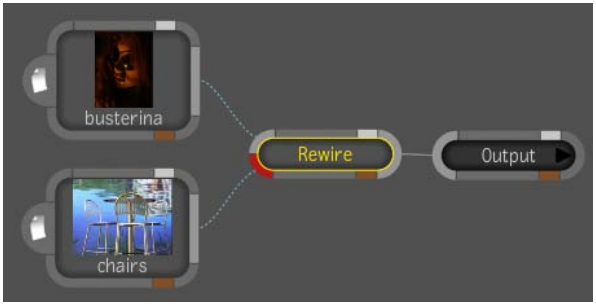
NOTE The Channel Replace tool has a masking input and its attributes are also animatable—see Pixel Masking on page 560 and Animation Concepts on page 660.

Channel Rewire Tool

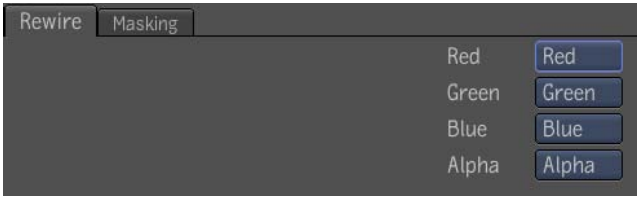
The Channel Rewire tool lets you create an output image that is generated by copying component values of the input image to (possibly different) component values of the output image. For each component of the output image, a selection list is used to choose the component of the input image.

To rewire an image:

1 Drag the Rewire tool from the Channel folder in the Tools tab to the dependency graph in the Schematic view.



2 Select the input component(s) from the selection list.



NOTE When Alpha is selected, a sixth choice, None, is available.

NOTE The Channel Rewire tool has a masking input and its attributes are also animatable—see Pixel Masking on page 560 and Animation Concepts on page 660.

Composition Tools

Use the Composition tools to combine front, back, and matte images using a combination of blending modes and compositing methods.

There are five composition tools:

Tool:	Description:
Blend	Blend two images using a simple blend factor—see Blend on page 366.
Blend & Comp	Composite front and back RGBA images with blend modes and compositing operators—see Blend & Comp on page 367.
Blend Matte	Combine two images using a matte to define visibility—see Blend Matte on page 368.

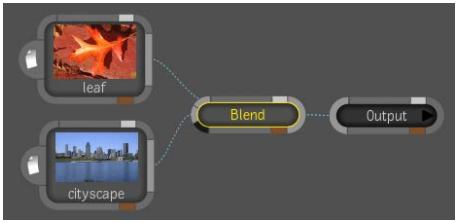
Tool:	Description:
Comp Ops	Apply compositing operators to two source images—see Comp Ops on page 369.
Math Ops	Apply mathematical compositing operators to the RGBA values of two source images—see Math Ops on page 370.

Blend

The Blend tool lets you blend two images by applying a blend factor. The image connected to the A input is displayed when the Blend Factor is set to 0; the image connected to the B input is displayed when the Blend Factor is set to 100. The Blend factor, like all parameters, can be animated, making it easy to use this tool to create a blending transition between compositions.

To blend two images:

1. Drag the Blend tool from the Composition folder in the Tools tab to the dependency graph in the Schematic view.
2. Connect the A and B images to the tool.
3. Set the Blend factor. If you set a value of 0, you see 100 percent of the image connected to the A input. If you set a value of 100, you see one hundred percent of the image connected to the B channel. In the following example, the blend factor is set to 46 percent.



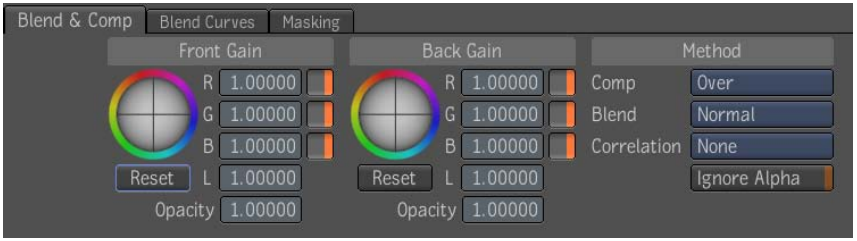
NOTE The Blend tool has a masking input and its attributes are also animatable—see Pixel Masking on page 560 and Animation Concepts on page 660.

Blend & Comp

The Blend & Comp tool is used to composite front and back RGBA images. While most compositing tools composite a front layer over an opaque background under the direction of a matte image, this tool offers full support for RGBA images, both for the front and back inputs, and computes an RGBA result.

You can specify a compositing operator to control the shape of your output and a blend mode to determine how the front and back are combined in the areas where they overlap.

The Blend & Comp tool is in the Composition folder in the Tools tab, and has the following parameters:



Use:	To:
Front Gain	Multiply the front by a color factor prior to using it in the blend. The default is 100%; the range is [0,10].
Front Opacity	Control the opacity of the front in the compositing. If the opacity is less than one, the front will be more transparent and you will start seeing the back through it. The default is 100%; the range is [0,1].
Back Gain	Multiply the back by a color factor prior to using it in the blend. The default is 100%; the range is [0,10].
Back Opacity	Control the opacity of the back in the compositing. If the opacity is less than one, the front will be more transparent and you will start seeing the back through it. The default is 100%; the range is [0,1].
Comp	Determine which compositing mode will be used (the default is Over)—see Compositing Operators on page 352.
Blend	Determine which blend mode will be used (the default is Normal). Click the Blend button to view other available modes—see Blend Modes on page 345.

Use:	To:
Correlation	Specify how the two input mattes are correlated. This can be used to improve the quality of the composite in special cases. For example, if you composite two mattes that share a good portion of their outline, you should let the system know if they are Adjacent or Superposed. By default, the correlation mode is None, assuming that, normally, the input mattes are not correlated.
Ignore Alpha	The blend mode is applied to every pixel in the image and the alpha channels are completely ignored. The output alpha is simply a copy of the alpha of the back input. By default, this option is false.
Reset	Resets the Gain settings to default values.

NOTE The Blend and Comp tool also has a Blend Curves tab—see Blending Curves on page 347.

Blend Matte

The Blend Matte tool lets you mix two images using a matte image to define the areas of visibility between the front and back images.

To blend two images using a matte:

1. Drag a Blend Matte tool from the Composition folder in the Tools tab to the dependency graph in the Schematic view.
2. Connect the front, back and matte images to the tool.
3. Select the Blend Matte Channel: Alpha, Red, Green, Blue, or Luma, as appropriate for the image you are using as the matte. The default choice is Alpha.

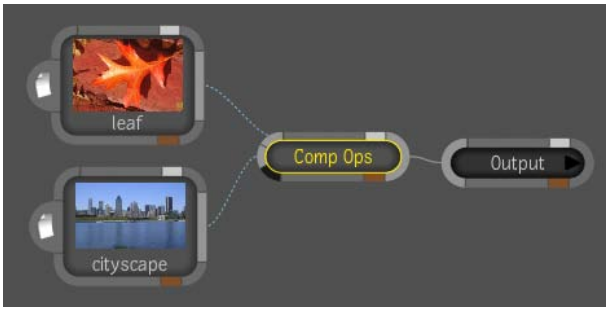
   Select Invert if you want to invert the result.

4. Select the tool output: Composite, Front, or Front and Matte.

NOTE The Blend Matte tool has a masking input and its attributes are also animatable—see Pixel Masking on page 560 and Animation Concepts on page 660.

Comp Ops

The Comp Ops tool lets you apply a variety of compositing operators to two source images.



To apply Comp Ops operations:

1. Drag a Comp Ops tool from the Composition folder in the Tools tab to the dependency graph in the Schematic view.
2. Connect a source image to input A and another source image into input B of the Comp Ops node.
3. Click the Operator button and select a compositing operator.

Select:	To:
A over B	Place input A on top of input B according to the alpha of the foreground image.
B over A	Place input B on top of input A according to the alpha of the foreground image.
A in B	Place the RGBA of input A into the alpha of input B.
B in A	Place the RGBA of input B into the alpha of input A.
A out B	Place the RGBA of input A into the invert alpha of input B.
B out A	Place the RGBA of input B into the invert alpha of input A.

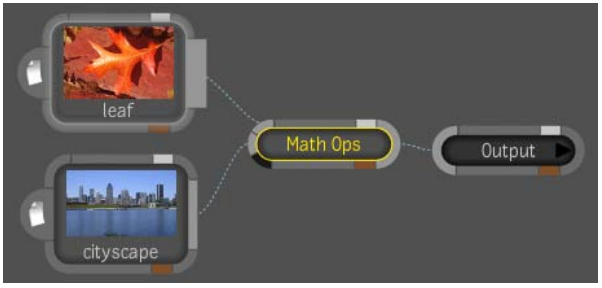
Select:	To:
A atop B	Place the RGBA of input A over the RGBA of input B but through the alpha of input B.
B atop A	Place the RGBA of input B over the RGBA of input A but through the alpha of input A.
A xor B	Create the union of the RGBA of inputs A and B minus the intersection of the RGBA of inputs A and B.

NOTE The Comp Ops tool has a masking input and its attributes are also animatable—see Pixel Masking on page 560 and Animation Concepts on page 660.

Math Ops

The Math Ops tool lets you apply mathematical blending operations to the RGBA values of two source images.

Use the following procedure to apply mathematical blending operations.



To apply mathematical blending operations to an image:

1. Drag the Math Ops tool from the Tools tab to the dependency graph in the Schematic view.
2. Connect a source image to input A and another source image into input B of the Math Ops node.

3 Click the Operator button and select a math operator.

Select:	To:
Add	Add to the RGBA of input A, a percentage (using the Blend value) of the RGBA of input B.
Div	Divide the RGBA values of input A by the RGBA values of input B.
Max	Create a composite image where the final output will be the maximum RGBA values of either input A or input B.
Min	Create a composite image where the final output will be the minimum RGBA values of either input A or input B.
Mix	Mix the RGBA values of input A with the RGBA values of input B: 0% = completely A, 100% = completely B.
Mult	Multiply the RGBA values of input A by the RGBA values of input B.
Screen	Mimic the effect of combining two film negatives together. Both layers are inverted, then multiplied, and finally the results are inverted back.
Sub	Subtract from the RGBA of input A a percentage (using the Blend value) of the RGBA of input B.

4 Blend the images. The blend factor controls the blend between the result of the math operation and the original input.

NOTE The Math Ops tool has a masking input and its attributes are also animatable—see Pixel Masking on page 560 and Animation Concepts on page 660.

Multi-stream Compositing

Use the Streams tools to perform identical processing on multiple image streams avoiding the creation of identical sub-graphs to process a number of images in the same way. The Streams tools can also be used to support a stereo workflow in Composite —see Stereo Viewing on page 374. There are three Streams tools:

Use:	To:
Merge Streams	merge multiple single-stream image inputs into a single multi-stream output. It has a single multi-valued image input socket,

Use:	To:
	and a single output socket. You can connect as many inputs as you like to the Merge Streams tool.
Extract Streams	extract single-stream image outputs from the Merge Stream node using the Selector value editor.
Stereo Streams	input left eye and right eye image streams to create a 2-stream stereo image. The Stereo Streams tool takes two single-stream inputs and combines them into a single dual-stream output, placing the left image in stream 0, and the right image in stream 1.

In the application, the default stream is stream 0. For example, if a color picker is used in the player, it will pick stream 0. If you pick a color in the Keyer, it will pick the color from stream 0. If you click Fit to Source in a warp tool, it will fit to stream 0.

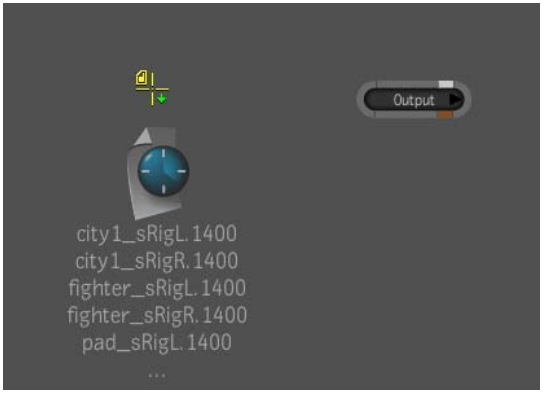
NOTE Paint strokes will only be applied to stream 0.

Performing Multi-stream Processing

You can perform identical processing on multiple image streams. The following example workflow applies a blur to five imported layers.

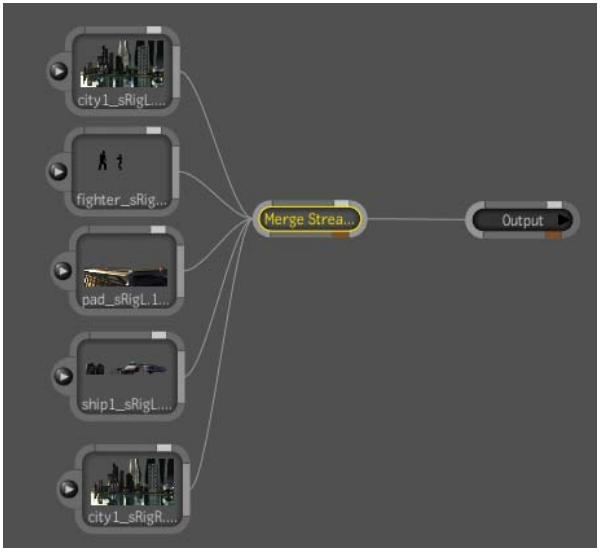
To apply a blur to multiple layers:

1. Import multiple layers into Composite. Composite creates a composition per layer.
2. Perform a multiple selection of the imported layers, drop them into a new composition and swipe through the Link option. Composite creates a link node per layer in the new composition.



3 Drag a Merge Streams tool from the Streams tool folder and drop it into the composition.

4 Connect all layers to the inputs of the Merge Streams node.



5 Drop a Blur tool into the composition and connect the Merge Streams output socket to its input then change the blur parameters.

The blur is applied identically to all five streams.

NOTE To view the individual streams, use the Player 0, (next stream) and Shift + 0 (previous stream) to cycle to the next and previous streams produced by the Player target. You can also view a stream by swiping through the Player south gate, opening the Streams tab and selecting the stream you want to view from the Stream value editor.

Stereo Viewing

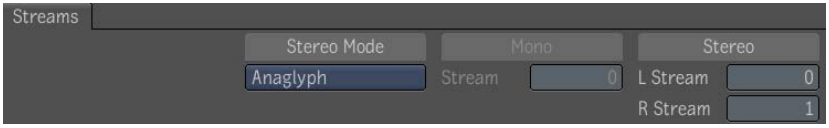
Viewing stereo image pairs in Composite is supported through an anaglyph display mode in the player, and by quad-buffered, 3D DLP, and left/right segmented frame plug-in video preview devices.

If you do not have stereo footage available, you can use the Reaction tool to create new footage by rendering a scene using a stereo camera rig—see Setting up a Stereo Camera Rig on page 293.

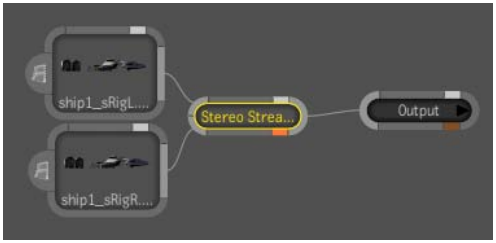
An anaglyph is a stereoscopic motion or still picture in which the right component of a composite image usually red in color is superposed on the left component in a contrasting color to produce a three-dimensional effect when viewed through correspondingly colored filters in the form of glasses. Anaglyph mode is applied after the display modifier. For example, if the display modifier performs gamma encoding, or tone mapping, the result is encoded for stereo display using an anaglyph after the gamma encoding or tone mapping.

To view a stereo image pair in stereo anaglyph mode in a player view:

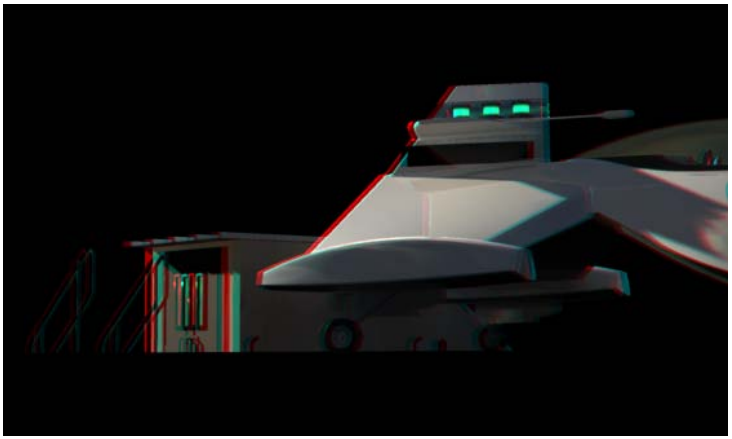
1. Swipe south in a Player view and select the Streams tab.
2. Select Anaglyph from the Stereo Mode menu.



3. From the Tools tab, select Stereo Streams from the Streams folder and drag it to the Schematic view.
4. Connect the stereo image pair left image to the left input of the Stereo Streams node and the right image to the right input of the Stereo Streams node.



The stereo image pair is displayed in anaglyph mode in the player. You can view the image with inexpensive tinted glasses.



Composite 2011 supports special-purpose stereo viewing hardware, such as quad-buffered stereo monitors, or 3D DLP (TM) monitors. These stereo viewing devices are supported through Composite video preview devices. You must select a video preview device that matches your stereo viewing hardware from the Video Preview device list in the User Preferences window. The default video preview device is None.

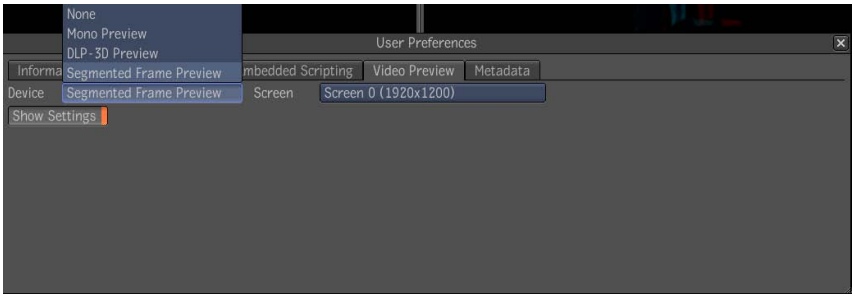
Select:	**To:**
None	keep the default setting. Use None when you do not want a video preview device to be active.
Mono	view a video preview device that is just a window that shows one stream. You can put the window on another monitor if you have one. You can then view the contents of a player in a window on a second monitor. If you only have a single monitor, use **Alt** + **Tab** to view it.

Select:	To:
DLP-3D Preview	view stereo pairs using DLP -3D monitors that support active shutter glasses—see Active Shutter Glasses on page 378.
Segmented Frame Preview	view stereo pairs using circular polarization monitors with passive polarizing glasses—see Polarizing Glasses on page 377.
Quad-buffer	view stereo pairs with polarizing glasses using monitors based on linear polarization —see Polarizing Glasses on page 377.

NOTE Quad-buffer will only appear in the Device list if you have a second graphics card to output the left and right streams.

To view stereo pairs in video preview devices other than the Player:

1. Select the Video Preview device from the Device list from the User Preferences tab. If your video preview device is None, a Stereo video preview device will show nothing.If you switch to a stereo video preview device and you don't have a stereo pair, the stereo video preview device will show the same image in the left and right eyes.



2. Change the stereo mode to Segmented Frame Preview
3. Click in a Player view to make it current. The stereo pair is displayed.
4. Select a node in the dependency graph and change its parameters. Changes are made in the 3D scene both in the Player and in the video previewing device.

Supported Stereo Output Graphics Hardware

The following section describes some of the multiple display configurations possible on a Composite workstation.

Composite supports systems with standard graphics cards with DVI outputs.

- **Single graphics card, single DVI output** This configuration implies that stereo will be shown in the Composite UI. Therefore, stereo display is restricted to anaglyph in the Player view.
- **Single graphics card, dual DVI output** A typical configuration would be one DVI output to drive the Composite UI, and another to drive a single-input stereo display as a Composite stereo video preview device.
- **Dual DVI output plus single DVI output graphics cards** The single DVI output card drives the Composite UI, and the dual output card drives a dual DVI input stereo display as a stereo video preview device.
- **Dual dual DVI output graphics cards** As per the Dual DVI output plus single DVI output graphics cards descriptions above.

Supported Technologies

The following section describes both the viewing glasses and type of stereoscopic display monitors used with the glasses that are supported by Composite.

NOTE If your stereo video preview device only support 8 bits per color component, consider using a player view display modifier that can provide 8 bit output. Examples include the 1D LUT, 3D LUT, and Convert Depth tools.

Polarizing Glasses

Passive polarizing glasses are worn by the viewer. The display device shows left and right images simultaneously. Devices that use this technology include projectors and monitors. Examples of these devices are:

- **Circular polarization** Segmented Frame Preview mode monitors that use circular polarization. Its disadvantage is half vertical resolution in 3D (1920x600).
- **Linear polarization** Quad-buffered stereo mode displays use two monitors and linear polarization to create a full-resolution stereo image. In such

systems, the observer head orientation is important to avoid ghosting artifacts.

Active Shutter Glasses

The viewer wears LCD glasses that open the left or the right eye in synchronization with left and right images shown by the display device. The synchronization signal is broadcast by an emitter to the LCD glasses. Devices that use this technology include projectors and monitors. Examples of these devices are:

- **DLP monitors** DLP monitors that support active shutter glasses. The left and right images are shown at double update rate (120 Hz), using a checkerboard pattern. Their disadvantage is half horizontal resolution.

Passive Colored Glasses

- **Anaglyph** Left and right images are tinted red and cyan, and viewed using inexpensive tinted glasses. An important disadvantage is poor color reproduction.

Image Processing Tools 17

Topics in this chapter:

About Image Processing Tools

This chapter describes the image processing tools available to you in Composite. They are described in the order in which they appear in the Tools tab.

Filtering Tools

You can use Filtering tools to apply a range of looks, feels, and transitions to your composites.

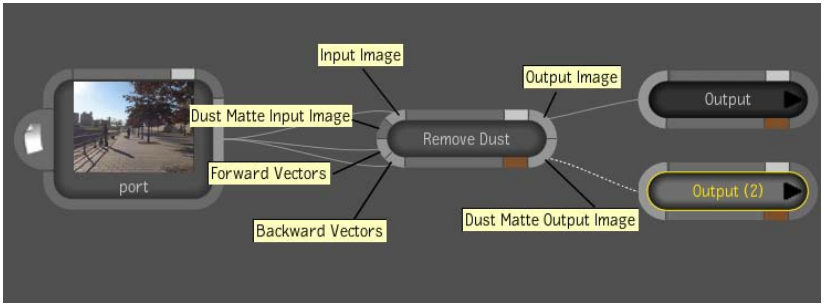
Tool:	Description:
Remove Dust	Removes dust, dirt, hair and scratch artifacts from sequences of images—see Remove Dust Tool on page 380.

Tool:	Description:
Blur	Applies a blur to an image—see About Blurs on page 385.
Lens Blur	Accurately simulates the defocus effect of a real camera on an HDR image—see Lens Blur Tool on page 400
Median	Removes impulse noise with an edge-preserving smoothing filter—see Median Tool on page 406.
Sharpen	Increases clarity of an image—see Sharpen Tool on page 408.
Unsharp Mask	Provides more sharpening control of fine detail in an image—see Unsharp Mask on page 408.

Remove Dust Tool

Use the Remove Dust tool to remove dust, dirt, hair, and scratch artifacts from sequences of images. Dust removal, encompassing what is sometimes called dust-busting, dust repair, scratch removal and scratch repair, is traditionally a time-consuming task, requiring visual inspection and manual correction steps for each frame of film or video. This tool provides you with a means to automate this task as much as possible, and to easily tweak the results manually. The tool automatically detects dust and scratches, and automatically repairs them. You can view the results of the detection phase and easily correct anything using a simple mouse stroke.

The Remove Dust tool detects dust in images and removes the dust from the images. The tool has a primary color image input and outputs two images, the repaired color image and a mono-channel defect matte. An optional second input can be used to specify the defect matte as a mono-channel image, and any dust detected by the tool is added to the dust supplied by the defect matte. If the defect matte input is not present, the tool initializes the defect matte to empty. Optional third and fourth inputs supply forward and/or backward motion vectors to the tool which it can use for the detection and/or the correction phase.



Dust Removal Workflow

The dust removal workflow can be divided into three steps:

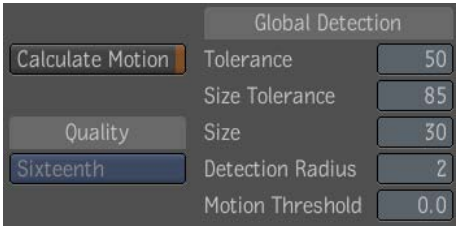
- **Dust Detection** This automatically creates the defect matte by examining the images, using motion vectors if available. This step is optional, as the defect matte may be supplied to the tool as the second input. Even if no defect matte is supplied, you may choose to bypass detection and perform all labeling by hand. A single tolerance parameter controls the dust detection, where a value of 0 means no dust is detected, and a value of 100 means all pixels are dust. You can tweak the value slightly from its default of 50 to get a reasonable set of dust pixels. Two other parameters control the expansion of the dust to make sure the whole dust object is covered, not just the center.

- **Manual correction of the defect matte** If any pixels have been incorrectly labeled as dust or not dust in step 1, manual correction of the defect matte can be performed by drawing appropriate shapes on the dust matte. These locally change the detection and repair parameters within the geometric region of the shape. Each object drawn has its own dust detection parameters, which overrides the global (automatic) values used in step 1. In addition, each object has other parameters to control how correction is performed. Of course, drawing shapes is optional.

- **Defect repair** Using the final defect matte, an image processing operation is performed to fill in the corresponding pixels in the output image with corrected pixels. This process uses motion vectors if they are available. Your can choose between a spatial repair, which uses only the current frame, or a temporal repair, the default, which uses neighboring frames and motion vectors. Spatial repair should only be used in areas of the image where the motion vectors are inaccurate or there are occlusions making it difficult for the algorithm to find the corresponding correct pixel on neighboring frames.

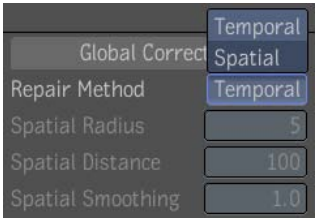
Global Detection and Repair

The Global Detection UI has the following parameter controls:



Use:	To:
Calculate Motion	calculate forward and backward motion vectors.
Quality	control the quality of the motion vectors by applying the motion analysis only to lower-resolution versions of the input image, up to the resolution specified by the quality parameter.
Tolerance	determine how many pixels are classified as dust. If the Tolerance is set to 0, then no dust is detected, effectively disabling the dust detection. If a dust channel from a film scanner is applied, the Tolerance can be set to 0 to avoid dust detection. Otherwise, the dust channel will be the union of the scanner matte and the detected matte. If the Tolerance is set to 100, then all pixels are dust. A value of 100 might be used with a shape to label all pixels within a small region as dust.
Size Tolerance	control into which regions the dust is expanded. A value less than or equal to Tolerance means do not expand.
Size	control how expanded the detected dust is expanded. A value of 0 means do not expand.
Detection Radius	control the smoothing effect on the first step of dust detection where every pixel is assigned a likelihood of being dust.
Motion Threshold	remove false dust in areas of motion, as its value is raised.

The Global Correction UI has the following parameter controls:

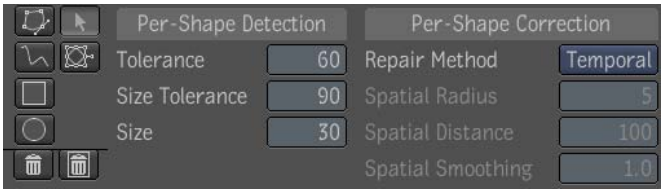


Use:	To:
Repair Method	select either the Temporal or Spatial repair method. The default method is temporal repair. This means to replace a dust pixel with the average of the corresponding pixels in the previous 2 and next 2 frames. If motion vectors are supplied on the 3rd or 4th input, they will be used to define the correspondence; otherwise, the corresponding pixel on another frame is just the pixel at the same position as the dust pixel. Spatial repair is used in areas where the motion vectors are incorrect or the previous and next frames do not contain the corresponding pixel needed (due to occlusion or intensity changes). In this case, the dust is filled using texture infilling and copies reasonable pixels from other places within the same frame. Note that there are no parameters for the Temporal repair method. When the Temporal repair method is selected, the Radius, Distance and Smoothing parameters will be grayed out.
Spatial Radius	set how large a texture patch to use for matching areas around the dust.
Spatial Distance	set how far to search from the dust to find a pixel to put in place of the dust.
Spatial Smoothing	set how smooth to make the infilling of the dust.

Per-Shape Detection and Repair

Occasionally there will be parts of the image where some of the Global (automatic) parameters need to be changed locally to improve the dust detection and repair. For this, you can use a shape drawing facility based on the Garbage Mask tool. The UI is very similar to the Garbage Mask but does not have the mask list user settings or edge gradients, however all Garbage Mask hot keys affecting control points are the same—see Masking on page 537. The objects that are drawn are non-animated and appear only on one

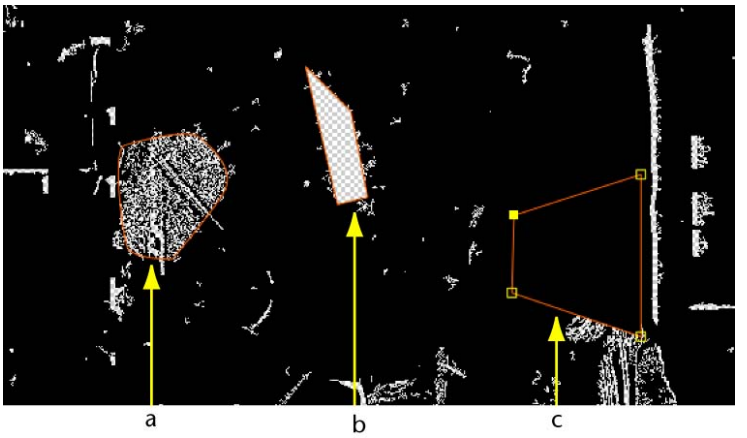
frame of the composition. You can select and edit the objects only by direct manipulation on the image wireframe overlays.



When in creation mode (either Rectangle, Ellipse, or Spline is selected) you can set the Detection and Correction values that will apply only within the shape to be drawn. When in selection mode (either Select or Transform), the UI looks the same as in creation mode, except that the values are for the currently selected shape. The two garbage can buttons provide deletion of selected shapes, and all shapes on the current time frame, respectively.

Each drawn shape has several values associated with it, the first 3 of which override the corresponding global (automatic) parameter in the area of the shape:

Use:	**To:**
Tolerance	override the global detection tolerance.
Size Tolerance	override the global size tolerance.
Size	override the global dust expansion amount.
Repair Method	override the global repair method. The default is Temporal. You would only switch to Spatial Repair in areas where the motion is so complicated that the motion vectors are incorrect. Note that there are no parameters for the Temporal repair method. When Temporal repair method is selected, the Radius, Distance and Smoothing parameters will be grayed out.
Spatial Radius	set how large a texture patch to use for matching areas around the dust.
Spatial Distance	set how far to search from the dust to find a pixel to put in place of the dust.
Spatial Smoothing	set how smooth to make the infilling of the dust.



(a) Tolerance and Size Tolerance raised to label more dust in this area. (b) Tolerance and Size Tolerance of 100, to label the entire interior as dust. (c) Tolerance and Size Tolerance of 0, to label the entire interior as not dust.

Defect Matte Viewing

The defect matte is output on the second output, thus allowing the use of context points—see Setting Context Points on page 191 to view the input defect matte and the defect matte after dust detection has been applied. You can get the second output node from the Utilities folder in the Tools tab.

About Blurs

Use the Blur tool to finish shots that require directional, radial, modulated, and vectors blurs. This includes shots that require some amount of depth of field or motion blurs. In modulated blurs, you can vary the amount of blur from pixel to pixel. The ability to vary the amount of blur applied at each pixel is sometimes useful to model specific physical processes or be used for purely artistic goals.

Blur Tool UI

The Blur tool UI is composed of five tabs:

Tab	Controls
Blur	Settings for the directional Gaussian blur and the Radial blur—see Directional Gaussian and Radial Blur on page 386.
Modulation	Settings for the Modulated blur—see Modulation Blur on page 391.
Vectors	Settings for the Vectors blur—see Vectors Blur on page 395.
Output	Settings affecting the image output by the tool—see Output Controls on page 399.

Directional Gaussian and Radial Blur

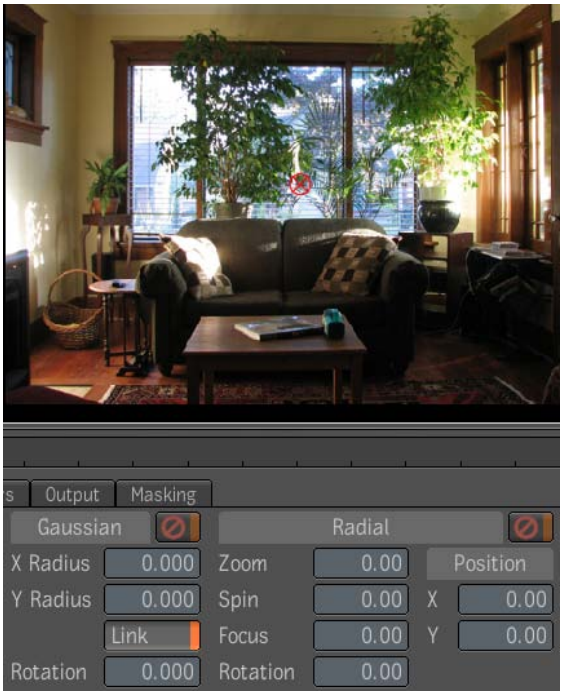
The Directional Gaussian blur filter lets you scale and rotate its elliptical shape. This allows you to blur an image by a certain amount in a given direction, and by a different amount in a direction perpendicular to it. Each pixel is blurred by the same amount.


Output
Masking
Gaussian
X Radius
0.000
Y Radius
0.000
Link
Rotation
0.000
Radial
Zoom
0.00
Spin
0.00
Focus
0.00
Rotation
0.00
Position
X
0.00
Y
0.00

Use:	To:
X Radius	Set the amount of blur to apply in direction of the X axis (before taking into account the rotation parameter).
Y Radius	Set the amount of blur to apply in direction of the Y axis (before taking into account the rotation parameter).
Link	Couple the X Radius and Y Radius so that when you change the X Radius or the Y Radius the other changes in the same proportion.
Rotation	Rotate the X and Y axis of the Gaussian by a given angle. The angle is specified in degrees.
Mute	Mute the Directional Gaussian Blur contribution of the Blur tool. The other blurs are still applied (if not also muted).

The Radial Blur simulates the effect of motion blur that would be generated by the movement of the camera due to zooming and spinning around a given point. The zooming and spinning movements are simply converted into motion vectors representing the local displacement at each pixel. These motion

vectors are then used to construct blur ellipses aligned with each vector. The size of the major axis matches the length of the displacement vector. The size of the minor axis is kept fixed at a small value. That value is chosen to keep aliasing artifacts at an acceptable level while not introducing too much blurriness.



Use:	To:
Zoom	Set the amount of blur to apply in the radial direction (before taking into account the rotation parameter). The amount of blur is specified in degrees; the further a pixel is located from the center of the radial blur, the more it is blurred. Expressing the Zoom parameter in degrees allows it to share the same units as the Spin parameter. This lets you express a given amount of blur in either the radial or the tangential direction.
Spin	Set the amount of blur to apply in the tangential direction (before taking into account the rotation parameter.) The amount of blur is specified in degrees; the further a pixel is located from the center of the radial blur, the more it is blurred.
Focus	Controls how the strength of the blur relates to distance from the center of the effect. Increasing the amount of focus keeps the center of the effect in focus at the expense of the outer rim. With a Focus value of zero, the strength of the blur increases linearly with distance from the center.

Use:	To:
Rotation	Rotate the radial and tangential direction by a given angle. The angle is specified in degrees. This causes the blur effect to spiral.
X,Y Position	Specify the center of the radial blur. You can also click and drag the red manipulator in the player to set the location of the center of the blur.
Mute	Mute the radial blur contribution of the Blur tool. The other blurs are still applied (if not muted then themselves).

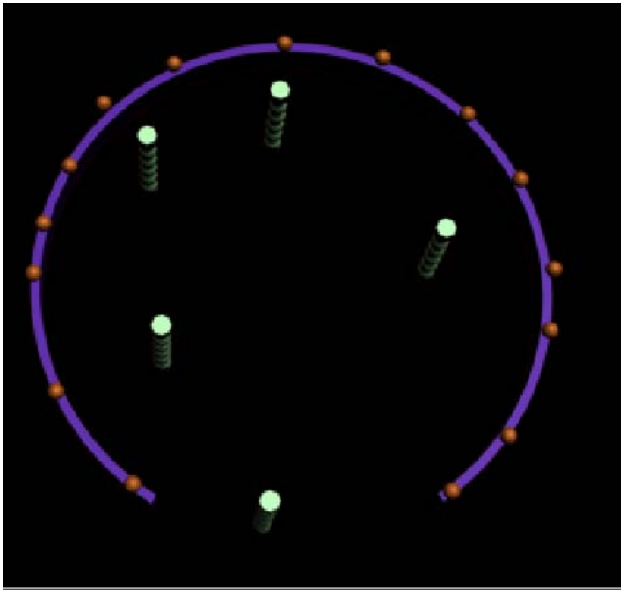
Modulation Blur

A particular color channel (red, green, blue, alpha or luminance) of the modulation input image can be used to modulate a modulated motion blur. The end effect is that the modulation image will apply a varying amount of scaling at each pixel. The elliptical shape of the blur filter is scaled uniformly by this modulation amount.

The Modulation blur tool has the following parameters:

Use:	To:
X Radius	Set the amount of blur to apply in direction of the X axis (before taking into account the rotation parameter).
Y Radius	Set the amount of blur to apply in direction of the Y axis (before taking into account the rotation parameter).
Link	When enabled, changing the X Radius or the Y Radius causes the other one to change in the same proportion.
Rotation	Rotate the X and Y axis of the Gaussian by a given angle. The angle is specified in degrees.
Channel	To specify which channel of the “Modulation Image” is used to control the modulated blur. The valid channels are: Red, Green,

Use:	To:
	Blue, Alpha, Luma and None. When None is selected, the image is blurred as if the Modulation Channel had a constant value of 1. Note that this behavior is different than muting the entire modulated blur. This is an aid for trying to figure out what the various parts of the tool contribute.
Offset	To offset the selected modulation channel before it affects the modulated blur. The offset is subtracted from the channel. The output image will be in focus where the modulation channel is equal to the modulation offset; it will be progressively blurrier for values of the modulation channel above and below the modulation image. When the modulation tab is displayed and the Player is in Display Tool output mode, selecting the offset picker tool then clicking anywhere in the player will set the Offset parameter to the value of the selected modulation channel under the pointer. This is useful for selecting the in-focus object in a scene.



View from top showing relative positions of objects in the scene.

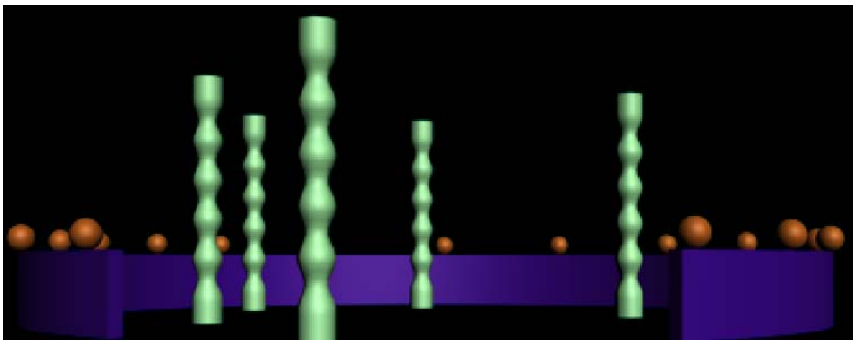
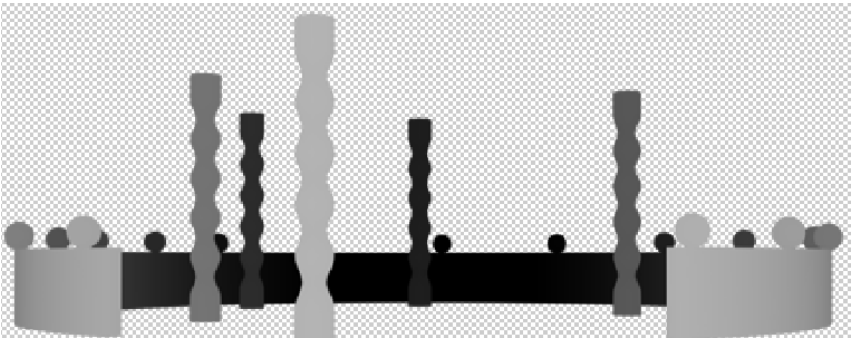
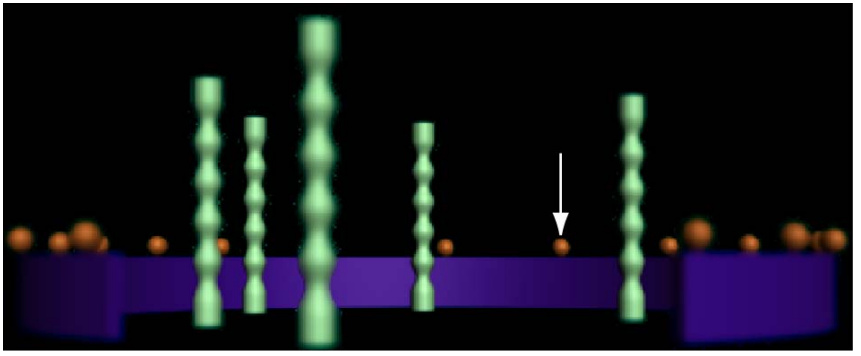


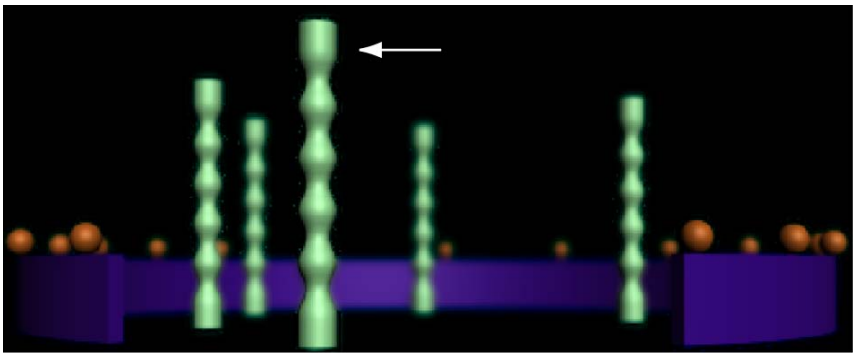
Image with no modulated blur.



Z Depth modulation image - closer objects appear as lighter shades of gray; farther objects appear darker.



Luma channel offset value of sphere is 0.011 and matches that of the modulation image retaining focus.



Luma channel offset value of column is 0.704 and matches that of the modulation image retaining focus.

Although the Modulation Blur tool can simulate depth of field, artifacts may appear for a number of reasons and are inherent 2.5D motion blur and depth of field.

Depth of field is a process that occurs in 3D. Out-of-focus objects blur on top of objects that stand further away. They never blur on top of objects in front of them. The modulated blur does not make this distinction about objects that are in front or behind thus causing artifacts every time a silhouette edge appears in an image. Rendering depth of field therefore requires segmentation of the image along the silhouette edges. Please note that it is not always possible to partition a scene into separate objects using silhouette edges.

Anti-aliased images also cause some problems in that the Z-channel cannot be used to determine which portion of the pixel belongs to the background object and which part belongs to the foreground. When an object becomes out of focus, one starts to see parts of the scene that were not visible before through the blurred edge. To replicate that effect as a 2D post-process, the depth of field tool must guess what the background pixels might look like behind foreground objects. To do this, segmenting and matting out foreground objects and then reconstructing the background using in-filling techniques become necessary, which can be a difficult task.

To correct some of the Z-channel problems, the 3D elements of a scene can be rendered as separate layers. The scene would be manually segmented into objects all having approximately the same Z. A single layer would not have any significant Z discontinuities. Each layer can be properly anti-aliased. The Z-information of the foreground element and the background element would be available at partially covered edge pixels and the color of the background objects behind foreground elements would also be available. However, the Z-information is not defined at the transparent pixels of each layer. A renderer is likely to assign them some very far-away value. To the modulated blur tool,

this represents a huge discontinuity in its blur modulation and the Z-discontinuity ends up as a discontinuity in the blurred image. To help remove the resulting artifacts, the parameters (Min. Alpha and Max Radius) under the Extend label can be set to generate the missing Z information.

Use:	To:
Min Alpha	Select at which alpha value to start generating the missing modulation values. Anything opaque is unaffected up to the Min. Alpha value.
Max Radius	Increase the radius (in pixels) of the modulation value generation. Increase this value until the artifacts are removed. Note that increasing the Max Radius value beyond the point where the artifacts are removed will result in slower processing.

Vectors Blur

Use the Vector Blur tool to specify the length and the width of the blur ellipses. The blur ellipses are then oriented independently for each pixel in the direction of the vectors of the Forward Vector input image and uniformly scaled by the length of the vectors. You can also apply an extra constant rotation to all the blur ellipses.

Using this definition of vector blurs you can blur only in the direction of the vector field. To do this, set the amount of blur in the perpendicular vector direction to zero. No matter how much it is scaled, it will always stay null. Then, use the minimum blur radius control to clamp to a finite value to avoid sampling aliasing.

NOTE If the ROD of the Forward Vectors image is smaller than the ROD of the image produced by the Blur tool, the pixels lying at the edge of the Forward Vectors image are repeated to cover the missing region.

The Vectors blur tool has the following parameters:

Use:	To:
Length	Set the amount of blur to apply in direction of the forward vectors (before taking into account the rotation parameter). The amount of blur applied per pixel is also modulated by the length of the vectors.
Width	Set the amount of blur to apply in a direction perpendicular to the forward vectors (before taking into account the rotation parameter.) The amount of blur applied per pixel is also modulated by the length of the vectors. It would often be left at 0, so that the vector length would not affect it.
Link	When enabled, changing the Length or Width parameter causes the other one to change in the same proportion.
Rotation	Set the amount of vector rotation around their own origin.
Reference	Enable the reference vector offset. If reference is set, the vector at the Reference Position is subtracted from all forwards vectors in the image. The result is that the pixel at the reference position will not be blurred, and so will pixels with similar forward vectors.
Weight	Set the amount of blur to apply at the reference position. A value of zero means that no blur should be applied. This is the default. A value of one means to blur by the same amount as the forward vectors at the Reference Position. This is equivalent to disabling the use of a reference vector offset. A value of one half means to blur half as much as the forward vectors at the Reference Position.
Reference Position X and Y	Set the nominal coordinates of the reference position. When the reference is enabled, a manipulator is displayed in the player to allow the user to interactively set the reference position by simply dragging the manipulator (click-drag-release). The reference position can also be set using the Tracker.

Use:	To:
Mute	Mute the Vector Blur contribution of the Blur tool. The other blurs are still applied (if they are not themselves muted).

Using the Modulation blur tool to simulate motion blur can be difficult due to some of the same inherent issues as when trying to simulate depth of field using the Modulation blur tool;

- Need to segment out at silhouette edges.
- Can not deal with anti-aliased images at silhouette edges.
- Needs the background pixels behind foreground elements.
- Discontinuity in the motion vector field ends-up as discontinuity in the blurred image.
- Need to out-fill motion vectors over transparent region.

These limitations can be worked around, if the 3D department has already partitioned the image into layers of constant movement. You can then use the Min. Alpha and Max Radius parameters of the Vectors blur tool to extend the motion vectors field to get more realistic motion blur effects.

Use:	To:
Min Alpha	Select at which alpha value to start generating the missing vector information.
Max Radius	Increase the radius (in pixels) of the vector field generation. Increase this value until the artifacts are removed. Note that increasing the Max Radius value beyond the point where the artifacts are removed will result in slower processing.
View Vectors	Toggle the vector view.
Interval	Set the interval between shown vectors. By default, this value is 16 (i.e. show a vector every 16 pixels). This controls the density of the displayed vectors.
Display Scale	Set the display Scale of the shown vectors. By default, this value is 1.0 (i.e. show a vector with its original length). This controls the displayed length of the vectors. These values only control the player display. They have no influence on processing.

Use:	To:
Color	Set the color of displayed vectors. Click on the color pot to display the color picker—see Color Picker on page 154.

NOTE The Blur tool has a masking input—see Pixel Masking on page 560. The Blur tool's X and Y attributes are both animatable by setting keyframes or using expressions—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Vector Motion Blur with Extended Alpha

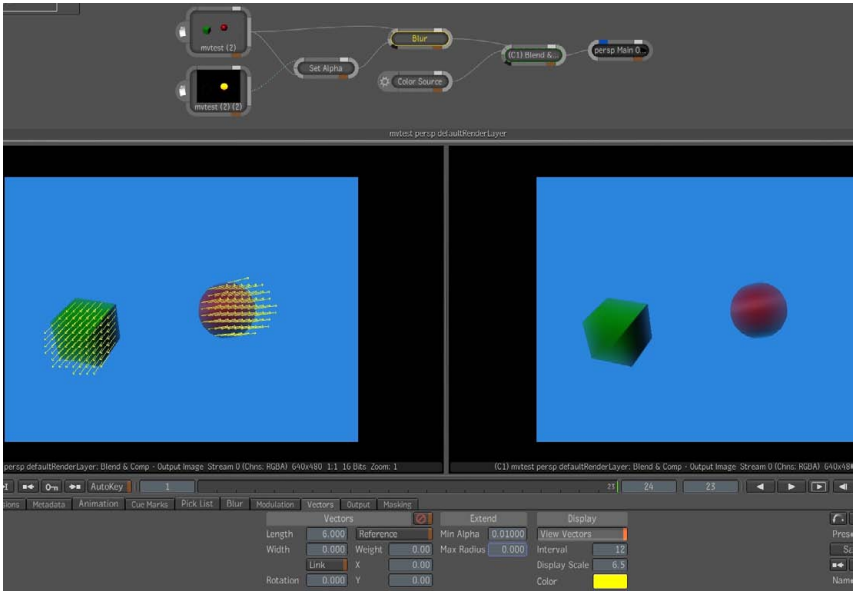
The extended functionality of the Blur node requires an alpha channel on its motion vector input. Since Maya does not allow you to generate motion vectors with an alpha channel, you should copy it from your diffuse/beauty pass in Composite using a SetAlpha node. This should give you the desired result.

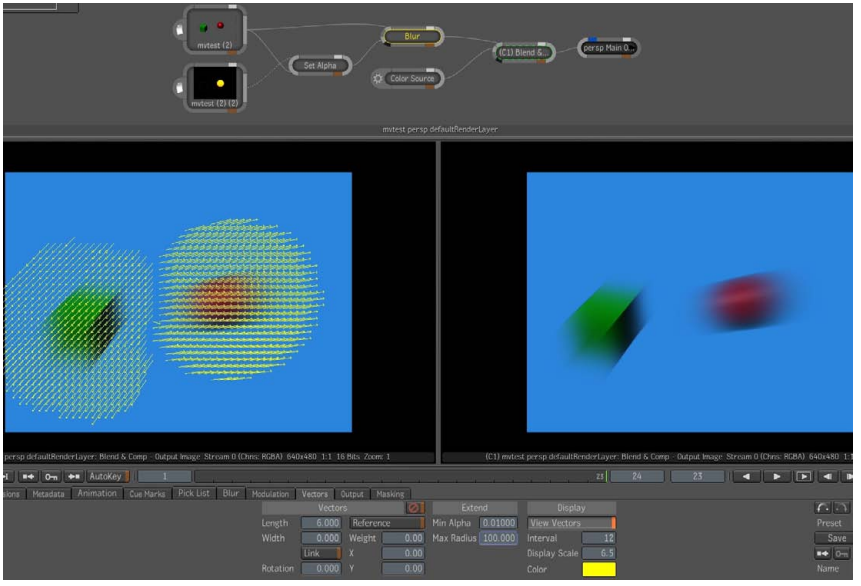
You can see the effect of extending the motion vectors outside of the moving object by turning on the view vectors option in the Vectors tab of the Blur tool.

The Blur node interprets the alpha channel of the Input Image input differently than the alpha of the motion vectors. This allows you to selectively motion blur objects of an opaque RGB image. Ensure that OpenEXR files are used when rendering motion vectors in Maya.

The first of the following two images displays the Blur node without the Alpha Extend, the second is with the Alpha extend.

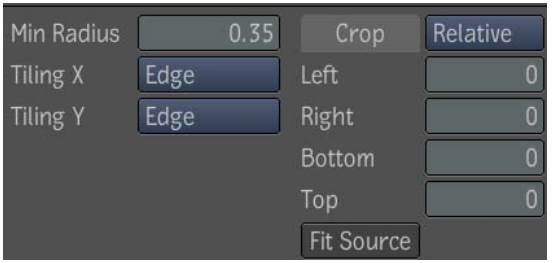
NOTE To extend the vectors beyond the edge, you must increase the Max Radius.





Output Controls

The Output UI allows you to control the following parameters:



Use:	To:
Min Radius	Specify a minimum of blur to apply in any direction. This is useful to minimize aliasing artifacts that might occur when performing blurs that exhibits a high degree of directionality.
Tiling X and Y	Specify how the input image should be extended outside its region of definition (ROD). This is important for two reasons. First, the convolution kernel of the Blur tool might need to access source pixels outside the ROD to produce pixel falling inside the source ROD. Secondly, the Blur tool allows the output image to be uncropped arbitrarily. The supported tiling modes are Edge (default), Transparent, Repeat, and Mirror.
Crop	Crop (and uncrop) the image produced by the Blur tool. The cropping controls work exactly like the ones found in the Crop tool with the exception of the addition of an Auto Crop mode. When in Auto Crop mode, the Left, Right, Bottom and Top controls are greyed out. Instead the output ROD is computed automatically based on the following assumptions; only the parameters of the Gaussian Blur are used to determine the output ROD. This includes the Gaussian X radius, Y radius and Rotation; the Input ROD is enlarged by an amount proportional to the rotated" X and Y radius. It is enlarged sufficiently so that the profile of the Gaussian filter drops sufficiently close to zero at the boundary of the enlarged kernel. This behavior is especially useful when the X and Y Tilling modes are set to Transparent. The supported crop modes are Relative (default), Window, Absolute, and Auto Crop.

Lens Blur Tool

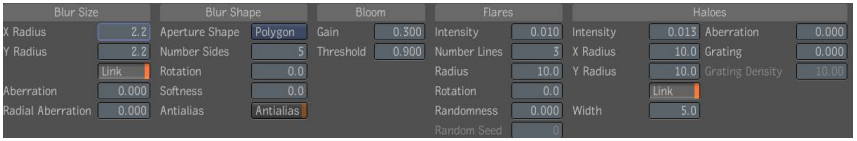
The Lens Blur tool allows you to simulate rack defocus and add photographic effects to 3D rendered scenes or live footage. You can also create lens flares, streak lines, and halos around bright lights and reflections.

Lens Blur UI

The Lens Blur UI is composed of four tabs: Lens Blur, Modulation, Output, and Masking.

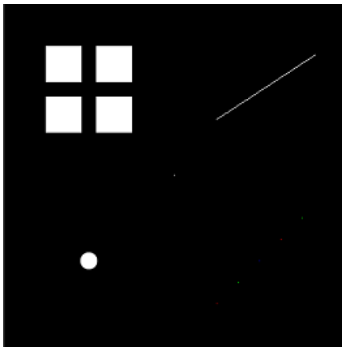
The Lens Blur tab consists of a set of parameters for setting the main blur effect, as well as controls to create bloom, flares, and halos.

To create the main blur effect, use the controls on the left side of the UI.

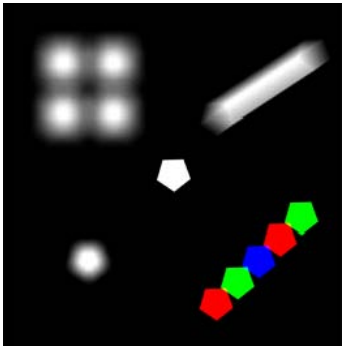


Use:	To:
X Radius	Set the amount of blur to apply in the direction of the X axis.
Y Radius	Set the amount of blur to apply in the direction of the Y axis.
Link	When enabled, changing the X Radius or the Y Radius causes the other one to change in the same proportion.
Aberration	Set the aberration. Setting the aberration parameter to greater than 0 will make the radius different for each of the three color planes, introducing color fringes.
Radial Aberration	Set the radial aberration. Setting the radial aberration greater than 0 will scale the image about the center differently for each color plane, producing color fringes that are offset radially, simulating transverse aberration.
Aperture Shape	Select either regular polygon shapes or ellipses.
Number Sides	Set the number of sides of the polygon blur shape.
Rotation	Set the rotation of the shape (in degrees).
Softness	Extend the blur outward from the edge of the polygon with an exponential falloff curve. Softness is specified as a distance (the same as the radius).
Antialias	Turn on and off the higher quality rendering of the polygon/ellipse blur.

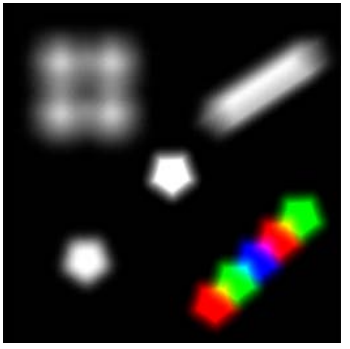
The following images show the types of effects you can generate using the main blur controls.



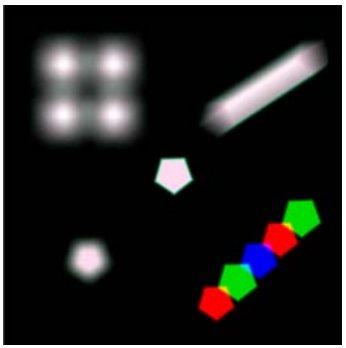
Input image (some single pixels not visible)



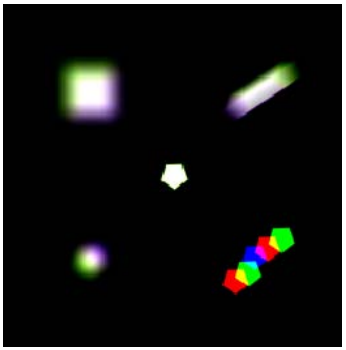
Pentagon blur with softness set at 2 pixels



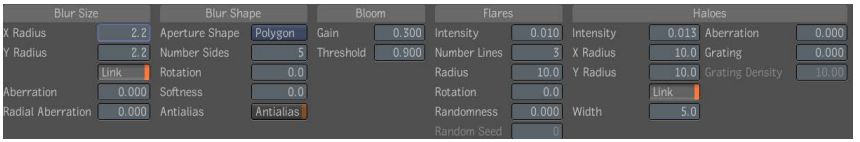
Pentagon blur with softness set to 20 pixels



Pentagon blur with aberration

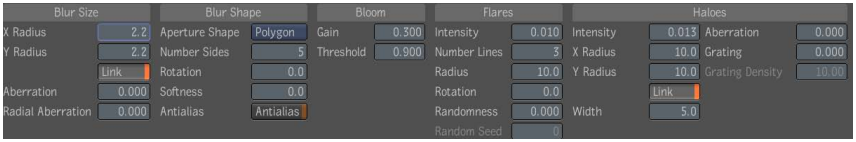


Radial chromatic aberration

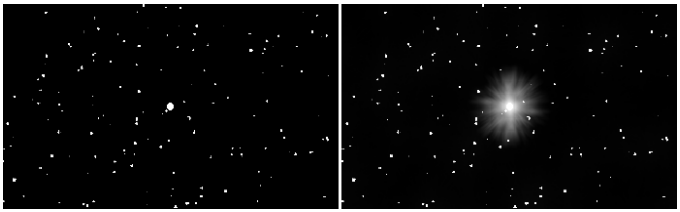


The Bloom controls let you remap the higher luminance range to increase the bloom appearance of the blur. This is especially useful for 8-bit images which cannot represent values greater than 1. With bloom, you can remap the values to make it seem like an HDR image, giving you more realistic blooms around bright light sources and reflections.

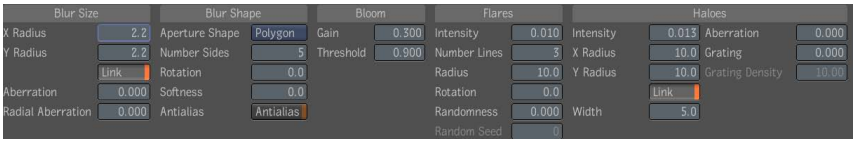
Original image



The Flares parameters let you add lines and streaks emanating from bright lights and reflections. You can set the intensity of the flares, as well as the number of flares. An intensity of 0 (the default) will disable flare generation. The rotation of the lines can be animated to simulate flares that seem to rotate as the camera moves relative to the light sources. The Randomness parameter controls how evenly distributed the lines are. A value of 0 makes the flare lines uniformly distributed around a circle, while a value of 1 makes them completely randomly distributed. You can also set a seed value to choose a particular sequence of lines. The effect of adding flares will be somewhat like the linear patterns in the right image below.

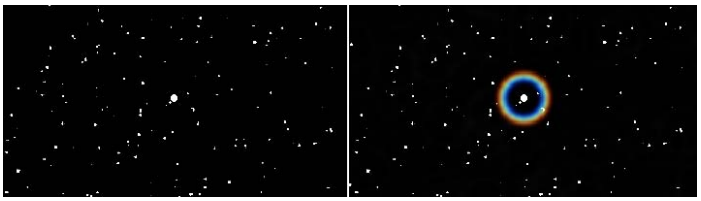


Original image



The halo is a ring of brightness of a specified radius from the center of bright lights, and with a specified width. If the radii differ among the color channels, the effect is a rainbow halo like in the following right image. This is controlled by the Aberration parameter. There is also an intensity parameter to adjust the strength of the halo; the default value of 0 disables the halo creation.

The Grating is a set of lines of random intensity, uniformly distributed around the circle. Increasing the Grating Density parameter increases the number of lines, and the smoother it appears.

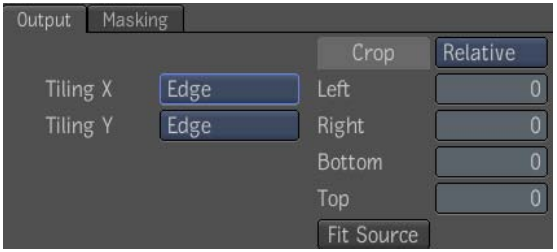


Original image

Modulation Tab

The Modulation tab looks just like that of the Blur, including the extrapolation (extending) capability—see Modulation Blur on page 391. You can choose what channel of the modulation image is used and apply an offset to the modulation image. The modulation can affect the X and Y radius, or the rotation of the polygonal or ellipse shape.

Output Tab



The Output tab is the same as that of the Blur tool, but without the Min Radius parameter—see Output Controls on page 399.

Masking Tab

The Lens Blur tool has a masking input and a Masking tab—see Pixel Masking on page 560.

Median Tool

The Median tool is an edge-preserving smoothing filter that works particularly well for removing impulse noise.

For grayscale images, the median filter works by ranking the pixels under the kernel according to their value and selecting the median to replace the pixel at the center of the kernel. This approach effectively removes spikes in the original image without the blurring typically introduced by common smoothing kernels (e.g. Gaussian).

For color images, there is no single concept of ranking, so several different criteria are supported. Component-wise ranking computes the median of each color component independently, however, this can result in pixel colors that did not belong in the original image. Luminance ranking computes the median of the luminance of each pixel under the kernel. This approach is fast and does not introduce new colors in the original image, however, luminance is not the best criteria for similarity in a color image. RGB Vector does not really rank the pixels under the kernel, but rather it chooses as the median the one pixel with the smallest sum of square distances (in RGB space) to all the other ones under the kernel. It selects the pixel that is closest to the center of the point cloud obtained by looking at the pixels under the kernel as points in 3D space. This approach is computationally intensive, but can yield better results than either of the other ranking criteria.

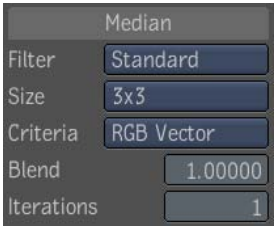
The median filter uses a square neighborhood and can round the corners of axes-aligned rectangular objects in an image.

As any noise reduction filter, the median filter may also affect the sharpness of small details in the input image. When this problem arises, the result of the median filter can be blended with the original image to decrease its effect.

Finally, the median filter can be applied recursively. Repeated applications using a small kernel size yield better results than a single application using a large kernel.

NOTE When using the filter recursively, the blending described above is applied at each iteration.

The Median tool has the following parameters:



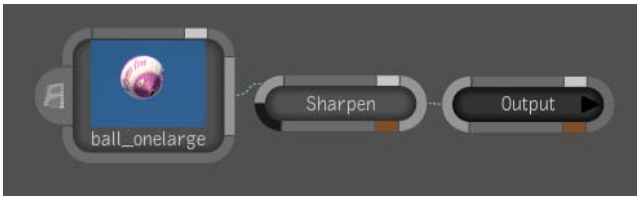
Use:	To:
Filter	Control the choice of filter used: ■ Median (default) ■ Hybrid Median
Size	Control the size of the filter: ■ 3x3 (default) ■ 5x5 ■ 7x7 ■ 9x9 ■ 11x11
Criteria	Control how the median value is chosen: ■ Component-wise ■ Luminance ■ RGB Vector (default)
Blend	Control how the result of one median iteration is combined with the original input. This parameter can vary between 0 and 100% (default).
Iterations	Control the number of times that the filter is applied recursively. This parameter can vary between 1(default) and 9.

Sharpen Tool

The Sharpen tool lets you increase the clarity and focus of an image. The Sharpen tool applies a sharpening filter to a number of adjacent pixels in the input image and increases their contrast.

To apply sharpening modifications to an image using the Sharpen tool:

1 Drag the Sharpen tool from the Tool tab to the dependency graph in the Schematic view.



2 Adjust the sharpen amount by dragging the value slider, or by typing in a value.

NOTE The Blur tool has a masking input—see Pixel Masking on page 560. The Sharpen tool's Amount value is animatable by setting keyframes or using expressions—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Unsharp Mask

The Unsharp Mask tool lets you sharpen the edges and other fine details in the input image.

Unsharp masking provides more control on the sharpening process and will generally produce better results than the Laplacian filter, but at a higher computational cost.

Unsharp masking works by removing the low frequency spatial information from the image and emphasizing the high frequency details. The algorithm basically uses a Gaussian filter to produce a smooth version of the original

image, called *unsharp mask*. The unsharp mask is then subtracted from the original image, removing low frequencies.

The Unsharp Mask tool has the following parameters:

- **Unsharp Mask Radius X, Radius Y, and Link** Control the radius of the blur (just like in the Blur tool). By default, the X and Y radii are linked and set to 1.20; otherwise, their range is the same as that of the Blur tool.
- **Strength** Controls the strength of the effect of masking that is added. Basically, the output image is computed as I+s*(I-M) where I is the original image, M is the unsharp mask, and s is the strength parameter (in 0 to 500 percent; default 100%).
- **Threshold** Defines how large the difference between the original image and unsharp mask must be before the original pixels are changed. A pixel is changed if the difference in any of its components is larger than or equal to the threshold (in 0 to 100%; default is 0%).

NOTE This tool can either affect the RGB components or the Alpha component, but not both at the same time.

Formatting Tools

In some cases you will be working with images of different sizes and color depths. Use Formatting tools to change these parameters.

There are five image formatting tools:

Tool:	Description:
Convert Depth	Changes an image's color depth—see Convert Depth Tool on page 410.
Crop	Removes rows or columns of pixels next to the edges of the input image—see Crop Tool on page 410.

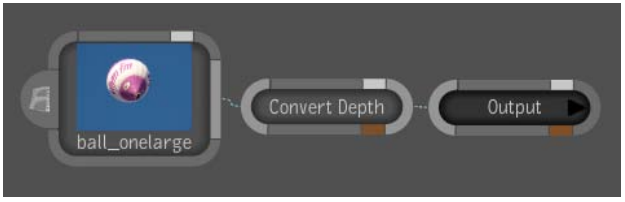
Tool:	Description:
Floating Windows	
Premultiply	Premultiplies an input image—see Premultiply Tool on page 413.
Unpremul-tiply	Unpremultiplies an input image—see Unpremultiply Tool on page 414.
Resize	Changes the resolution of an image—see Resize Tool on page 415.

Convert Depth Tool

The Convert Depth tool lets you change the color depth of an input image.

To change the color depth of an image:

1 Drag the Convert Depth tool from the Tools tab to the dependency graph in the Schematic view.



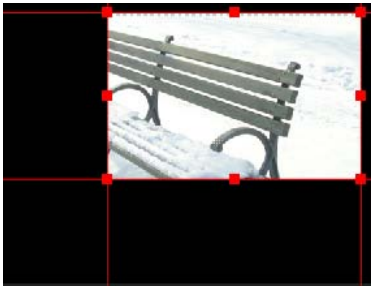
2 Select the target color depth from the color depth list.

Crop Tool

The Crop tool lets you change the size of an image. When you perform a crop, adjacent rows or columns of pixels next to the edges of the input image are removed. You can crop an image relative to its input size or perform an

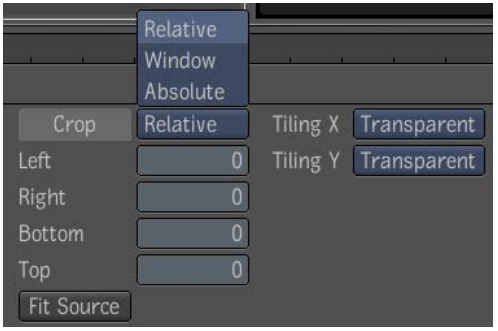
absolute crop if you want to retain certain portions of the image. To assist in cropping, you can keep a fixed image size regardless of input dimensions.

Original image



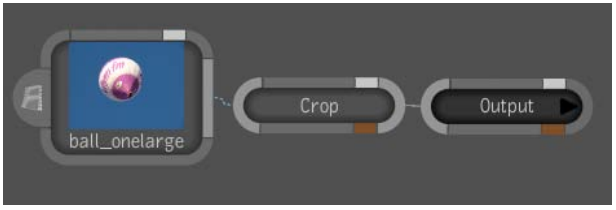
Cropped image

You can also animate the Crop tool's parameters by setting keyframes or using expressions—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.



To crop an image:

1 Drag the Crop tool from the Tools tab to the dependency graph in the Schematic view.



2 Select a crop type: Relative, Window, or Absolute.

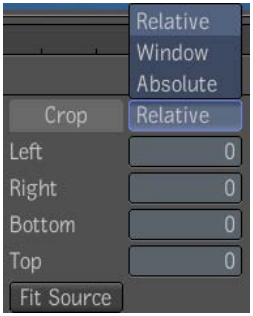
Use:	To:
Relative Crop	Crop an image relative to the size of the input image. The default value for all parameters (left, bottom, right, top) is zero. All parameters have a range of zero to the respective size (width and height) of the image being cropped. Cropping more pixels than an image has results in an invalid image size of zero.
Window Crop	Crop an image using absolute values. You can set values for Center X, Center Y, Width, and Height.
Absolute Crop	Crop an image using absolute values. You can set values for Left, Right, Bottom, and Top.
Tiling	Specify how the input image should be extended outside its region of definition (ROD).The supported tiling modes are Transparent (default), Edge, Repeat, and Mirror.

3 Crop the image by doing any of the following:

- Enter values in the Crop fields.
- Manipulate the crop box in the Player by dragging it.
- Adjust the red crop lines or crop handles in the Player.
- Resize the image symmetrically by using the **Ctrl** key and dragging until you get the desired image size.

4 Reset the values, if needed, to that of the input image by clicking the Fit Source button.

Tools that can change the resolution of an image (Garbage Mask, Lens Distort and 2D Transform) have extra controls to manage the crop.



The Auto Crop feature computes a fit-all region where the entire distorted image fits into the output without losing any part of the image. When you set the crop mode to Auto Crop, the output region at each frame will automatically be computed. The region is automatically resized and all crop fields except the mode are disabled.

Floating Windows

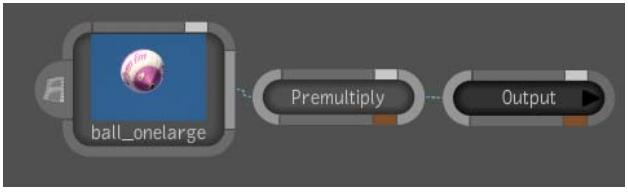
The Floating Windows tool allows you to...

Premultiply Tool

The Premultiply tool allows you to take an input image and create an image where each pixel's color component is multiplied by the pixel's alpha component. The multiplication result is clamped to the color range of the output image.

To premultiply an image using the Premultiply tool:

➤ Drag the Premultiply tool from the Formatting folder into the dependency graph in the Schematic view.



The image is premultiplied.

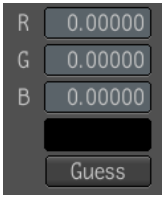
NOTE When using the Premultiply tool, make sure the output image is never processed by a tool that produces incorrect results while operating on an image with premultiplied data. Tracking of whether images are stored in premultiplied format or not, does not occur.

Unpremultiply Tool

The Unpremultiply tool lets you recover an alpha-unpremultiplied foreground image from an alpha-premultiplied image whose foreground may have been composited on a solid color other than black.

The Unpremultiplied tool takes an alpha-premultiplied image as its input and outputs a corresponding alpha-unpremultiplied image. It assumes the input image consists of a foreground composited over a solid color background; if the background is a color other than black, you can explicitly specify this background color so that the tool can correctly separate the foreground layer from this background and output the foreground layer alone. Optionally, your can ask the tool to guess the background color.

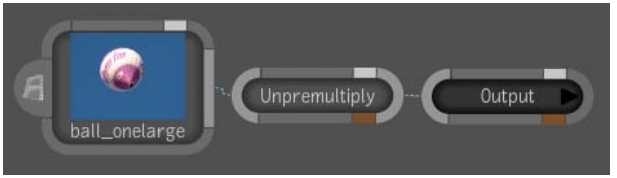
The Unpremultiplied tool has a single parameter:



Use:	To:
Background	Specify the background color so that the tool can correctly separate the foreground layer from the background and output the foreground layer alone.
Guess	Have the tool analyze the input image and guess the background color and set the color widget to that color.

To unpremultiply an image:

➤ Drag the Unpremultiply tool from the Tools tab to the dependency graph in the Schematic view.



The image is unpremultiplied.

NOTE When using the Unpremultiply tool, make sure the output image is never processed by a tool that produces incorrect results while operating on an image with unpremultiplied data. Tracking of whether images are stored in premultiplied format or not, does not occur. All tools except this one assumes unpremultiplied data.

Resize Tool

The Resize tool lets you change the resolution of an image and its frame bit depth when working with images of different resolutions.

Resize Tool UI

The Resize tool UI is divided into three areas:

- Destination settings area.
- Reference area (to modify crop box properties).
- Resize area.

Destination Settings Area

Use this area to select format, resolution, image and pixel aspect ratios, as well as bit depth of the destination image.

Select:	To:
Format	Select from a list of many standard resolutions, as well as a Custom option that you can use to specify non-standard resolutions.
Width and Height	Display the frame width and height of the selected resolution preset. If you select Custom from the resolution presets box, use these fields to enter the frame width and height values you want to use.
Image Aspect Ratio	Change the image aspect ratio.
Pixel Aspect Ratio	Change the pixel aspect ratio.
Depth	Select the output depth.

Reference Area

Use the Reference area to set crop box properties.

Select:	To:
Color	Change the color of the crop box. By default, the crop box is red. If there is a lot of red in the image you are resizing, you may want to change the color of the crop box to see it better. Clicking the color pot will take you to the color picker where you can set desired color for the crop box.
X and Y	Change the X and Y position of the crop box.

Select:	To:
Width and Height	Change the width and height of the crop box. You can either type the dimensions into the respective Width and Height attribute value fields, or you can grab the edges of the crop box with the cursor and resize.
Fit To Source	Fit the crop box to the source image.

Resize Area

Use the Resize area settings to define the destination format into which the source sample is fit.

The Resize area has the following parameters:

Pictures often appear fuzzy when scaled to a different size. The fuzziness of an image is controlled by two processes: resampling and filtering.

The filtering processes attempt to smooth the transformed pixels. When an image is being resampled to a different resolution, the old pixel map needs to be interpolated to a new one. This requires some form of image interpolation and a number of interpolation methods can be used.

Use:	To:
Filtering	Select a filter type. You can choose from; Box, Triangle, Bell, B-spline, Lanczos3, or Mitchell. These filters range in quality from the Box filter being of the lowest (unsuitable for further processing, but with the least amount of processing time), to the Mitchell filter producing the best possible results.
Fit Methods	Select a fit method. You can choose: Fill, Crop, Letterbox, or Crop Edges. If you select Letterbox or Crop Edges, you can also select or deselect the Keep Aspect parameter.
Center	Select to center the destination resolution.

Animate Resize Settings

Animate the position of the crop box to pan and scan a 16:9 composition to a 4:3 format. Pan and scan is a technique widely used for resizing 16:9 compositions to a 4:3 resolution when the area of interest in the shots is to one side of the 16:9 frame. You can follow the area of interest in the shot by animating the crop box along the X-axis.

To pan and scan a 16:9 composition to a 4:3 composition:

1. Drag the Resize tool from the Tools tab to the dependency graph in the Schematic view, and connect it to the composition.
2. From the Destination settings area, select the 4:3 resolution to which you want to resize the source.
3. Select a filter from Filtering.
4. Select the Crop Edges fit method.
5. Select Center and Keep Aspect buttons.
6. Select the Fit To Source button (in the Reference area) to make the crop box the same size as the source frame.
7. Go to the first frame.
8. Make sure Autokey is selected so you can create position keyframes for the crop box automatically.
9. Scrub through the clip to identify areas where the area of interest includes the left and right edges of the clip outside the crop box.
10. Animate the position of the crop box accordingly by dragging the crop box. You may also type the values into the respective X, Y, Width, and Height attribute value fields—see Setting Keyframes Automatically on page 680.
11. Play the resized composition.

Resizing a Composition using the Letterbox Fit Method

In this procedure, resize a 16:9 composition to a 4:3 destination resolution using the Letterbox fit method.

To resize a 16:9 composition from source to a 4:3 destination resolution using the Letterbox fit method:

1. Drag the Resize tool from the Tools tab to the dependency graph in the Schematic view, and connect it to the composition.
2. From the Destination settings area, select the 4:3 resolution to which you want to resize the source. You can also set custom width and height in the W and H fields.
3. Select the Letterbox fit method. By definition, when using the Letterbox fit method, black bars appear above and below the destination composition. However, the fill color in Composite is already black and these bars get cropped from the destination resolution.
4. Select a filter from Filtering.
5. The composition is resized.

Image Generation Tools

There are seven image generation tools that let you create images with a wide range of properties:

Tool:	Description:
Bilinear Ramp	Creates a multi-color, four-quadrant image based on RGBA values— see Bilinear Ramp Tool on page 421.
Checker-board	Creates a multi-color checkerboard image based on RGBA values— see Checkerboard Tool on page 423.
Color Source	Creates a single color image based on RGBA values— see Color Source Tool on page 424.
Linear Ramp	Creates a multi-color, two-quadrant image based on RGBA values— see Linear Ramp Tool on page 425.
Noise	Simulates or matches film grain—see Noise Tool on page 427.
Radial Ramp	Creates a multi-color, radially-ramped image based on RGBA values—see Radial Ramp Tool on page 426.
Slate	Lets you add production information over an image or clip — see Slate Tool on page 429.

NOTE All image generation tools are output tools and therefore, have only an output node connector and no input connector. All image generating tools can be used to generate masks with those tools that have masking inputs—see Pixel Masking on page 560.

Setting Image Generation Properties and Formats

Each of the image generating tools has a common UI area where properties can be applied.

The following table lists the common property controls and their function.

Property	Description
Format	Provides a selection of film formats.
Image Width/Height	Sets the width and height of the image produced by the generator node.
Image As-pect Ratio	Sets the image aspect ratio of the image produced by the generator mode.
Pixel Aspect Ratio	Sets the pixel aspect ratio of the image produced by the generator mode.
Rate	Sets the rate of the stream produced by the generator node.
Channels	Sets the number of channels of the images produced by the generator node. Possible channel combinations are: ■ A ■ RGB ■ RGBA
Depth	Sets the depth per channel (in bits) of the images produced by the generator node. Possible image depths are: ■ 8-bit

Property	Description
	■ 16-bit ■ 32-bit
Repeat	Determines how the generator node outputs its result outside its source time range. Possible modes are: ■ No Repeat: There is no repetition at all, the generator outputs black frames outside its time range. ■ Hold: First and last frames are repeated. ■ Loop: Repeats the entire sequence. ■ Ping-Pong: Repeats the sequence from beginning to end, to beginning, and so on.
Mark in/Mark out	Defines the total number of frames the node generates for the image.
Start Time/End Time/Offset	Defines the time range (and any offset required) where the generator node is to output a result. It follows global time.
Lock Duration	Locks the Start/End time values.

NOTE Height and Width are animatable attributes—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

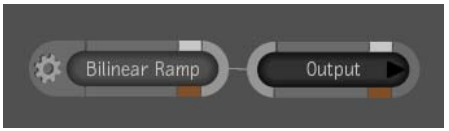
Bilinear Ramp Tool

The Bilinear Ramp tool lets you create images with a variety of properties and in several formats. The Bilinear Ramp tool is comprised of five sets of color channel value fields that let you to generate a multi-color image based on RGBA values. One set of values is for generating the fill color, and four sets are for generating start and end ramped colors residing in a four-quadrant region of definition (ROD). The fill color is defined as those pixels residing outside the ROD. Two other controls let you to position the X and Y axes of the ramps at any location in the image.

NOTE The following procedure assumes that you have a Schematic view and a Player view open.

To create a bilinear ramped image:

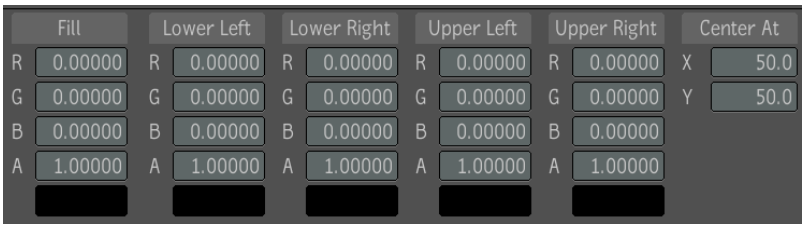
1 Drag the Bilinear Ramp tool from the Tools tab to the dependency graph in the Schematic view and attach it to the Output node.



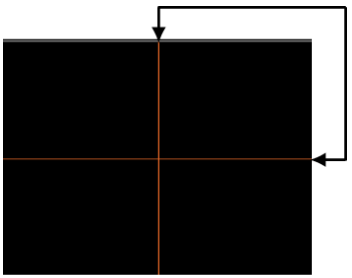
2 Apply image properties—see Setting Image Generation Properties and Formats on page 420.

3 Adjust the color values in the Fill, Lower Left, Lower Right, Upper Left, and Upper Right.

4 Modify the location and direction of the X and Y ramps.



NOTE You can also manipulate the linear ramp by clicking and dragging the horizontal and vertical bars.



NOTE You can also type values directly into the value fields and press **Enter**.

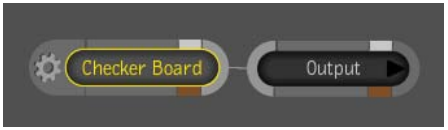
NOTE The Bilinear Ramp tool's Fill, Lower Right, Lower Left, Upper Right, Upper Left, and Center X and Center Y parameters are animatable by setting keyframes or using expressions—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Checkerboard Tool

The Checkerboard tool lets you create images with a variety of properties and in several formats. The Checkerboard tool is comprised of three sets of color channel value fields and X and Y Period fields. One set of values is for generating the fill color, and two sets are for generating X and Y colors residing in the region of definition (ROD). The fill color is defined as those pixels residing outside the ROD. Two other controls let you adjust the size and look of the checkerboard in the X and Y directions.

To create an image using the Checkerboard tool:

1. Drag the Checkerboard tool from the Tools tab to the dependency graph in the Schematic view and attach it to the Output node.



   The Checkerboard tool UI appears.

2. Apply image properties—see Setting Image Generation Properties and Formats on page 420.

3. Adjust the Fill, Color values, and Period values by dragging inside the value fields.

NOTE You may also type values directly into the value fields and press **Enter**.

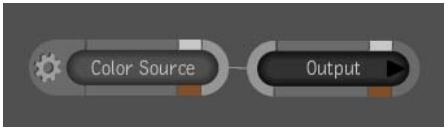
Color Source Tool

The Color Source tool lets you create images with a variety of properties and in several formats. The Color Source tool is comprised of two sets of color channel value fields that allow you to generate a single color image based on RGBA values. One set of values is for generating the fill color and the other set is for generating the color residing in the region of definition (ROD). The fill color is defined as those pixels residing outside the ROD.

NOTE The following procedure assumes that you have a Schematic and a Player view open.

To create an image using the Color Source tool:

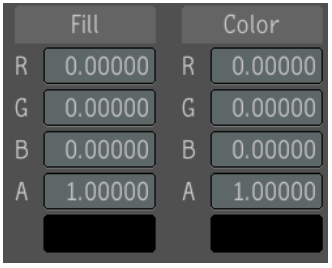
1. Drag the Color Source tool from the Tools tab to the dependency graph in the Schematic view and attach it to the Output node.



   The Color Source tool UI appears.

2. Apply image properties—see Setting Image Generation Properties and Formats on page 420.

3. Adjust the Fill and Color values by dragging inside the value fields.



   NOTE You may also type values directly into the value fields and press **Enter**.

NOTE The Color Source tool's Fill and Color parameters are animatable by setting keyframes or using expressions—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

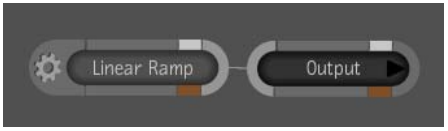
Linear Ramp Tool

The Linear Ramp tool lets you create images with a variety of properties and in several formats. The Linear Ramp tool is comprised of three sets of color channel value fields that allow you to generate a multi-color image based on RGBA values. One set of values is for generating the fill color, and two sets are for generating start and end ramped colors residing in the region of definition (ROD). The fill color is defined as those pixels residing outside the ROD. Two other controls allow you to position the center of the ramp at any vertical or horizontal point in the image.

NOTE The following procedure assumes that you have a Schematic and a Player view open.

To create a linear ramped image:

1. Drag the Linear Ramp tool from the Tools tab to the dependency graph in the Schematic and attach it to the Output node.

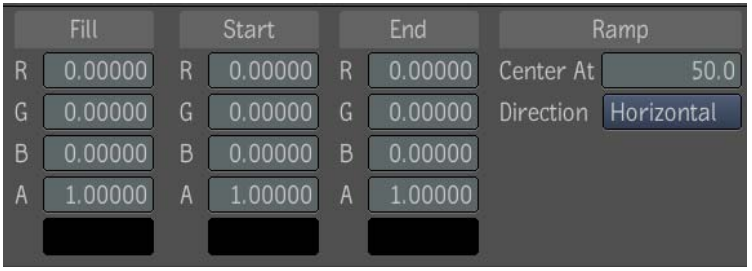


   The Linear Ramp UI appears.

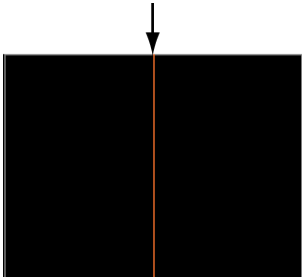
2. Apply image properties—see Setting Image Generation Properties and Formats on page 420.

3. Adjust the Fill, Start, and End color values by dragging inside the value fields.

4. Modify the location and direction of the linear ramp.



NOTE You can also manipulate the linear ramp by clicking and dragging the vertical bar.



NOTE You can also type values directly into the value fields and press **Enter**.

NOTE The Linear Ramp tool's Fill, Start, End, and Ramp Center parameters are animatable by setting keyframes or using expressions—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

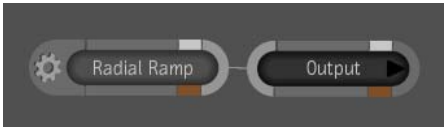
Radial Ramp Tool

The Radial Ramp tool lets you create images with a variety of properties and in several formats. The Radial Ramp tool has three sets of color channel value fields that allow you to generate a multi-color image based on RGBA values. One set of values is for generating the fill color, and two sets are for generating start and end radially ramped colors arranged around a central axis region of definition (ROD). The fill color is defined as those pixels residing outside the ROD. Other controls let you to position the X and Y axes of the central ramp at any location, stretch the ramp vertically or horizontally, and define the falloff center point and range.

NOTE The following procedure assumes that you have a Schematic and a Player view open.

To create a radially ramped image using the Radial Ramp tool:

1 Drag the Radial Ramp tool from the Tools tab to the dependency graph in the Schematic view and attach it to the Output node.

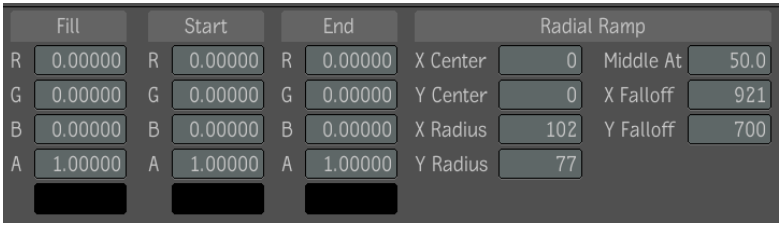


2 Apply image properties—see Setting Image Generation Properties and Formats on page 420.

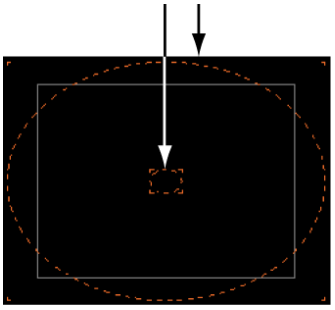
3 Adjust the color values in the Fill, Start, and End fields.

4 Modify the location and direction of the X and Y ramps.

5 Adjust the middle and falloff ranges.



NOTE You can also manipulate the radial ramps by left-clicking and moving them.



NOTE You can also type values into the value fields and press **Enter**.

NOTE The Radial Ramp tool's Fill, Start, End, and all associated Radial Ramp parameters are animatable by setting keyframes or using expressions—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Noise Tool

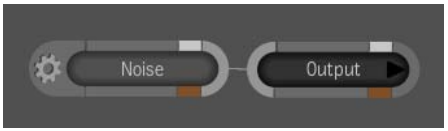
The Noise generating tool lets you add realism to computer-generated images, or to simulate or match film grain. The generator creates an image of random pixels. The Noise generating tool is comprised of three sets of color channel

value fields that allows you to generate a multi-color image based on RGBA values. One set of values is for generating the fill color, and two sets are for generating start and end color values. You can also adjust the level and seed (the method used to generate the noise).

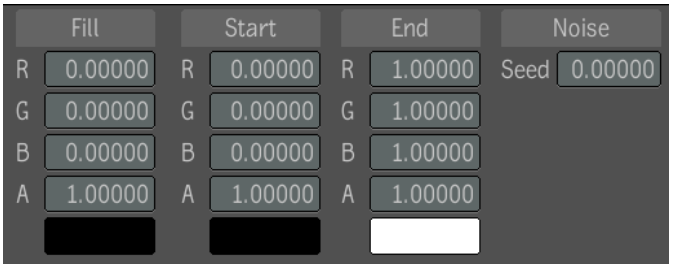
NOTE The following procedure assumes that you have a Schematic and a Player view open.

To create an image using the Noise tool:

1. Drag the Noise tool from the Tools tab to the dependency graph in the Schematic view and attach it to the Output node.



2. Apply image properties—see Setting Image Generation Properties and Formats on page 420.
3. Adjust the color values in the Fill, Start, and End fields.
4. Adjust the noise seed.



NOTE You can also type values into the value fields and press **Enter**.

NOTE The Noise tool's Fill, Start, End, and associated Seed parameters are animatable by setting keyframes or using expressions—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Slate Tool

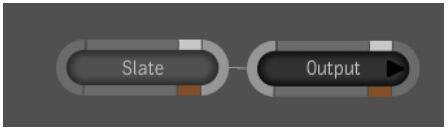
The Slate tool lets you add production information over an image or clip, which is useful during the review/approval stage of a project. Although Slate is not a full text tool, you can adjust the size, color, opacity, position, and duration of the text, as well as add a drop shadow. You can also animate the attributes of the text.

When adding information to a composition, you can use variables for the composition's name, date, time, frame number, timecode, and length. You can also enter your own information about the composition, such as the project title, camera, lens, film stock, and so on.

NOTE The following procedure assumes that you have a Schematic and a Player view open.

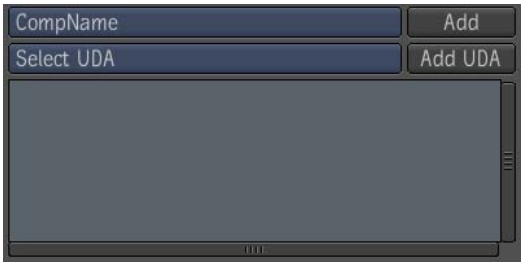
To add production information to a composition:

1 Drag the Slate tool from Image Generation folder in the Tools tab to the dependency graph in the Schematic and attach it to the Output node.



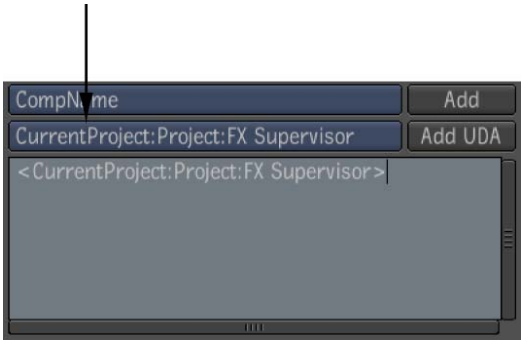
2 Apply image properties—see Setting Image Generation Properties and Formats on page 420.

3 To add a variable to your composition, select a variable from the list, and click Add. You can add multiple variables.



The composition's name appears in the Player.

4 To enter information about your composition, select the type of information you want to add and click Add UDA. You can add as much or as little information as needed.

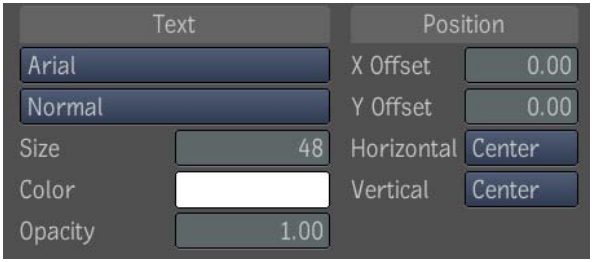


The field appears in the text box, but not in the Player.

5 Place the cursor inside the text field and edit the information.

6 Format the text and modify its location.

TIP To scroll the list of fonts, use your mouse wheel or the scroll bar beside the list.



7 Add a drop shadow to the text by clicking Drop Shadow and setting its position, color, and opacity.

8 Add a background behind the text by clicking Box and setting the color, opacity, and margin.

Transform Tools

Use the Transform tools to move, reorient, and animate images.

There are four image transformation tools:

Tool:	Description:
2D Transform	Apply chains of 2D transformations on images—see 2D Transform Tool on page 432.
Flip	Flips pixels left-to-right, top-to-bottom, or both—see Flip Tool on page 435.
Orient	Rotates an image—see Orient Tool on page 436.
Panner	Repositions an image in the X and Y directions—see Panner Tool on page 437.

2D Transform Tool

The 2D Transform tool lets you apply arbitrary transforms on an image and lets you add camera jitter, motion blur, or stabilizing to the image. You can also move, scale, rotate, and shear images in 2D with high quality filtering as well as concatenate transforms to avoid re-filtering.

The 2D Transform tool has the following parameters:

Transform generator

Use:	To:
Transform generator	Create, delete, order, mute, solo and reset transforms. Solo (S) takes precedence over Mute (V) (a transform that's both soloed and muted is visible) while soloing a transform does not affect the muteness of other transforms, thus soloing a transform then un-soloing it does not lose state. ■ The main reset button (in the Tools Options) is the equivalent of deleting and re-adding the tool (all values are reset, and additional transform generators are deleted). ■ The tool maintains the selection of the transforms. ■ Adding a new transform selects the new transform and deselects the rest.
Transform Type	Set the type of transform. You can choose from 1 point, 2 point, 4 point, or scale, rotation, and translation and set source and destination values, as well as adjust offset parameters for tracking or stabilizing purposes—see Tracking and Stabilizing on page 753.
Transform value editors	Set the values of your transforms, you can also use the direct manipulators in the Player to transform the image—see 2D Transforms on page 478.

To produce an affine transformation on an image:

1. Add a Transform 2D tool at the desired point in the dependency graph (if one is not already there). One transform is automatically added. You can add additional transforms using the New button.
2. Use the controls that appear in the UI to the right of the super tool controls or the direct manipulators that appear in the Player. The direct manipulation controls are the same as those found in the Warps—see 2D Transforms on page 478.
3. To shear an image, press the **Insert** key while the focus is in the player (this toggles the transform manipulator and the pivot manipulator), then using the rotate widget, set your pivot angle.
4. Press the **Insert** key again and use the scale widgets to shear the image.

NOTE Each transform influences transforms which follow it. The more it influences, the more 'global' it is, or conversely, the fewer it influences, the more 'local' it is. The pivot is independent within each transform.

Adding Motion Blur and Filters

Controls on the 2D Transform tool Output tab lets you add motion blur, filtering, and cropping to your image.

The Output tab has the following parameters:

Use:	To:
Motion Blur	Add motion blur to the result image. Motion blur can be on or off; when on, there are controls for shutter speed and offset; the time interval over which the shutter is open is centered at the current frame time by default; the offset can be used to move it elsewhere.
Filtering	Filter the output image—see Cropping and Filtering the Output on page 482.

Use:	To:
Crop	Crop the output image—see Crop Tool on page 410.

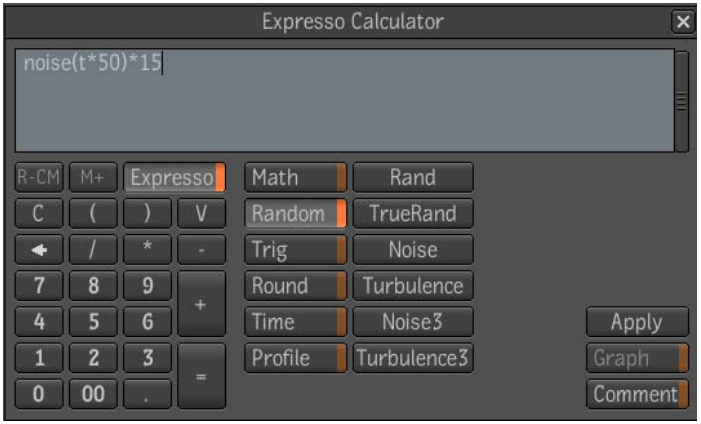
Simulating Camera Jitter Using the 2D Transform Tool

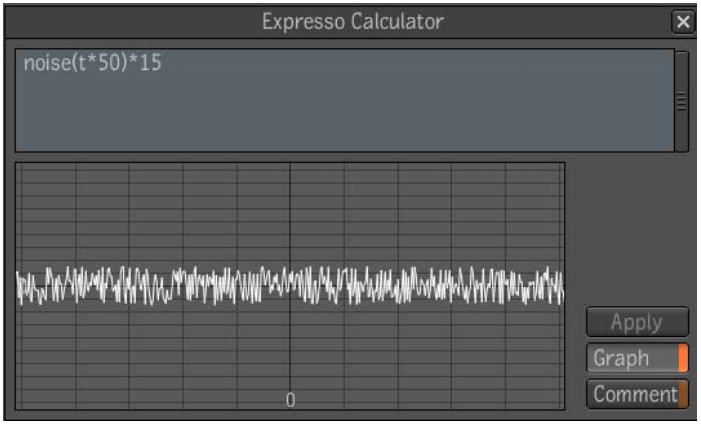
NOTE The following example describes one way to simulate camera jitter using the 2D Transform tool combined with a Noise or Rand expression—see Random Number Functions on page 749.

To simulate camera jitter using the 2D Transform tool:

1. Place a Transform 2D tool at the desired point in the dependency graph (if one is not already there).
2. Select the 2D Transform tool.
3. Right-click in the X (or Y) value editor and select Set Expression. The Expresso calculator appears.
4. Select Random, then Noise, and type the following expression: noise(t*50)*15. This expression can be broken down as follows:

Where:	Returns:
noise (t)	A value between -1 and 1.
noise (t * 50)	A value between -1 and 1 and where “50" represents frequency. The larger the value, the longer the frequency of the resulting noise.
noise (t * 50) * 100	Values between -100 and 100, where 100 represents amplitude.





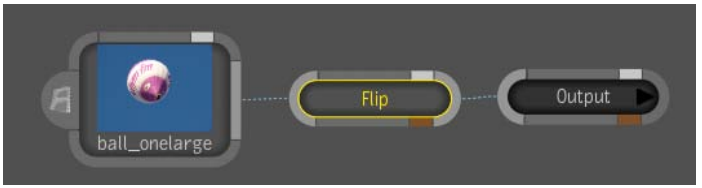
NOTE You may also want to add motion blur to the output for a more realistic effect.

Flip Tool

The Flip tool lets you take an image and flip the pixels left-to-right, top-to-bottom, or both when the Link button is enabled. The default setting is vertical flip.

To modify an image using the Flip tool:

1 Drag the Flip tool from the Tools tab to the dependency graph in the Schematic view. The image will be vertically flipped.



2 Select the desired flip parameters: Vertical, Horizontal, or both.

NOTE The Flip tool's Vertical and Horizontal attributes are both animatable by setting keyframes or using expressions—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Orient Tool

The Orient tool lets you rotate its input by 0, 90, 180, and 270 degrees (with the rotation parameter being an enumeration) and further flip the result vertically and/or horizontally.

The vertical and horizontal directions refer to the image after the rotation has been applied.

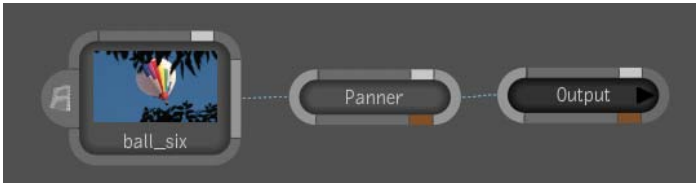
NOTE When possible, use this tool in lieu of more general transform tools because no resampling the input image required resulting in no loss of image quality.

Panner Tool

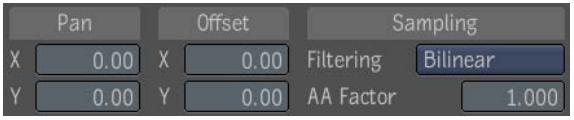
The Panner tool lets you reposition the image in both the X and Y directions, as well as to reposition the image using the X and Y offset created when used with Tracker tool data inside an expression—see Validating and Applying the Expression String on page 729 and About Tracking and Stabilizing on page 754.

To reposition an image using the Panner tool:

1. Drag the Panner tool from the Tools tab to the dependency graph in the Schematic view.



2. Select the desired reposition parameters: X, Y, or both, and select the desired offset of the X and Y axes.



3. Select the filtering option.
4. Select the anti-aliasing factor.

NOTE The Panner tool's X and Y axes attributes are both animatable by setting keyframes or using expressions—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Pixel Expressions

18

Topics in this chapter:

About Pixel Expressions

Use one of Composite's pixel expression tools to create customized effects and apply color management solutions. There are two pixel expression tools:

Tool:	Description:
PXL Tool	Use the powerful PXL language to create your own plug-in tools—see PXL Tool on page 440.
CTL Tool	Use the CTL tool to facilitate color transforms and other pixel-based procedures—see CTL Tool on page 463.

PXL Tool

The PXL tool provides the ability to write a C-like program to control the creation of its output image. The program executed by the PXL tool is written in a new language called PXL, the Pixel eXpression Language (pronounced “pixel”). The PXL language offers a rich set of features like conditionals, looping, and numerous built-in functions. The PXL tool offers users the ability to create new effects by writing a PXL language program. The PXL tool can be used to implement any point or gather operator, as it can read any pixel in its input images.

The PXL tool's primary use cases are two-fold. One use case is for quick creation of a short, non-reusable PXL program, for simple image generation or transformation. Another will be to write longer scripts to create customized effects that are not supplied by Composite, which will be shared by many users. Examples might include:

- Image generators
- Warping
- Transitions (fade & dissolve)
- Blur & convolution
- Tone mapping
- Noise generators
- Filters
- Masking
- Convert color spaces
- Compositions
- Color correction

PXL Tool Inputs and Outputs

The PXL tool has a single image output. It has a single, fixed primary input (which can be unconnected, when used as an image generator). Further image inputs can be added to the tool, which allows for an arbitrary number of secondary image inputs to the tool. These secondary inputs can also be

removed from the tool. Because it supports masking, the PXL tool also has a standard masking image input.

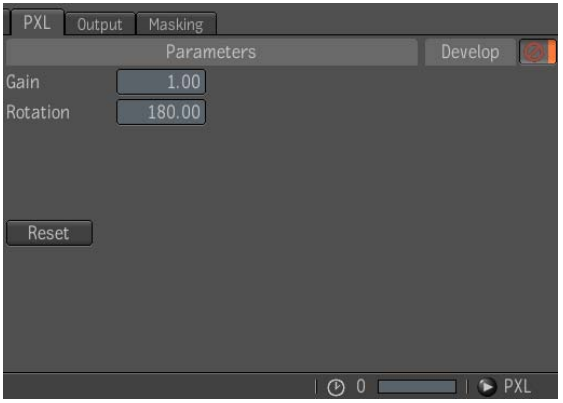
The PXL tool allows the user to create an arbitrary number of animated scalar float inputs, to be used as parameters to the PXL script. Because of UI space restrictions, only the first ten scalar parameters will be shown in the UI; however, all parameters can be seen in the composition browser. These parameters are passed as arguments to the main() function of the PXL script.

PXL Tool UI

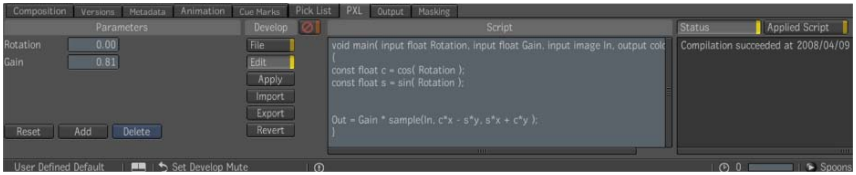
The PXL tool UI has 3 tabs: the main UI tab, the Output tab, and the Masking tab. The main tab supports two main modes, Develop mode and User mode.

User mode is meant to address the needs of users of the PXL script, who are interested in changing the parameters to the script, but not necessarily the script itself. Develop mode also has parameter controls as in User mode, but has a UI to edit the script using the Composite internal editor, or read the script directly from a file (to support a developer who wishes to edit the script using an external editor), and view script compilation errors.

The PXL User mode UI is shown below. It shows a PXL tool where two parameters have been added, here named Gain and Rotation. The Develop functionality is muted, so its UI is not shown. The Reset button under the parameters is used to reset only the parameters, not the whole tool, which is done with the Reset button in the General Tool UI.

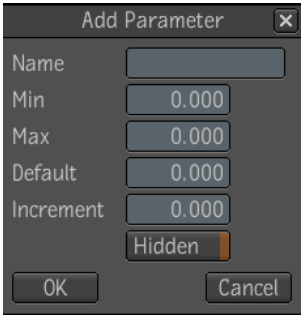


One of the UI layouts of the PXL Develop mode is shown below. In Develop mode, Develop is un-muted, so that developer functionality is exposed in the UI.

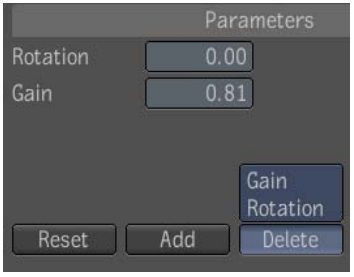


The area on the left displays the same parameters that are shown in User mode, as well as two buttons to add and delete parameters. Parameters cannot be edited, only added and removed.

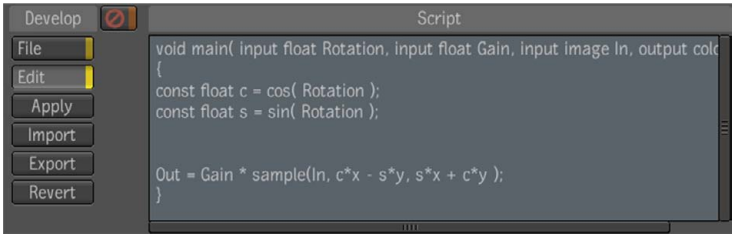
Clicking the Add button pops up a modal dialog where the user types in the name of the parameter, its minimum, maximum, and default values, as well as the minimum increment to be used when changing the value in the UI. Additionally, clicking the Hidden button means that a particular parameter will only appear in Develop mode. This modal dialog also includes a Cancel button.



Clicking the Delete button pops up a menu where you select the parameter to be removed.



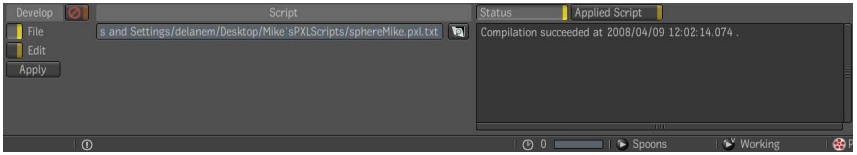
The Develop controls are to the right of the Parameter controls. The File and Edit radio buttons control whether the PXL script source is read from an external file, or from the Composite internal editor.

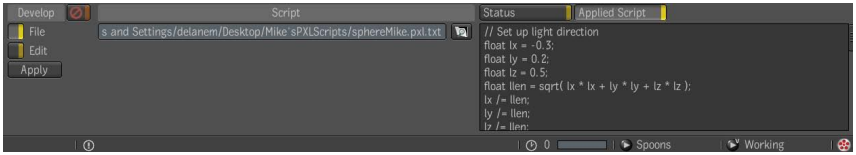


The above graphic shows Edit mode, where the PXL script developer is editing the script inside the Composite internal editor (titled Script), which is displayed to the right of the develop controls.

Select:	To:
Apply	compile the edited script and, if successful, applies it as the new image processing script.
Import	display a modal dialog to read an external file into the Composite internal editor. The contents of the external file replace the contents of the Composite internal editor.
Export	display a modal dialog to write the contents of the Composite internal editor to an external file.
Revert	replace the contents of the Composite internal editor with the currently-applied script, thereby reverting the edited script back to what it was when previously applied.

The script status and the applied script display share the right portion of the PXL UI. Push toggle buttons on the top of that column to determine which text viewer is shown. Both text views are read-only views. The status text viewer shows compilation errors from the latest applied script, if any. If there are no errors, the display shows a success message. The applied script viewer shows the currently-applied script. This can be useful as a reference, when the PXL script is actively undergoing editing, and you want to know what the currently-applied script is.



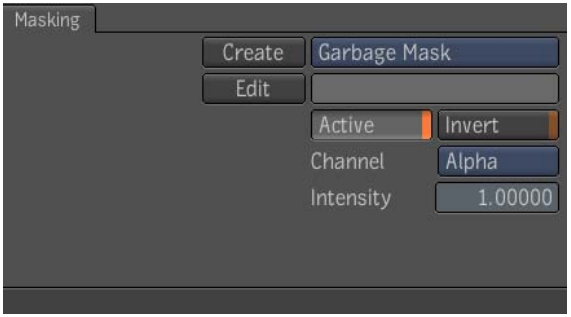


The other option while in Develop mode is File mode where the PXL script is being read from an external file. In this mode, the PXL file is edited with an editor external to Composite, then saved to disk. You will then click on the Apply button to have Composite read the script from the file and apply it. In File mode, the Import, Export, and Revert buttons of the Develop controls are not relevant, as they pertain to the Composite internal editor, and are therefore not shown.

Output Tab

Because the PXL tool can be an image generation tool, it requires an Output tab to define the output properties. To define output properties, it uses the same tab as found in Composite image generators. If the primary input is connected, all options in this tab will be greyed-out and disabled, as is the case with the Slate tool—see Slate Tool on page 429.

Masking Tab



The PXL tool has the same masking capability that is available on other image modifier tools, in the standard Masking tab—see Creating Masks on page 540.

Sharing PXL Tools

An important use case of the PXL tool is that even if you are not an experienced PXL script writer you can still use the PXL tool to quickly write a simple expression to create a basic effect. Use cases might be multiplying one of the pixel channels by a constant, or multiplying two input images together, or a simple conditional test to reveal out of gamut pixels as a false-color image. Typically, these PXL tools will not be reused.

Another important use case is where a PXL script will be written by an experienced developer, and distributed to others to be used in User mode. In such a case, an easy means of distribution is important. Distributing the PXL script alone is insufficient, as parameter names and min, max, default values will not be available. Tool presets allow users to name, save persistently, and distribute presets for any given Composite tool. This is exactly the functionality required for you to develop custom tools written with the PXL tool. You can write your own color corrector, keyer, or any other effect using a PXL script. You can then package it as a PXL tool preset.

You can load and save PXL tool presets using the Preset controls in the Tools Options area located at the far right of the tool UI—see Tool Presets on page 146.

PXL Presets

There are several PXL tool presets installed with Composite and they are described below:

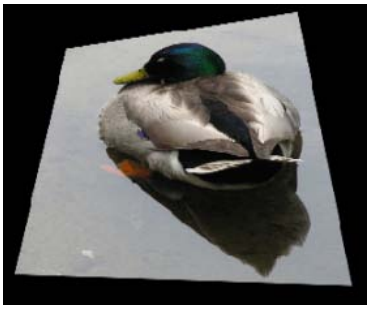
- **Blur 5 x 5** A simple 5 x 5 blur kernel with hard coded weights.

Original

Blurred

- **Clouds** Creates a cloud texture.

- **Corner Pin** Provides eight parameters to control the four corners of the image, and computes a perspective transformation to place the image based on the corners. By setting expressions to link the eight parameters to a garbage mask rectangle, a simple interactive corner pinner can be made.



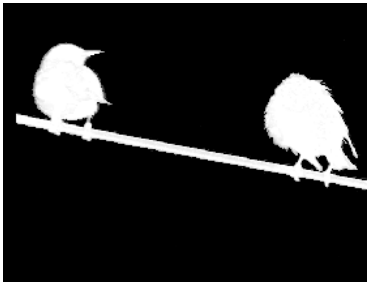
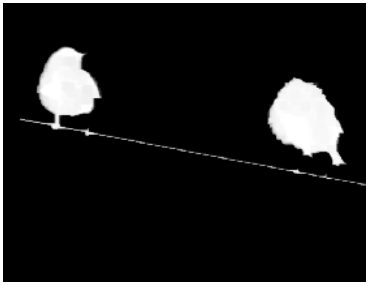
- **Directional Blur** Performs an anti-aliased blur using a rotated rectangle to achieve a blur in a particular direction.

Original Blurred

■ **Emboss** Simple emboss, based on gradients of luminance.


Original Embossed

■ **Erode Alpha** Erodes alpha using a simple square kernel.

Original Eroded Alpha with a 10 pixel radius

- **Gamma** Applies a simple gamma exponential to all channels.

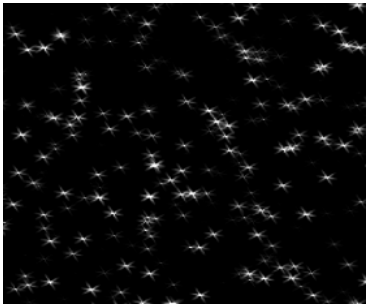

Original Gamma correction applied

- **Lattice** Adds a lattice of shaded tubular bars to the image channels.

- **Lens Flare** Provides 1 to 5 lines of flares centered on bright points on the image. You can control the weight of the lines relative to the weight of the source pixel at the center of the blur. There is also a falloff gamma

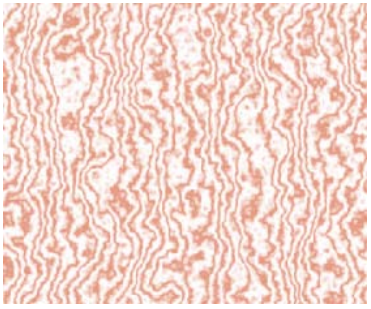
factor for the weight of the lines as they get more distant from the centre of the blur.

Original

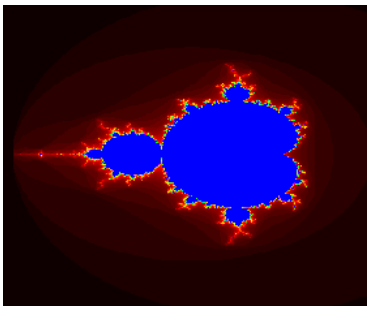


Lens flare applied

■ **Marble** Creates a marble texture.

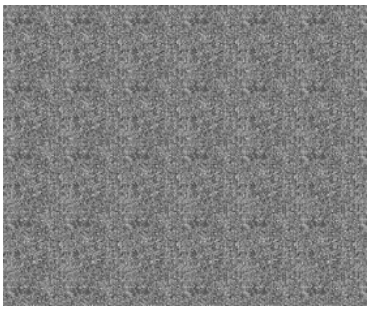


■ **Mandelbrot** Generates a mandelbrot fractal image.



- **Noise Wipe** Performs a wipe transition between two images where each pixel switches at a random time.

- **Noise** Creates a pattern of random noise.

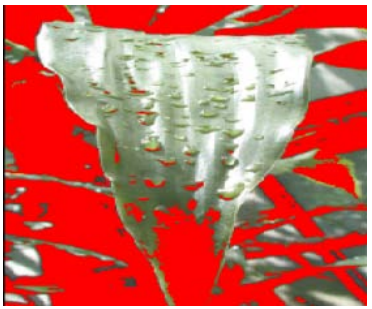


- **Num Colors** A preset that reduces the number of colors to create images with banding. It can also add dithering noise to reduce the banding.

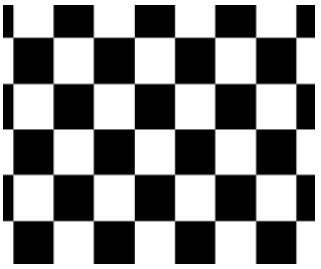
NumColors = 6, Dither = 0

NumColors = 2, Dither = 1

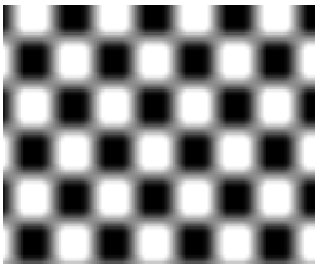
- **Out of Range** Marks pixels outside of a specified range with a specified color, such as red in the following image.



- **Polygon Blur** Performs a simple anti-aliased polygon blur, to simulate a lens effect. You can choose the number of sides, the radius, and rotation angle.

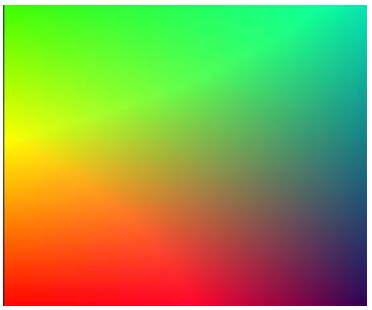


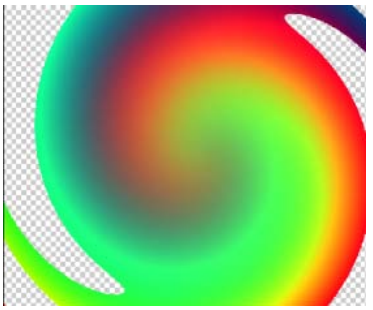
Original image



Polygon Blur applied

- **Ripple Twirl** A twirl effect with some extra ripples.





Original image Ripple Twirl applied

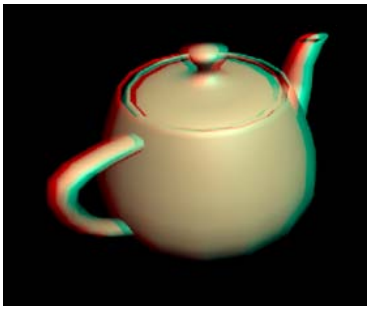
■ **Sharpen 5 x 5** A simple 5x5 sharpening kernel with hard coded weights.


Original image Sharpened

■ **Spherical Mapping** Applies spherical mapping to the current selection.


Original image Spherical mapping applied

- **Stereo Anaglypha** A preset that provides two methods of combining colors into a single Red/Blue glasses image (left image) and one method for cross-eyed viewing (right image).



Red/Blue glasses Cross-eyed viewing

- **Wood Frame** Creates a wood frame. You can specify the frame's thickness and lighting qualities.

- **Wood** Creates a wood texture. You can specify the scale, size, and fineness of the grain.

PXL, the Pixel eXpression Language

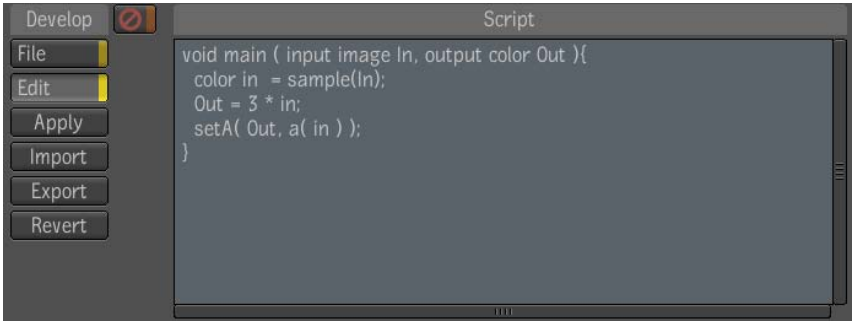
The following section outlines the traits inherent to the PXL language.

Example Script and Procedure

The following example PXL script and procedure multiplies the input pixel R, G, and B components by 3.

To multiply the input pixel R, G, and B components by 3:

1. Add a PXL tool to the dependency graph.
2. Click the Edit button and type the script into the script editor.



The pixels in the image are multiplied by a factor of 3.

Image Processing Algorithm Categories

There are three types of image processing operators: the point operator, the gather operator, and the scatter operator. They are based on a workflow in which there are one or more input images and a single output image:

- **Point operator** Image processing operation that only reads pixels from the input images at the current position, and only writes one pixel to the output image, at the same position.
- **Gather operator** Image processing operation that reads pixels from the input images from any position, and only writes one pixel to the output image, at the current position.
- **Scatter operator** Image processing operation that reads pixels from the input images from any position and writes one or more pixels to the output image at any position.

NOTE The PXL tool can perform point and gather operations, but not scatter operations.

Fundamental Types

PXL supports a specific set of fundamental types. It is case sensitive. Like C, all the variables must be declared before they can be used in an expression statement, unless the variable is a built-in variable, or a function argument. Unlike the C++ language, it is not possible to define new types with objects or structures. Here is the list of supported fundamental types:

- image
- float
- color/vec4

The color and vec4 types are aliases of one another, and are provided for convenience and readability. Otherwise, they are syntactically identical for PXL, and can be used interchangeably. Any reference in the documentation to type color can be understood as vec4, and vice versa.

NOTE PXL does not support vectors, arrays, or matrices.

The color / vec4 type is a quadruplet of floating point values for R, G, B, and A (or equivalently, X, Y, Z and W). These values are unclamped (even for A), which fully supports high dynamic range scene-referred color manipulation.

No explicit type casting or type conversion is supported. However, many mixed-type assignment operators and functions are provided.

If a variable of image type is used where type rules would require a color type, the language automatically calls the built-in single-argument sample (image) function. This is provided as a convenience.

As PXL has no Boolean type, the float type is used to represent Boolean values. Any value that is different from 0.0. PXL Boolean operators return 1.0 as a true value.

Qualifiers

A variable can be defined as *const*. In this case, the variable cannot be modified after his initialization. An error will be issued if a *const* variable is modified after initialization.

As previously described, function arguments can be qualified as input or output. These are currently only used by the PXL tool to pass in data to the main() function and read out the output pixel value. The initial value of an output argument is undefined.

Comments

Comments use the same syntax as C. Two consecutive slashes ("// Comment") are used for starting a single line comment. The slash-star ("/*") is the beginning token for multi-line comment, while the star-slash ("*/") is used to stop the multi-line comment.

Control Statement

PXL supports the "if" statement, the "while" statement and the "for" statement. They also use the same syntax as in the C language. The only exceptions are that PXL has no break or continue statements to affect looping.

if Statement

```
ifStatement:
   if ( expression ) statement
   if ( expression ) statement else statement
```

while Statement

```
whileStatement:
   while ( expression ) statement
```

for Statement

```
forStatement:
   for ( forInitStatement ; expression ; forUpdateStament ) statement

forInitStatement:
   variableDefinition
   assignment
   expressionStatement

forUpdateStatement:
   simpleAssignment
   simpleExpressionStatement
```

Operators

PXL supports the following operators, which are a subset of those found in C. Of note are the lack of bitwise manipulation operators, as well as the lack of a modulo operator, however the modulo is available through the built-in mod() function—see Built-in Functions on page 460.

Operator	Priority	Supported Type	Description
()	1	any expression	Grouping
x++	2	float, color	postfix increment
x--		float, color	postfix decrement
! x	3	float	negation
- x		float, color	unary minus
+x		float, color	unary plus
++x		float, color	prefix increment
--x		float, color	prefix decrement
x * y	4	float, color	multiplication
x / y		float, color	division
x + y	5	float, color	addition
x – y		float, color[4]	subtraction
x < y	6	float	less than
x > y		float	greater than
x <= y		float	less than or equal to
x >= y		float	greater than or equal to
x == y	7	float, color	equal
x != y		float, color	not equal
x && y	8	float	logical and
x || y	9	float	logical or
x , y	10	float, color	Comma operator. Returns right argument.
x *= y	11	float, color	multiplication assignment
x /= y		float, color	division assignment
x += y		float, color	addition assignment
x –= y		float, color	subtraction assignment
x = 7		float, color	assignment

NOTE Operations on color / vec4 are done on a per-channel basis.

Variable Declarations

The naming of variables uses the same rules as C. A variable name must begin with an alphabetical character, followed by none or more alphanumeric characters or underscore character. It must not contain any white spaces.

All variables are initialized by Composite upon declaration, depending on type:

- Float variables are initialized to 0.
- Color / vec4 variables are initialized to (0, 0, 0, 1).
- Image variables are initialized to an uninitialized image. An uninitialized image will return (0, 0, 0, 1) on sample(), has a pixel aspect ratio of 0, and has a size of (0, 0).

Numeric Constants

Name	Type	Value
M_E	float	e (approximately 2.7182818)
M_PI	float	π (approximately 3.1415927)
FLT_MAX	float	The largest positive float.
FLT_MIN	float	The smallest positive float
FLT_EPSILON	float	The smallest positive float such that 1 + ε - 1 is representable.

Built-in Variables

PXL has built-in variables to ease script writing. These variables are “x”, “y”, and “t”. The first two are the floating-point normalized image reference frame (x, y) coordinates of the pixel being computed—see Calculating Image Size in Composite on page 191. “t” is the floating-point current time, in seconds. All built-in variables are declared constant by the system, so they cannot be assigned to. PXL exposes the IRF coordinates of each pixel in its x and y built-in variables. This means that in PXL horizontally adjacent pixels will have a value of the x built-in variable that differs by 0.9 for an NTSC image. Thus, an NTSC image (with 720x486 pixels) is actually 648x486 IRF units (720 * 0.9 = 648), which represents a 4:3 image aspect ratio (648/486 = 4/3), as expected.

Variable Name	Type	Description
x	float	Contains the x normalized image reference frame (NIRF) position of the current manipulated pixel.
y	float	Contains the y NIRF position of the current manipulated pixel.
t	float	Return the current time in seconds.

Built-in Functions

PXL provides a number of built-in functions. Functions can return any of the PXL fundamental types, or can return void. Built-in function argument

overloading is supported by the sample() function to provide two implementations, one with a single argument, the other with 3 arguments.

NOTE In the following table, all references to type color also refer to vec4, and vice versa.

Name	Arguments	Return type	Description
color/vec4 functions			
color/vec4	float r, float g, float b, float a	color	RGBA color "constructor".
r	color	float	Extract red component of argument color.
g	color	float	Extract green component of argument color.
b	color	float	Extract blue component of argument color.
a	color	float	Extract alpha component of argument color.
x	vec4	float	Extract x component of argument vec4.
y	vec4	float	Extract y component of argument vec4.
z	vec4	float	Extract z component of argument vec4.
w	vec4	float	Extract w component of argument vec4.
setR	color, float	void	Set red component of argument color.
setG	color, float	void	Set green component of argument color.
setB	color, float	void	Set blue component of argument color.
setA	color, float	void	Set alpha component of argument color.
setX	vec4, float	void	Set x component of argument vec4.
setY	vec4, float	void	Set y component of argument vec4.
setZ	vec4, float	void	Set z component of argument vec4.
setW	vec4, float	void	Set w component of argument vec4.
dot	color, color	float	Inner product.
Elementary Functions			
acos	float x	float	Return the arc cosine of x. The computation is in radian.
asin	float x	float	Return the arc sine of x. The computation is in radian.
atan	float x	float	Return the arc tangent of x. The computation is in radian.
atan2	float x, float y	float	Return the arc tangent of x / y, but takes the sign of x and y into account to determine the resulting quadrant. The computation is in radian.
cos	float x	float	Return the cosine of x. The computation is in radian.
sin	float x	float	Return the sine of x. The computation is in radian.
tan	float x	float	Return the tangent of x. The computation is in radian.
degrees	float angle	float	convert angle unit from radians into degrees
radians	float angle	float	convert angle unit from degrees into radians
cosh	float x	float	Return the hyperbolic cosine of x.
sinh	float x	float	Return the hyperbolic sine of x.
tanh	float x	float	Return the hyperbolic tangent of x.
exp	float x	float	Return the exponential of x.
log	float x	float	Return the logarithm of x.
log10	float x	float	Return the base-10 logarithm of x.
pow	float x, float y	float	Return the x to the power of y.
pow10	float x	float	Return the 10 to the power of y
sqrt	float x	float	Return the square root of x.
abs	float x	float	Return the absolute value of x.
floor	float x	float	Return the integral part of x.
ceil	float x	float	Return a number up to the next integer.

Name	Arguments	Return type	Description
round	float	float	Round argument.
mod	float x, float y	float	Return the floating-point modulo of x.
hypot	float x, float y	float	Return the hypotenuse of a right triangle where x and y are the edges adjacent to the right angle.
min	float x, float y	float	Return the minimum value between x and y.
max	float x, float y	float	Return the maximum value between x and y.
Image functions			
sample	image	color	Return the color of the pixel of the argument image at the current position.
sample	image, float x, float y	color	Return the color of the pixel of the argument image at the specified position.
width	image	float	Return the image width in NIRF coordinates.
height	image	float	Return the image height in NIRF coordinates.
xmin	image	float	Return image lower left x NIRF coordinate.
ymin	image	float	Return image lower left y NIRF coordinate.
xmax	image	float	Return image upper right x NIRF coordinate.
ymax	image	float	Return image upper right y NIRF coordinate.
getOutputImage	void	image	Return output image.
getPixelAspectRatio	image	float	Return pixel aspect ratio of argument image.
isSame	image, image	float	Return true (1.0) if images are same.
Other			
rand	void	float	Pseudo-random number based on rand().
noise3D	float, float, float	float	Pseudo-random Perlin noise function.

Interface with Composite Executable

The Composite executable interfaces with the PXL script in the following way:

- The function declaration is optional. If present, it must be called “main()”.
- Function arguments can be any of the PXL basic types. Arguments to “main()” of type “image” with the “input” qualifier are the PXL tool image inputs, and must match the name of the input image socket. This implies that accessing the primary image input must be done with an input image argument named “In”. Arguments to “main()” of non-image type with the “input” qualifier are PXL tool parameter inputs, and are matched by name and type with the input parameters of the PXL tool. For example, a float parameter named “Gain” would be passed in the “main()” function as “input float Gain”. Arguments to “main()” of type “color” with the “output” qualifier are the pixel outputs. The first version of the PXL tool supports a single pixel output, which must be called “Out”.
- The return type is limited to void and the return statement is not supported.

If no function header is present for main(), Composite will create one automatically. It will include all defined image inputs, and all defined parameter inputs.

No warning is given if main() function arguments are not referenced by the function. However, a reference to a non-existent parameter or input image is an error.

All images read by the PXL script are read at the current time t. There is no way to read images at a time different from t within a PXL script. To do so, a user must use external Composite Retimer or Time Offset tools before inputting images to the PXL tool.

CTL Tool

Use the CTL tool to write a color transformation language (CTL) expression to control the resulting color of a pixel.

NOTE The CTL tool can only perform point operations—see Image Processing Algorithm Categories on page 457.

The CTL tool uses the same UI as the PXL tool—see PXL Tool UI on page 441.

The CTL tool supports every feature of CTL except print statements. For a more complete list, refer to the CTL manual: *http://ampasctl.sourceforge.net/CtlManual.pdf*. The main function of the script will be called for each pixel of the output image.

Before applying a script, several conventions must be respected:

- The entry point of the script is the “main” function. This function must be present in every script.
- The main function return value is ignored.
- The main function must have four parameters for each input image. They correspond to the RGBA channels of the pixel of the input image. They must be of type float, and be qualified as “input varying”. The name of these parameters must be the component letter in lower case (i.e. “r”, “g”, “b”, and “a”), concatenated with the name of the input image. Because the primary input of the CTL tool is called “In”, the corresponding arguments to the main() function for that input image will be:
  - input varying float rIn
  - input varying float gIn
  - input varying float bIn
  - input varying float aIn

- Additional images (if any) follow the same convention: for an additional input image Bg, the corresponding arguments to main would be:
  - input varying float rBg
  - input varying float gBg
  - input varying float bBg
  - input varying float aBg

- 4 parameters of the main() function must be present to define the output pixel color. They correspond to the RGBA channels of the output pixel. They must be of type float, and must be qualified as "output varying". The name of these parameters must be the component letter in lower case concatenated with the literal string "Out":
  - output varying float rOut
  - output varying float gOut
  - output varying float bOut
  - output varying float aOut

- 1 parameter of the main function must be present for each input parameter. They must be the same name and type as the parameter of the UI. Ex:
  - input uniform float P1

The following is a sample script and procedure that blends two images.

To blend two images:

1. Connect a CTL tool node into an output node in the Schematic view.
2. Connect image 1 to the In input of the CTL tool.
3. Create a second input for the CTL tool by right-clicking on the node and selecting Add input from the menu.
4. Name the new input Img1 when prompted.
5. Connect a second image to Img1 input.
6. Select the CTL tool node and click the Edit button to open the Composite internal editor.

7 Type the following script into the editor or copy then paste it in using **Ctrl** + **C** (for Windows and Linux) or **Cmd** + **C** (for Mac OS) and **Ctrl** + **V** (for Windows and Linux) or **Cmd** + **V** (for Mac OS):

```
void main(
input varying float rIn,
input varying float gIn,
input varying float bIn,
input varying float aIn,
input varying float rImg1,
input varying float gImg1,
input varying float bImg1,
input varying float aImg1,
output varying float rOut,
output varying float gOut,
output varying float bOut,
output varying float aOut
)
{
rOut = ( rIn * aIn + rImg1 * aImg1 ) / ( aIn + aImg1 );
gOut = ( gIn * aIn + gImg1 * aImg1 ) / ( aIn + aImg1 );
bOut = ( bIn * aIn + bImg1 * aImg1 ) / ( aIn + aImg1 );
aOut = 1;
}
```

The two images are blended.

NOTE Composite fully supports the CTL import statement and the CTL_MODULE_PATH environment variable used to search for CTL programs in a set of directories on disk. As such, it can be convenient to structure CTL code in Composite such that the main() function above is used simply as the entry point that calls other CTL functions defined in separate files that are imported with the import statement. You may set the CTL_MODULE_PATH in the Tool panel of the Project Preferences panel.

Accelerating CTL Transforms

Because CTL is an interpreted language, the performance may be slow. Composite gives you the option of accelerating a CTL program by converting it to a 3d-lut.This will be faster but may contain errors relative to the original CTL program.

To enable acceleration, click the 3D LUT Enable switch in the CTL tool's Processing tab. The Pre-conditioning option allows you to adjust the distribution of grid points in the 3d-lut. If the input image is in scene-linear color space, setting the Pre-conditioning to Log will arrange the grid points in a roughly logarithmic manner which will help distribute the approximation errors in a more perceptually uniform way.

Warping

19

Topics in this chapter:

About Warping

The Warping tools let you distort the physical shapes and contours of images.

Warp 2D

The Warp 2D tool lets you:

- Warp an image to change the geometry of some elements.
- Correct slight perspective errors in shots.
- Provide customized stylistic distortion effects like twirl or ripple, but with much more control over the animation of the geometry.

The basic operation of the Warp 2D tool involves drawing source and destination shapes and linking pairs of these together. The correspondence between points on the source and destination shapes defines constraints on the transformation of the image.

Another type of shape constraint is a fence shape which functions like a pair of source and destination shapes that are constrained to always be equal, keeping a part of the image from moving from its source position. The resulting transformation tries to make a deformation of 2D space that warps the image to satisfy the specified shape positions, while automatically repositioning the unconstrained parts of the image to make the overall warping as smooth as possible.

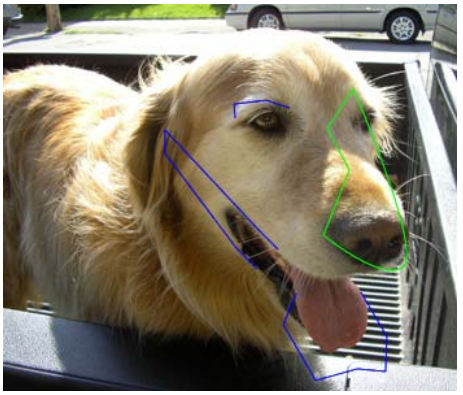
The types of shapes supported include those in the Garbage Mask and Remove Dust tool: Splines (open or closed), Rectangles, and Ellipses.

After drawing the shapes which define the transformation, you can set or animate an interpolation parameter which controls the amount of warping.To create a static warp of an image you will just set this to 1.Sometimes you may want the image to smoothly change from its original shape to a warped shape and will animate the interpolation parameter from 0 to 1.There is also a control to use a per-shape interpolation amount to have various features deform at different times.

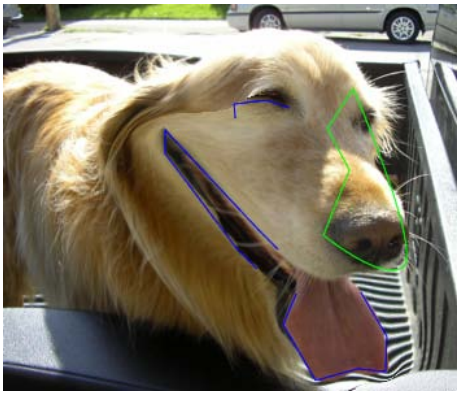
Warping Workflow

The warping of a single image involves a multi-step process. You first work in source space (by viewing the input image by pressing the 5 hotkey while focused in the viewer) drawing several shapes which define the important features that will be deformed and the fence shapes which will lock parts of the image in place (like the dog's left eye down to its nose).

You will then switch out of source space into destination space (by pressing the 6 hotkey while focused in the viewer) and begin editing the destination shapes.

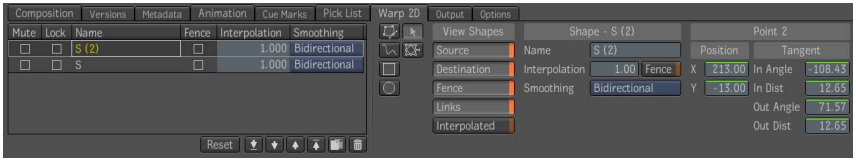


As you drag around a destination shape, the image underneath is warped accordingly. Once you have dragged all the destination shapes to their desired final positions, you may want to add another shape pair or fence shape to refine the warp. You will have to switch to source space to draw the starting position, then back to destination space to move the shape around and see the result.



Warp 2D UI

The Warp 2D UI consists of three tabs: Shape Drawing, Output, and Options.

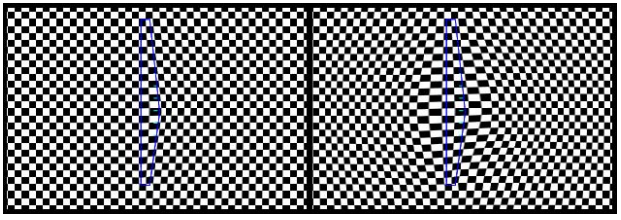


Shape drawing closely mimics that of Garbage Mask and Remove Dust tools. The types of objects that can be created are:

- Splines, open or closed
- Freehand shapes.
- Rectangles
- Ellipses
- Points (just a spline with a single vertex)

Like the Garbage Mask, there is a list box with the named shapes and individually editable parameters. The Warp UI also has all the components of the Garbage Mask for transforming shapes, setting drawing options for rectangles and ellipses and for loading and saving user settings—see Garbage Mask Controls on page 539.

Each shape has a parameter which controls the smoothing across its boundaries, named Smoothing. For open shapes, this can have one of two values: None or Bidirectional. The setting None means that each side of the shape is smoothed independently and there can be discontinuities in the warp across the shape. The setting of Bidirectional means that smoothing is applied across the shape and there will be no discontinuity in this region. If the shape is closed, then in addition to these two values, there are two other possible values: Inside and Outside. A value of Outside means that the inside of the shape is smoothed independently of the outside, but the outside is smoothed to match the inside. A value of Inside is the opposite of this.



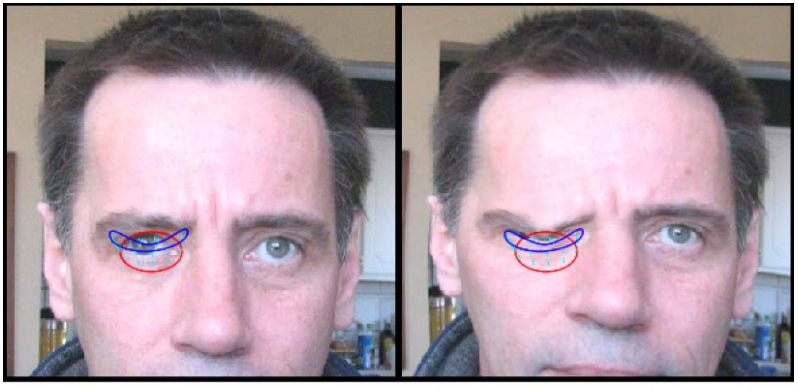
Smoothing = None

Each shape also has a Boolean parameter, Fence. When this is enabled, this shape acts like a pair of source and destination shapes which are constrained to be equal. This holds the image under the shape in place. When the Fence parameter is enabled, the destination shape and link points disappear. Fence shapes are treated differently from regular shapes. Fence shapes have their own visibility, and the link lines and link editing manipulations do not apply because the correspondence is defined to be the identity along the curve. However, each shape has a toggle to determine if it is a Fence or not, so you can always change a fence shape back to a regular pair of shapes.

NOTE When you create Fence shapes, you will usually want to set the Smoothing attribute of the Fence shape to None, to ensure that there is no motion crossing the border of the fence.

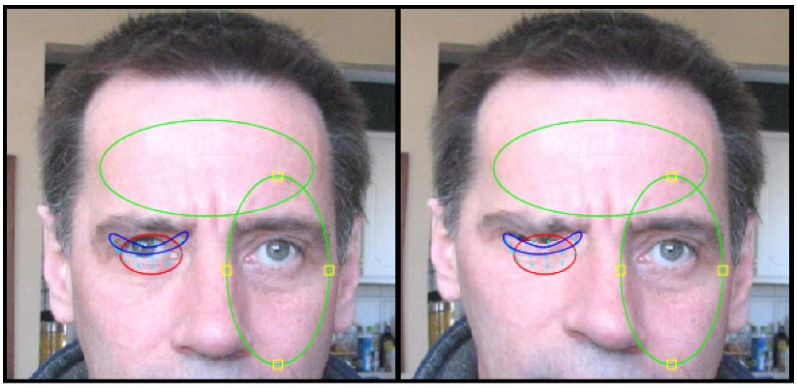
Each shape has an Interpolation parameter which is only relevant if the Timing mode is set to Shape (described in the Output tab section). Each shape also has a Mute switch to disable its effect on the warp, and a Lock switch that locks the points, shape, edges, and tangents of the shape.

You can change the name of a shape under the Name heading in the shape list. With the cursor in the text field, press **F2**, type in a new name and press **Enter**. You can also assign a name to a shape once you have selected what type of shape you want to draw. Under the Create Shape label, select the default shape name and type in a new one then press **Enter** or click outside the text field.



a b

(a) Source and Destination shapes (b) Warp result



a b

(a) Fences added to limit warp to the right eye (b) Warp result

The Warp 2D tool will have two outputs:

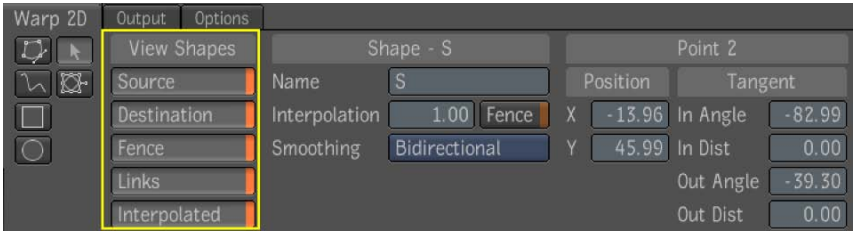
- Result (warped by the amount of the interpolation parameter)
- Input image warped by 100%

While editing, you can use the standard hotkeys (5 and 6) to switch between the input image and the second output, in order to view the source and destination spaces.

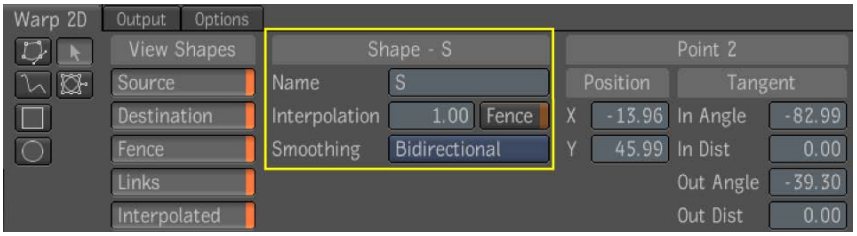
Shapes are displayed in the viewer with a color that depends on its type, e.g., source, destination, fence, links, or interpolated shapes.

Shape	Color
Source	Red
Destination	Blue
Fence	Green
Links	Cyan
Interpolated	Magenta

You can distinguish between and edit source and/or destination shapes by toggling on/off the visibility of each of the following classes of shapes:



- Source shapes
- Destination shapes
- Fence shapes
- Links (defines correspondences between the source and destination shapes)
- Interpolated shapes (viewable only, these cannot be edited, but show the interpolated position of the shapes based on the current interpolation parameter)



Other controls on the Shape Drawing tab include:

Use:	**To:**
Name	Give the selected shape a name. Select the default name in the text field and type in a new name then either press **Enter** or click outside the text field to apply the name.
Interpolation	Control how much warp is applied to the image—see Output Tab on page 475.
Fence	Enable/disable the shape as a fence.
Smoothing	Apply smoothing across the shape's boundaries.

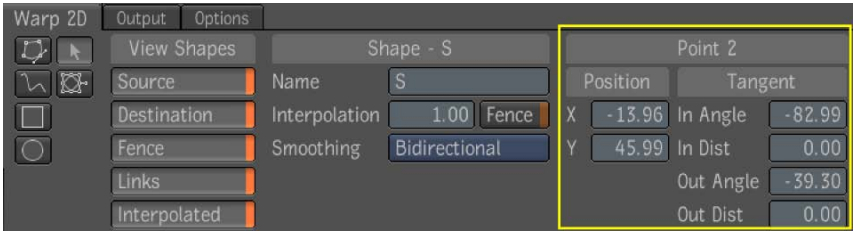
You can also adjust control points of the shape using the same controls the Garbage Mask uses—see Editing Masks on page 543. You can also copy and

paste a selected source or destination shape either in the same Warp 2D tool or from another.

To copy and paste a source or destination shape:

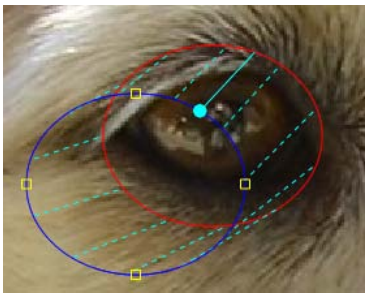
1. Select a source or destination shape to copy.
2. Use the **Ctrl + C** (for Windows and Linux) or the **Cmd + C** (for Mac OS) hotkey combination to copy the shape.
3. Use **Ctrl + V** (for Windows and Linux) or **Cmd + V** (for Mac OS) to paste the entire animation of the shape points and tangents.

NOTE The transform, and other properties are not copied.



Curve Correspondence

In addition to the usual spline editing interactions, you can also edit the correspondence between two linked shapes. In the image below, red curves depict source shapes, blue curves depict destination shapes, and cyan lines represent the correspondence between the pairs of linked curves. These lines show the direction the warped image will move as the interpolation parameter is animated from 0 to 1. You can add link points on either curve (using the **Ctrl** key) which can then be dragged along the curve to change the directions of the lines. Adding a point on one curve will also create a corresponding point on the opposite curve. Dragging one of these points sideways along the curve will skew the correspondence directions. Correspondence positions are animatable.

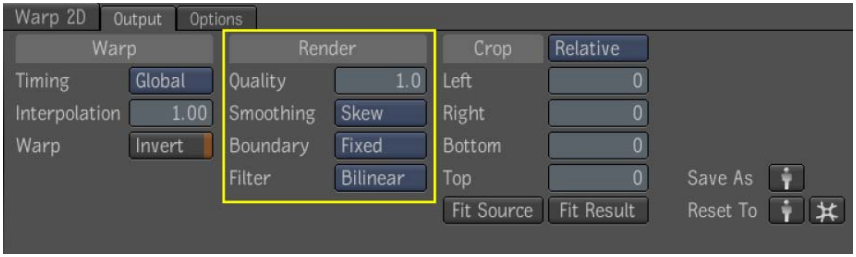


Output Tab

There are several parameters which control how the warped image is rendered and are located under the Warp label.

Use:	To:
Timing	Set the timing of the deformation. The default state is Global where all shapes deform at the same time based on the interpolation you set (see below), or you can select the timing to be on a per shape basis allowing you to deform various features on differing schedules.
Interpolation	Define how much of the warp is applied in the final rendering. A value of 0 means that the image is passed through unchanged. A value of 50 means that the image is deformed 50% of the distance toward the destination shapes.
Warp Invert	Send the warp backwards. Used in the morphing workflow—see Morphing Between Two Images on page 478.

There are several parameters which control how the image warp transformation is computed and they are located under the Render label.



Use:	To:
Quality	Control the quality versus speed of the deformation as higher quality warping takes more computation time. If you experience discontinuities or warps that are not smooth, or if the animation has regions that do not warp smoothly over time you should increase the Quality setting.

The warp of the image is computed to satisfy the constraints defined by the drawn shapes, and still be as smooth as possible. The choice of smoothing criteria is Rigid or Skew.

Use:	To:
Smoothing	Select the smoothing criteria; skew is a mode where the image transformation is locally constrained towards a 2D affine transformation, and the image is allowed to skew as it tries to find the smoothest warp. In Rigid mode, the image is locally constrained to an angle preserving similarity transformation where skew is discouraged but not impossible.

a b

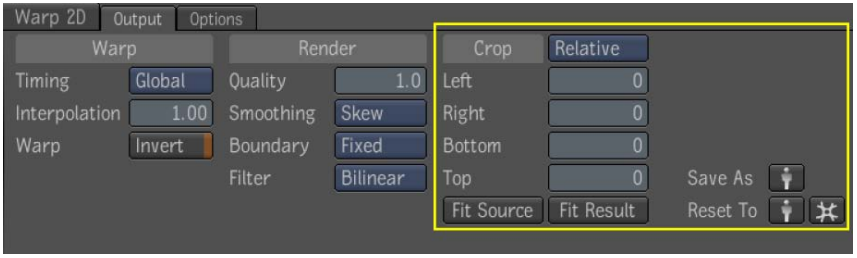
(a) Skew (b) Rigid

Use:	To:
Boundary	Set the image boundary to either Fixed or Free. Fixed holds the border of the image in place while Free lets the image expand beyond the image border.

a b

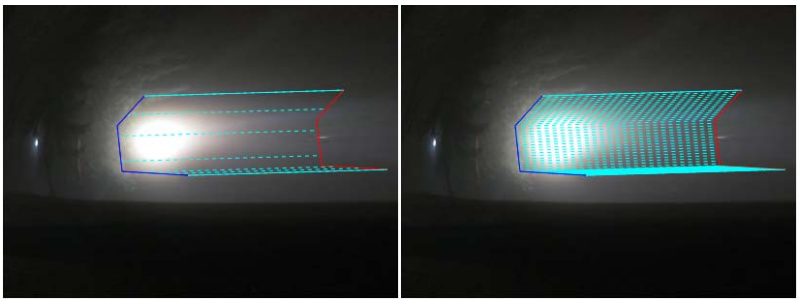
(a) Fixed (b) Free

Like the Garbage Mask, the Warp 2D tool's Output tab contains a crop tool and user settings controls—see Output Tab Settings on page 558.



Options Tab

The Options tab contains the same options and User Settings as the Garbage Mask Options tab—see Options Tab Settings on page 559, with the exception of the Links View parameter which allows you to change the spacing of the links between source and destination shapes.



Links spacing = 30

Morphing Between Two Images

Although the Warp 2D tool warps a single image, you can morph between two images using the following workflow:

To warp between two images:

1. Add a Warp 2D tool node to image A.
2. Place all your source shapes onto image A.
3. Change the viewer to be viewing image B using a context point, and align all the destination shapes on the corresponding features of image B.
4. Animate the Warp from 0 to 1 interpolation, which warps image A toward B.
5. Copy the Warp tool, select the Invert-Warp button and connect Image B as its input.This will warp Image B backwards towards A.
6. Add a Blend tool node between the two warp outputs and animate it from 0 to 1.

2D Transforms

With the exception of Lens Distort and the Warp 2D tool, all warping tools let you translate, rotate, scale and change the center of the input, effect and output image. Transformations on the input image will be done before applying the effect and its transforms. The transformations on the output image will be done after applying the effect and its transforms—see 2D Transform Tool on page 432.

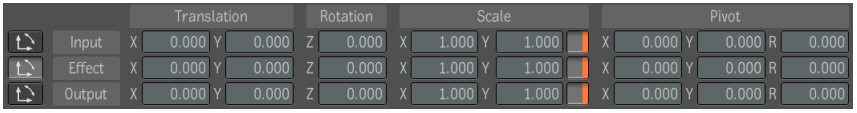
Affine transforms can be applied for free performance-wise, i.e. they can be applied at the same time as the warps without slowing down the tools. By combining transforms into a single resampling operation, the produced image has less degradation than if separate transform and warp tools were used. In addition, the combined transforms/warps approach is not susceptible to the bottleneck issue. For example, specifying an Output transform that enlarges the output image would give a completely different result that putting a resize tools afterward. In the former approach, the image would be directly computed at the final resolution, while in the latter approach the intermediate image would be at a lower resolution and the lost information could never be recovered.

Affine transforms allow integration with the tracker because the affine transforms are expressed using socket names that the tracker recognizes. The tracker can be accessed by right-clicking on the value editors or the Input/Output/Effect UI Container label.

Since Warping tools can be keyframed, you can create dramatic changes in image shapes when animated over time.

All Warping tools have masking inputs so you can limit the effect of the warp to a precise area of the image.

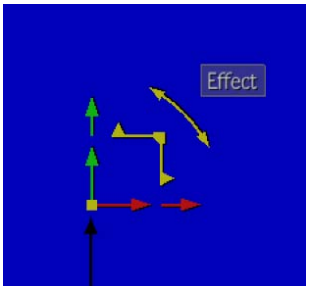
All Warping tools have the following common parameters:



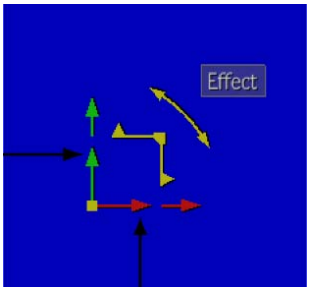
To translate the input image, warp effect, or output image:

➤ Do one of the following:

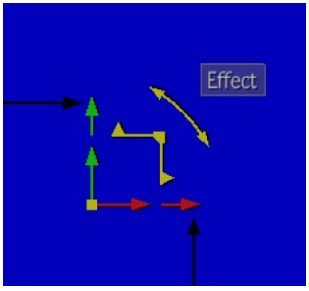
- Type values into the X or Y Translation value editors and press **Enter**.
- Drag the square found at the bottom left of the direct manipulator along the X and Y axes. Note that you must have Manipulators selected in the Display tab in Player Options.



The image can be moved along the Y or X world axis, respectively by dragging either the vertical or horizontal arrow protruding from the square.



The vertical and horizontal arrows extending further from the square act on the Y and X image axis respectively.

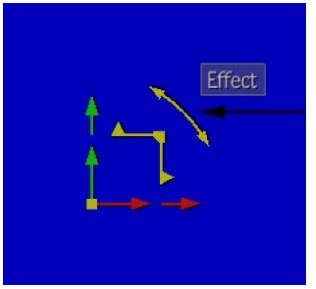


To rotate the input image, warp effect, or output image:

➤ Do one of the following:

- Type values into the Z Rotation value editors and press **Enter**.
- Move the curve at the upper right of the direct manipulator.

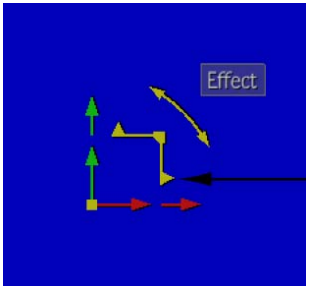
Note that the world and image axes are equivalent when there is no rotation.



To scale the input image, warp effect, or output image:

➤ Do one of the following:

- Type values into the X and Y Scale value editors and press **Enter**.
- Move the right angle in the centre of the direct manipulator. The arrowheads at each end restrict the scale to the respective axis.



To set the pivot point:

➤ Press the **Backspace** key to activate the pivot manipulator then do one of the following:

- Type values into the X and Y Pivot value editors and press **Enter**. Note that the scale manipulator is not available. The rotation manipulator is used to shear an image.

NOTE You can use the following hotkeys to quickly select a manipulator; press **M** to select the Input manipulator, press comma (,) to select the Effect manipulator, or press period (.) to select the Output manipulator.

Cropping and Filtering the Output

The Crop allows you to change the size of an image. Crop manipulators are visible when selecting the Output tab—see Crop Tool on page 410.

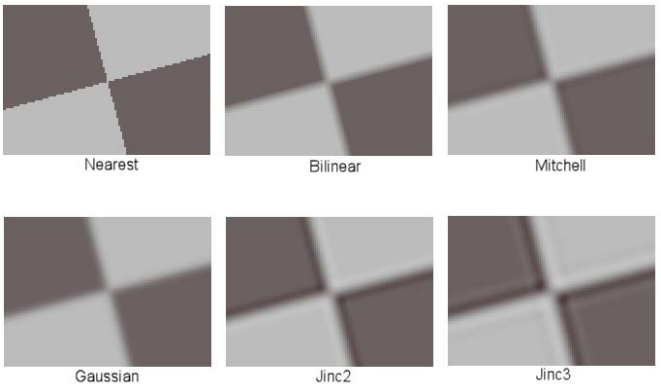
The filtering processes attempt to smooth the transformed pixels. When an image is being warped, there is a need to interpolate the old pixel map to a new one. This requires some form of image reconstruction and a number of resampling methods can be used.

All Warp tools have common filtering parameters:

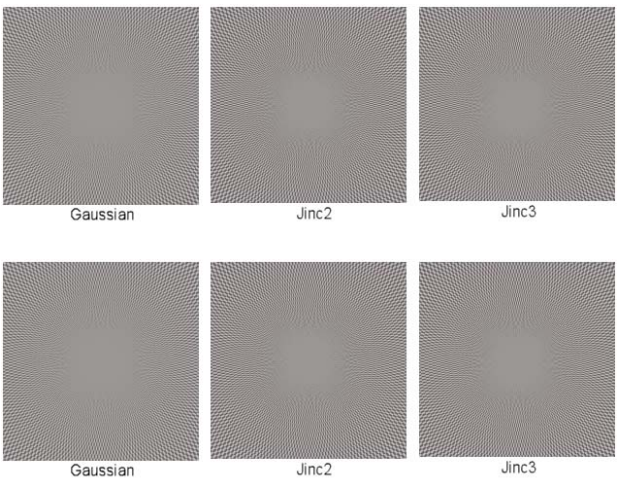
Filter	Description
Nearest	A box filter and the fastest way to resample an image since it only samples a single pixel of the input image to determine the value of a given pixel in the result image. Produces significant amount of aliasing.
Bilinear (default)	A separable triangular filter that takes into account more area when resampling.
Mitchell	Considered as one of the best magnification filters for images; has a good balance between ringing and sharpness.
Gaussian	Good magnification and magnification filter with no ringing, but introduces noticeable softness to the result image.
Jinc 2	Offers better sharpness than the Gaussian one, but at the expense of ringing. Similar to the sinc filter, but with better isotropic qualities, less ringing, and same sharpening and anti-aliasing.
Jinc 3	Offers better sharpness than the Gaussian and Jinc2 filters but at the expense of even more ringing. Similar to the Lanczos filter, but with

Filter	Description
	better isotropic qualities, less ringing, and same sharpening and anti-aliasing.

The next example illustrates the sharpening and ringing effects of the six filters.



The next example illustrates the anti-aliasing and blurring effects of the six filters. These examples were created with a checkerboard and polar tool to provide variance in frequencies in all directions.



Other common output parameters:

Use:	To:
AA Factor	Adjust the trade-off between anti-aliasing and blurriness.
Tiling X/Y	Set the repeat mode. Choose from; Transparent (default), Edge, Repeat, and Mirror.
Aspect	Control whether the mapping is stretched to cover the entire image region of definition (ROD) while in Image mode, or whether the mapping is only uniformly stretched to fit the height of the image (Circular mode). Note that in image mode, the mapping will be non-uniformly stretched. Particular mappings might be more suited to one aspect ratio mode or the other. Each tool specifies the proper default aspect ratio mode.

Warping Tools

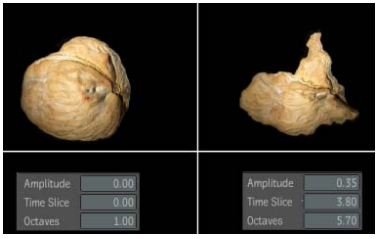
There are 10 Warping tools that let you distort the shape of your images:

Tool	Description
Crumple	For creating creases, kinks, and wrinkles in an image—see Crumple on page 485.
Displace	Offsets pixels in an image using pixel values of another image—see Displace on page 485.
Lens Distort	Rectifies or creates lens distortion—see Lens Distort on page 487.
Magnify	Enlarges a portion of an image in either the X, Y, or both axes—see Magnify on page 491.
Mirror	Creates a mirror effect in an image—see Mirror Tool on page 492.
Pinch	Compresses an image inward or outward—see Pinch Tool on page 492.
Polar	Changes the pixel definition of an image from Cartesian to the Polar coordinate system or vice versa—see Polar Tool on page 493.
Ripple	Creates ripple effects in an image—see Ripple Tool on page 494.
Twirl	Rotates and twists an image—see Twirl Tool on page 495.

Tool	Description
Wave	Creates a wave effect in an image—see Wave Tool on page 496

Crumple

The Crumple tool lets you create an image which appears crumpled like a piece of paper.



Original image

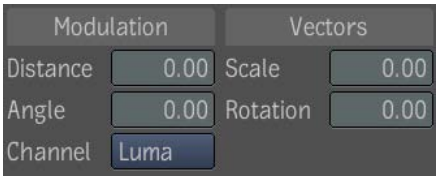
The Crumple tool has the following parameters:

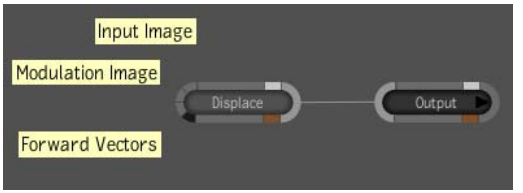
Use:	To:
Amplitude	Control the overall strength of the filter.
Time Slice	Make modifications to the fractal patterns yielding random results.
Octaves	Control the turbulence. Increase the octaves to get more iterations, therefore, a more turbulent crumpling effect.

Displace

The Displace tool lets you use a control image to drive the displacement used to warp the primary input image. There are two ways to displace a primary

image: Modulation and Vectors. Both ways can be combined together by specifying the Modulation Image and a Forward Vectors image.





Displacing with Modulation

You can use a single channel to modulate the displacement along a fixed direction. You can define the fixed distance and angle and the displacement increases or decreases by the intensity of the channels (Red, Green, Blue, Alpha or Luma) of the modulation image.

Use:	To:
Distance	Specify the distance of displacement in pixels.
Angle	Specify the angle of displacement all pixels in the image will be rotated by in degrees.
Channel	Select which channel from the modulation image will be used to displace the image in a fixed direction as specified by the distance and angle.

Input image

Displacing with Forward Vectors

You can use a forward vector image to define the displacement of the primary input image. Each pixel can be moved in a different direction. The red component of the image represents the X direction of the motion vector and the green component the Y direction. The vectors can be scaled and rotated before applying the result on each input image pixels.

Use:	To:
Scale	Scale all motion vectors of the forward vectors image.
Rotation	Apply a rotation to all motion vectors of the forward vectors image. The rotation is expressed in degrees.

Input image

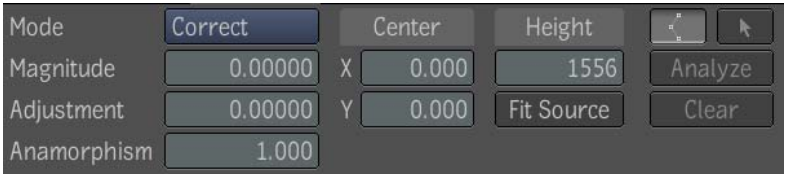
TIP Decreasing the opacity of the Modulation or Forward Vectors image will decrease the distance of the displacement.

Lens Distort

The Lens Distort tool lets you create or correct lens distortion that may be present in image sequences.

Barrel distortion is associated with wide angle (or minimal zoom) lenses. It causes the images to appear slightly curved outward like a barrel. You can notice this when you have straight features close to the image's peripheral sides.

The Lens Distort tool has the following parameters:



Use:	To:
Mode	Select either lens correction or distortion. This affects all other parameters. The other parameters will be considered either in a distortion or in a correction work flow. The distortion transformation is exactly the inverse of the correction, so keeping the same parameters and putting back to back distortion plus rectification will give back the original result, but filtered twice at each transformation resulting in possible degradation in image quality.
Magnitude	Set the first parameter (k1) of the lens distortion equation: rd = r + k1 * r^3 + k2 * r^5.
Adjustment	Set the second parameter (k2) of the lens distortion equation: rd = r + k1 * r^3 + k2 * r^5.
Anamorphism	Characterize anamorphic asymmetry. Note that this has nothing to do with the pixel aspect ratio of the captured image.
Center	Set the X and Y coordinates of the center of the distortion. Those parameters are expressed in pixels. The lens center can also be changed by clicking directly in the player. The center manipulator is always active in the player. The look of the manipulator is a cross. The cross can be manipulated by selecting it and moving it.
Height	To determine the region where the lens distortion or correction is applied. This parameter is initialized to the composition height.
Fit Source	To set the height input to the input height.This parameter is needed to be able to perfectly inverse a lens correction.

Expected Workflows

The following are common use cases for lens distortion or correction:

- Footage to be used as a background layer has noticeable lens distortion and footage to be composited over the background does not. The Lens

distort tool would be added to the dependency graph and used (in Correct mode) to rectify the distortion before compositing the new layer over it. The composite will then be made (any compositing operation) and a second instance of the correcting Lens Distort tool would be added after the compositing operation has taken place using the same values but in Distort mode applying the original distort to both layers.

- Applying lens correction on multiple compositions before compositing them together.
- Reproduce the lens distortion of one composition on another composition.
- Using the tracking data from one composition and applying it to the center of another distorted composition, or composited layers, producing interesting animated effects.

Analysis of Radial Distortion

Analysis of radial distortion computes the magnitude and adjustment parameter from a three point spline created in the player to identify distortion. The analysis is only available when the lens mode is set to "Correct". All parameters except the magnitude and adjustment are used to compute the new magnitude and adjustment. The magnitude input will be set at the current frame when the analysis is finished. Note that only one spline can be created at the time. The spline is not animatable.

To analyze radial distortion with the spline tool:

1. Set the Mode to Correct.
2. Draw a three point spline along edge of any distorted features within the image that should be straight. When three points are drawn, the select tool will be activated.

3 Modify the existing spline.

4 Click the Analyze button to analyze radial distortion from the created spline. The image is corrected and the Magnitude and Adjustment fields are updated.

5 To remove the spline, click Clear. Note that this removes the editable spline only, but does not remove the correction. If you add a new spline, corrections will be additive.

6 To begin a new analysis, click the Reset button in the Tools Options.

7 To correct an analysis, use Undo (**Ctrl + Z** for Windows and Linux or **Cmd + Z** for Mac OS).

Output

The Lens Distort tool has an Output tab with the following parameters:

Use:	To:
Crop	Change the size of an image—see Cropping and Filtering the Output on page 482.
Filtering	Smooth the transformed pixels—see Cropping and Filtering the Output on page 482.

Magnify

The Magnify tool lets you magnify a region of an image.

Original image

The Magnify tool has the following parameters:

Use:	To:
Amount	Control the power of the magnification. Default setting is 1.00000.

Use:	To:
Direction	Set the direction of the magnification. Setting to Horizontal results in a rectangular magnification region, enlarged in the horizontal direction. It can also be set to Vertical which enlarges the image in the vertical direction. Or it can be set to Both resulting in a circular magnification region, such as a magnifying glass. Default setting is Both.

Mirror Tool

This tool mirrors the image along the Y=0 axis. There are no additional controls in this tool beyond the standard warping controls.

Original image

Pinch Tool

The Pinch tool lets you pinch or squeeze an image either inward or outward from a defined center point.

Original image

The Pinch tool has the following parameters:

Use:	To:
Amount	Control the severity of the pinch. Positive pinch pulls the image inside the cone, making it appear to recede. Negative values stretch the image over the cone, moving the center point closer to your point of view.

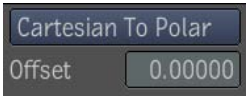
Polar Tool

The Polar tool lets you transform the input image from Cartesian coordinates to Polar coordinates or vice versa. When using Polar to Cartesian, the effect stretches a round object so that it appears to straighten out. When using Cartesian to Polar, the effect bends an image around its center so that it appears round.

An option menu is used to choose if the transformation is from Cartesian to Polar or Polar to Cartesian.

Original image

The Polar to Rectangular tool has the following parameters:



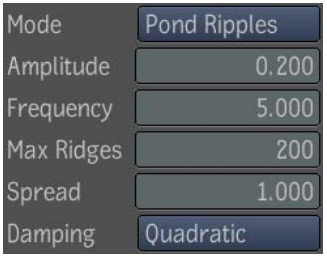
Use:	To:
Mode	To switch between Polar to Cartesian and Cartesian to Polar (default is Cartesian to Polar).
Offset	Rotate the result of the Cartesian to Polar conversion and offset the result of the Polar to Cartesian conversion. The value is expressed as an angle that represent a circular offset in Cartesian To Polar and an horizontal offset Polar to Cartesian mode.

Ripple Tool

The Ripple tool lets you create ripple-like effects from the center of the image outward.

Pond Ripples

The Ripple tool has the following parameters:



Use	To
Mode	Determine the ripple effect. ■ Pond Ripples create ripples with a 45 degree offset. ■ Out from Center pushes the ripples away from the center, extending the first half, and compressing the second half of each ripple. ■ Around Center rotates the crest of each ripple in a clockwise direction around the center.

Use:	To:
Amplitude	Scale the amount of warping distortion. Increase for more severe distortion.
Frequency	Set the number of waves per frame.

Use:	To:
Max Ridges	Control the total number of ripples that can be generated. For example, set Max Ridges to 5 to create 5 rings of distortion; the area inside and outside of the rings are unaffected.
Spread	Set the distance between the ripples and the center. Animate the spread to create continuous rippling of the image.
Damping	Control how the ripple amplitude is decreased over distance. ■ Use None so all the waves have the same amplitude specified by the amplitude parameter. ■ Use Linear so the amplitude of the waves at the center has the specified amplitude and decrease linearly to 0 at the image extremities. ■ Use Quadratic so the amplitude of the waves at the center has the specified amplitude and decrease quadratically to 0 at the image extremities.

Twirl Tool

The Twirl tool lets you spin, twirl and rotate the image.

Original image

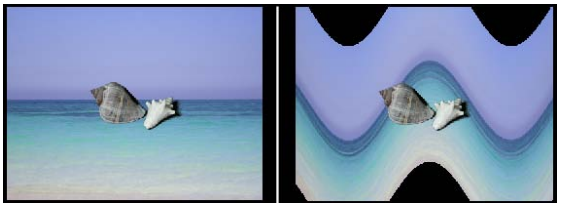
The Twirl tool has the following parameters:

Use:	To:
Amount	Control the severity of the twirl. Positive twirls the image clockwise. Negative values spin the image counter-clockwise. The twirl

Use:	To:
	angle can also be changed with a rotation manipulator in the player.
	Twirl the rotation.

Wave Tool

The Wave tool lets you simulate waves in a single direction (as opposed to circular waves as found in Ripple).



Original image

The Wave tool has the following parameters.

Use:	To:
Amplitude	Scale the amount of warping distortion. Increase for more severe distortion.
Compression	Warp pixels parallel to the wave direction.
Frequency	Set the number of waves per frame. Increase for more waves, decrease for fewer.
Phase	Shift the position of the waves along the direction of wave motion.

Effects Tools

20

Topics in this chapter:

About Effects Tools

Effects tools let you add lighting and shadow effects to your images.

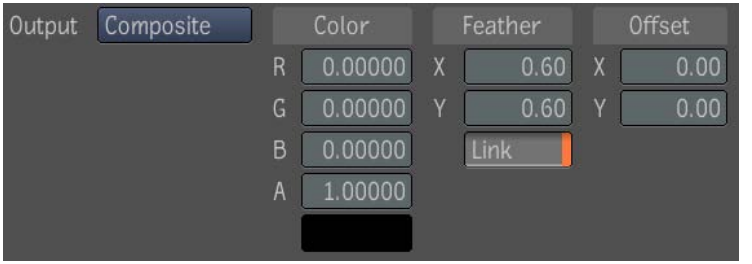
There are two effects tools available:

Tool	Description
Drop Shadow	Adds a drop shadow to an image—see Drop Shadow on page 497.
Glow	Generates subtle gradations of light in your composition—see Glow on page 498.

Drop Shadow

The Drop Shadow tool takes an input image and adds a drop shadow to it by taking the alpha channel of the input image, blurring it, offsetting it, and

coloring it; the resulting shadow can then be sent to the output or composited under the original input image.



The Drop Shadow tool has the following parameters:

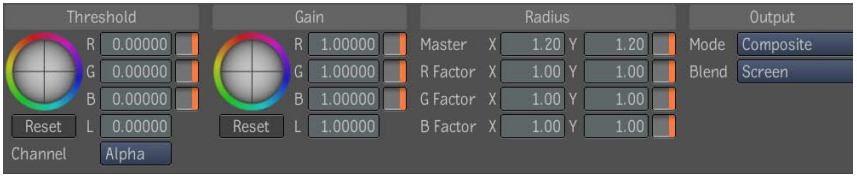
- **Output** Controls whether to output the shadow alone or composited under the input image. The UI will present a pull-down menu with Composite and Shadow Only items. By default, the output mode is Composite.
- **Color** Controls the color of the shadow; by default, the shadow color is black.
- **Feather Radius X, Radius Y, and Link** Controls the radius of the blur (just like in the Blur tool). By default, the X and Y radii are linked and set to 0; otherwise, their range is the same as that in the Blur tool.
- **Offset X and Offset Y** Controls the offset of the shadow with respect to the input image; by default, the offsets are 0. The Drop Shadow tool supports non-integer shadow offsets so that animating those parameters will still produce a smooth result. The offsets can be controlled using direct manipulation.

NOTE This tool always outputs an RGBA image. Its only affects mode is RGBA.

Glow

The Glow lets you generate subtle gradations of light in your composition.

The Glow tool has the following parameters:



- **Threshold** Lets you determine which parts of the image (usually the brightest) you want to apply the glow to. You can base the threshold either on a channel; Red, Green, Blue or Alpha, or Luma, or on a color component of the image (R,G,B or overall luminance). Values that fall above the set threshold will have a glow applied and values that fall below the threshold will be set to black.
- **Gain** Colors the glow you apply to an image.
- **X Radius and Y Radius** Applies a blurred edge to the glow. Using the Master slider will apply a blurred edge to both the X and Y radii uniformly. You can also select a single color component or select just the X or Y parameter to apply the blur to by deselecting the Lock button located at the right of the button.
- **Output Mode** Choose between working with either the composite of the image you are working with or just the glow applied to it. If you select Composite, it applies the blend mode selected.
- **Blend** Select the blend mode you want to apply to the glow. Select Blend to display available blend modes.

To set the threshold:

➤ Do one of the following:

- Drag the trackball towards the color you want to set as the threshold.
- Drag any of the red (R), green (G), or blue (B) channel fields to the right to increase, or to the left to decrease the threshold uniformly on all three channels. Alternatively, you can also drag the luminance (L) field to achieve the same result. Select the Reset button to return all values to 0.
- Select a single channel that you want to modify by deselecting the Lock button to the right of the channel, then increase or decrease the threshold. The other channels are not affected. However, the overall luminance field updates to reflect the change. Select the Reset button to return all values to 0.

- Type values into the channel fields and press **Enter**. In the case of modifying all channels uniformly, when values are typed into any field, the other fields update. The luminance field will update regardless of which method is used to modify the values. Select the Reset button to return all values to 0.

To set the gain on a glowed portion of an image:

➤ Do one of the following:

- Drag the trackball towards the color you want to add or decrease gain to.
- Drag any of the red (R), green (G), or blue (B) channel fields to the right to increase, or to the left to decrease the gain uniformly on all three channels. Alternatively, you can also drag the luminance (L) field to achieve the same result. Select Reset to return all values to 0.
- Select a single channel that you want to modify by deselecting the Lock button to the right of the channel, then increase or decrease the gain. The other channels are not affected. However, the overall luminance field updates to reflect the change. Select the Reset button to return all values to 0.
- Type values into the channel fields and press **Enter**. In the case of modifying all channels uniformly, when values are typed into any field, the other fields update. The luminance field will update regardless of which method is used to modify the values. Select the Reset button to return all values to 0.

To set the X or Y Radius to the edge of a glow:

➤ Do one of the following:

- Drag any of the red (R factor), green (G Factor), or blue (B Factor) channel fields to the right to increase, or to the left to decrease the radius uniformly both the X and Y radii. Alternatively, you can also drag the Master field to achieve the same result on all color factors.
- Type values into the channel fields and press **Enter**. To deselect the lock on any parameter, click the Lock button.

Managing Film Grain

21

Topics in this chapter:

About Managing Grain

Grain is a basic characteristic of film. Film grain consists of individual silver halide crystals that are randomly distributed across an image. The random distribution of the crystals creates the visual impression of graininess. Grain adds a distinctive look to film, and different film stocks have different grain signatures.

Grain added to composition.

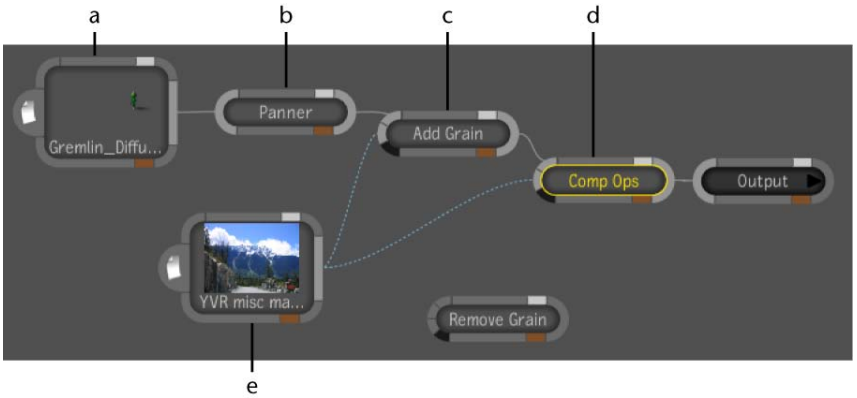
The grain management tools let you create convincing composites with film material. You may want to add grain to clips that originate from video to give them a film look.

The grain management tools are useful when:

- Mixing film clips originating from different film stocks; you may want to match the grain in the clips to make your results more consistent.
- Working with footage originally shot on film that was transferred to video, you may want to remove the grain if you want to mix film material with video material.
- Working with CG-generated elements, you may want a film look result.

Adding Grain to a Composition

The Add Grain tool lets you sample the grain from one image (the reference image) and apply it to another image (the input image) or sequence. In the following example, a CG character is used as the input image and the mountain background is used as the reference image (the grain will be analyzed from this image).

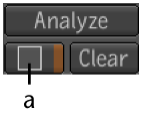


(a) Input image of CG character on which grain will be applied. (b) Panner tool to place the CG character at the desired location in the reference image. (c) Add Grain tool (d) Comp Ops tool to composite the CG character over the background(A over B). (e) Reference image from which the grain is sampled.

Once the dependency graph is built, you can decide which area of the reference image to sample. You can select samples from multiple areas of the image. You should use an area of the image that just contains grain, avoiding areas with image or edge detail. Once you have positioned the sample area, you can apply grain to the input image. After adding grain to the input image, you can manually adjust the response curves for the gain and size of the grain.

To add grain to a composition:

1. From the Tools tab, drag the Add Grain tool from the Grain Management folder to the dependency graph in the Schematic view.
2. Connect a reference image and an input image to the Add Grain tool.
3. (Optional) Add a mask to your input image to restrict the grain to a portion of the image—see Creating Masks on page 540.
4. In the Add Grain tab, do one of the following:
   - Click the Analyze button.
     Regions of the reference image that are appropriate for analysis are analyzed and grain is applied over the entire composition.
   - Click the Selection button and drag to define an area to analyze. For best results, define multiple regions of uniform color, avoiding areas that may resemble grain such as gravel, sand, or a forest. Click the Analyze button to create the grain. Click Clear to clear any areas you defined and start again.



(a) Selection button

The regions you defined are analyzed and grain is applied over the entire composition. Response curves are generated and displayed in the Animation Editor. You can now fine-tune the resulting curves—see Fine-tuning the Grain on page 505.

Removing Grain from a Composition

Removing grain from a composition allows you to composite two grainy clips that don't match, or simply to clean up a clip. The Remove Grain tool lets you remove as much grain as possible while minimizing damage to the image. You can specify how many forward and backward frames to include when removing grain from a composition. These frames can be warped by motion vectors to remove motion and improve the grain removal process. Increasing the number of frames will improve grain removal performance, but adds processing time. In some cases, it may also introduce motion artifacts if used without motion vectors or with bad motion vectors, especially in scenes with a lot of non-uniform motion.

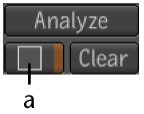
The Remove Grain node has two secondary inputs for receiving forward and/or backward motion vector data. When motion vectors are connected to these inputs, the forward and backward frames are first warped appropriately to remove motion before being used in the remove grain process. Also, you can automatically compute the motion vectors if they are not connected—see About Motion Vectors on page 711.

To remove grain to a composition:

1. From the Tools tab, drag the Remove Grain tool from the Grain Management folder to the dependency graph in the Schematic view.
2. Connect a reference image and an input image to the Remove Grain tool.
3. (Optional) Add a mask to your input image to restrict the grain to a portion of the image—see Creating Masks on page 540.
4. In the Remove Grain tab, do one of the following:
   - Click the Analyze button.

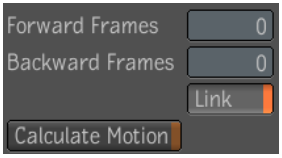
Regions of the reference image that are appropriate for analysis are analyzed and grain is applied over the entire composition.

- Click the Selection button and drag to define an area to analyze. For best results, define multiple regions of uniform color with no features. Click the Analyze button to create the grain. Click Clear to clear any areas you defined and start again.



(a) Selection button

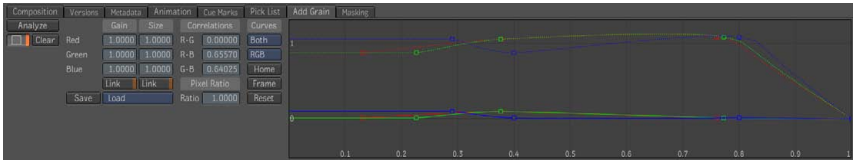
- Enter the number of forward and/or backward frames you w ant to use, and click Calculate Motion.



The motion vectors in the forward and backward frames is used to analyze the type of grain present in the composition. Grain is removed from the composition. Response curves are generated and displayed in the Animation Editor. You can now fine-tune the resulting curves—see Fine-tuning the Grain on page 505.

Fine-tuning the Grain

Once you have generated or removed grain using the Add Grain or Remove Grain tools, you can manually adjust the response curve for the gain and size in the Animation Editor. The solid curve represents the gain and the dotted line curves represent the size of the grain.



To fine-tune the grain:

➤ Use any of the following controls on the Add Grain or Remove Grain UI:

Use:	To:
Red, Green, Blue	Adjust the values of the RGB channels for the gain and size of the grain.
Link	Link the R, G, and B channels of the gain or size, so when you change the value of one channel, the others change as well.
Save	Save a grain profile for use in other compositions—see Saving and Loading Grain Profiles on page 507.
Load	Load a grain profile you previously saved—see Saving and Loading Grain Profiles on page 507.
Correlations	Specify correlations between the grain present in different color channels. There may be some statistical correlations between the grain present in different color channels depending on the film properties and development process.
Pixel Ratio	Adjust the discrepancies between pixel aspect ratio and grain aspect ratio.

Viewing the Response Curves

You can determine how to display the response curves in the Animation Editor.

To view the gain or size:

➤ Click the first button under Curves, and select Gain or Size.

To view the red, green, or blue curves:

➤ Under Curves, click the second button and select Red, Green, Blue, or RGB.

To display the response curves as generated by the Grain tool:

➤ Under Curves, click the Home button.

To frame the response curves:

➤ Under Curves, click the Frame button.

To reset the Animation Editor:

➤ Under Curves, click the Reset button.
The currently selected curve is set to 0.

Saving and Loading Grain Profiles

A grain profile is a set of parameters that describes grain appearance. Once you create a grain profile through the Add Grain or Remove Grain tool, you can save it and reload it for use in other compositions. If one or more of the grain parameters are animated, only the parameters at the current time are saved in a grain profile.

When saving or loading a grain profile using the Remove Grain tool, the number of forward and backward motion vectors and the state of the automatic motion vector computation are saved. The Add Grain tool does not use these values when you load a grain profile, but does put in some reasonable values when you save a grain profile.

To save a grain profile:

➤ In the Add Grain or Remove Grain UI, click the Save button and give a name to your profile.

To load a grain profile:

➤ In the Add Grain or Remove Grain UI, click the Load button and select a grain profile.

Pulling Keys and Creating Mattes

22

Topics in this chapter:

Keying Concepts

Understanding the concepts that are fundamental to the workflow and tools used for keying will help you work efficiently and effectively, producing the cleanest, most convincing result.

You can create composites by defining transparent regions in a foreground image, based on a specific range of color, to reveal a background image. This keying process is usually performed on footage with a subject in front of a blue or green screen. When you key out a color, you generate a matte: a black and white template indicating which parts of the image are transparent (black), and

which are opaque (white). Using techniques, such as softening the edge of mattes, you can refine the results to create realistic composites.

Keying

Keying is the process of isolating a region of an image by selecting pixels of a particular color and making those pixels transparent, or creating an alpha channel where those pixels are located. The main purpose of using the keyer tools in Composite is to generate mattes and remove color spill from a shot.

The two basic approaches to creating an alpha channel are based on:

- **Pixel color** The removal of pure green pixels for example.
- **Luminance** The removal of pixels that are very light.

You can also use garbage masks to key out undesired elements in an image. However, garbage masks will be discussed in a later section as creating garbage masks are a manual operation that involves creating paint objects or the creation and manipulation of splines (rotoscoping)—see Masking Concepts on page 538.

Mattes

A matte is an image used to define or control the transparency of another image. When you pull a key, you generate a matte that defines the transparency of the front source. There are several types of mattes.

Articulate matte A matte whose shape changes over time and is designed to accurately follow the contours of the object to which it corresponds.

Complementary matte The matte that results when the primary matte is inverted.

Difference matte A matte created by subtracting an image in which the subject is present from an otherwise identical image in which the subject is not present.

Edge matte A matte that includes only the outlines or borders of an object.

Fixed or static matte A matte that does not change position or shape over time.

Garbage matte A matte that isolates unwanted elements from the primary element in an image. Garbage mattes are also referred to as garbage masks.

Hold-out matte A matte that prevents a foreground element from completely obscuring an object in the background.

Traveling matte A matte that changes position or shape over time.

Color Models

A color model is a means of identifying colors in a source according to its component parts.

RGB A color model that defines pixels according to red, green, and blue channel values.

Hue

A specific color from the color spectrum, disregarding its saturation or value.

Saturation

The brilliance or purity of a given color. For example, the difference between a pastel and a pure color is defined by the amount of saturation.

Chrominance

Chrominance is an image processing property that defines the hue and saturation of a pixel.

Luminance

Luminance is an image processing property that defines the brightness of a pixel. Expressed in percentages, 0% luminance is black and 100% luminance is white. Values between 0 and 100% define a range of grays. Colored pixels have a luminance value equivalent to the mean of their RGB values.

Sharpness

Sharpness can be defined as the visual sense of the abruptness of an edge. It highlights the detail in an image or image sequence.

Softness

Colors that fall within the softness range are partially transparent. These areas appear gray (a dark gray to light gray range) in the matte. For example, softness makes the transition between the foreground and background of a composition more convincing by softening what would otherwise look like an abrupt cutout. Softness can also be used to retain transparent key-in materials or shadows that you want to appear in the result composition.

Tolerance or Threshold

The colors that fall within the tolerance or threshold range are transparent. These areas appear black in the matte.

Computer Generated Imagery

Images created or generated with a computer are referred to as Computer Generated (CG) images.

Keying Workflow

The procedures and tools used to pull a key depend on the source material you are working with, and on the complexity of the task. For example, sources shot under ideal lighting conditions with good contrast between the key-in and key-out content can be done very quickly. Other sources, with varying lighting conditions, fine detail at the edges of the key, dramatic camera movement, and other problem areas can take much longer to produce a clean result. Make sure you understand what keying task, or keying tasks, need to be performed so you can select the keying tools and the order of the keying tasks that will produce the best result.

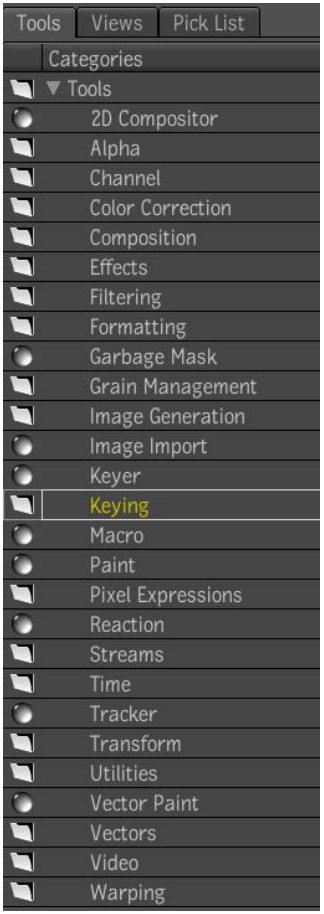
This chapter provides some keying scenarios using different keying tools to help you understand how to use the tools. When you pull a key, you may not necessarily need to perform all the keying scenarios in the order presented. Moreover, there will be instances where one tool may provide you with a clean key, and other times where a combination of keying tools will be needed to generate the desired keying result.

Selecting Keying Tools

When you key an image or an image sequence, you can use a keyer or a collection of keying tools to generate a matte or remove color spill from a shot. You may also need to generate more than one key to achieve the desired result.

To access the Keying tools:

1. Press the tilde key (~) or middle-click to display the Gate UI.
2. Swipe through the east gate to display the Tools tab.



3. Expand the Keying tool folder to access the Keying tools.

The Keying tools include Color Curves, Diamond Keyer, the Difference tool, and Luma Keyer.

To add a keying tool to the process tree:

1. From the Tools tab, drag a keying tool to the process tree in Schematic.
2. Drag the Keying tool over the output of a node that you wish to key. When the connection is highlighted, release the mouse.

   The keyer tool is added to the process tree.

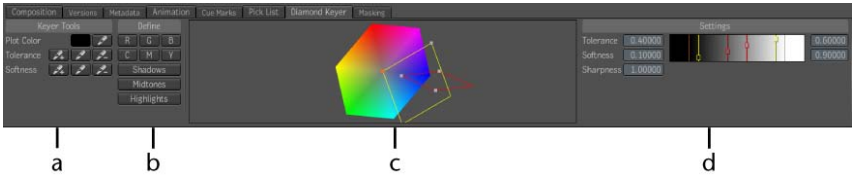
Extracting a Key Using the Diamond Keyer

The Diamond Keyer lets you extract keys in several different ways. You can pull a key based on:

- Pixel luminance values
- Shadows, midtones, or highlights
- Color channels
- Pixel color

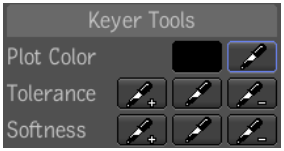
The Diamond Keyer

The Diamond Keyer tool is divided into four areas:



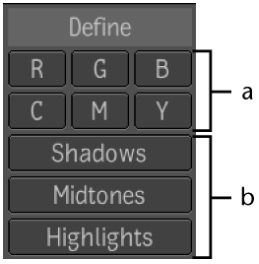
(a) Keyer Tools (b) Channel/Range Definition (c) Hue Cube (d) Luminance settings

The Keyer Tools area contains the following parameters:



Use:	To:
Plot Color	Plot a pixel or range of pixels to key in an image.
Tolerance	Set tolerance range based on current value of a sampled pixel or range of pixels.
Softness	Set softness (of the matte edge) range based on current value of a sampled pixel or range of pixels.

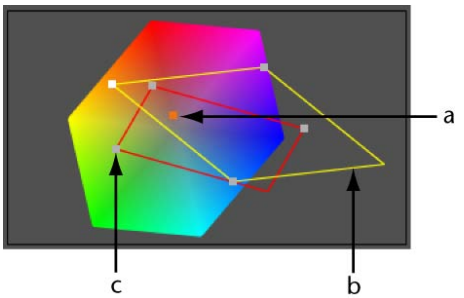
The Define area contains the following tools:



(a) Color Channels select (b) SMH select

Click:	To:
RGB or CMY buttons	Remove that color from the image.
Shadows, Midtones or Highlights buttons	Key a luminance range. Use these controls to extract a key based on an element's Z-depth information (grayscale value).

The Hue Cube allows you to visualize and manipulate color tolerance and softness.



(a) Plotted color within tolerance range (b) Softness wireframe diamond (c) Tolerance wireframe diamond

You can drag the three control points of the tolerance wireframe diamond to cover more or less of the hue spectrum.

To solve problem areas, use the Key Color eyedropper to plot a color in the image. The sampled color is indicated by an orange dot in the hue spectrum. By moving the control points of the tolerance line, you can add the plotted color to, or remove the plotted color from, the tolerance range.

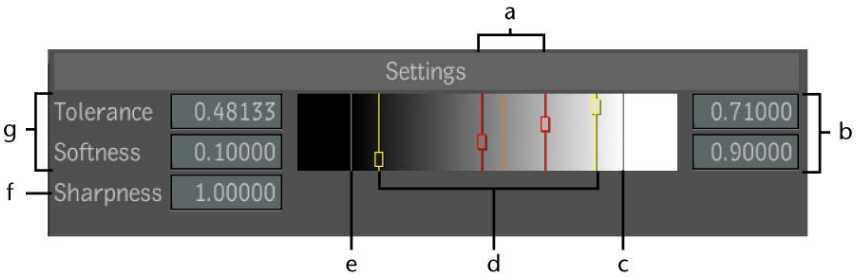
To increase the softness click the +eyedropper next to the softness parameter and sample the image. To decrease the softness, click the – eyedropper next to the softness parameter and sample the image.

In all cases, the values sampled are added to the current softness range. The corresponding hue and luma ranges are sampled. This range is indicated by yellow wireframe lines in the hue spectrum.

You can adjust the softness in the hue spectrum by dragging the three control points of the softness line to cover more or less of the hue spectrum.

To solve problem areas, use the same method as you would for tolerance problems. Use the Key Color eyedropper to plot a color in the image. The plotted color is indicated by an orange dot in the hue spectrum. By moving the control points of the softness line, you can add the plotted color to, or remove the plotted color from, the softness range.

The luminance Settings area contains the following parameters:



(a) Tolerance sliders (b) Tolerance and Softness value fields (white range) (c) 1.0 white indicator (d) Softness sliders (e) 0.0 black indicator (f) Sharpness value field (g) Tolerance and Softness value fields (black range)

Use:	To:
Tolerance	Change the tolerance's luma range (drag the cursor or enter a value in the Tolerance value field).
Softness	Change the luma range of the softness (drag the cursor or enter a value in the Softness value field).
Sharpness	Change the sharpness of a key. Sharpness filters the softness range independently of the pixel sampled from the key-in composition. Therefore, the softness range is filtered according to its luma values in the generated matte. Increasing sharpness reduces softness, decreasing sharpness increases softness.

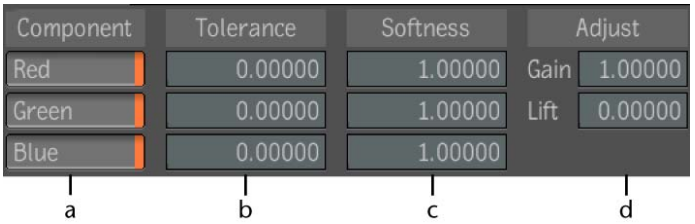
NOTE Tolerance, Softness and Sharpness are animatable attributes—see Marking Attributes for Keyframing on page 674.

Creating a Difference Matte Using the Difference Tool

The Difference tool lets you generate a matte image from two source images with the same background, but different foreground elements. This allows you to remove an image from one context and add it to another. You can even generate a composite image while creating the difference matte.

When you generate a difference matte, the difference between corresponding pixels of the two source images is calculated. The value of the pixel in the back image is subtracted from the value of the corresponding pixel in the front image, and the resulting value is used in the difference matte.

The Difference tool contains the following parameters:



(a) Channels (b) Tolerance value fields for RGB channels (c) Softness fields for RGB channels (d) Gain and Lift adjustment

The matte is created using Tolerance and Softness values. The Tolerance value specifies the difference level that is considered black. A high Tolerance value includes more black in the matte. The Softness value is used to soften the transition between the light areas and the dark areas of the matte by adjusting the amount of gray at its edges. Gray information is not included in the matte when the Softness is zero. Use a high Softness value to increase the gray.

Choose one or more color channels to use for the difference matte, and then set the tolerance, softness, gain and lift.

Extracting a Key Using the Luma Keyer

The Luma Keyer computes the luminance of the image and removes pixels based on a threshold value, affecting the alpha channel. The softness value can be added to set some fall-off for the key. When the softness is 0, only the pixels that fall within the tolerance range are keyed out. The luminance keyer can either output a matte or a pixel selection.

The Luma Keyer UI has four controls.

Use:	To:
Channel	Select the luma or the color channel luminance that will be used to pull a key.

Use:	To:
Key	Set the range of pixels to key based on the threshold range.
Threshold	Set the range of the pixels to key. Increasing the threshold removes less prevalent pixels from the composition.
Softness	Soften the edge of the key.

NOTE Key, Threshold, and Softness are animatable attributes—see Marking Attributes for Keyframing on page 674.

Removing Color Spill with Color Curves

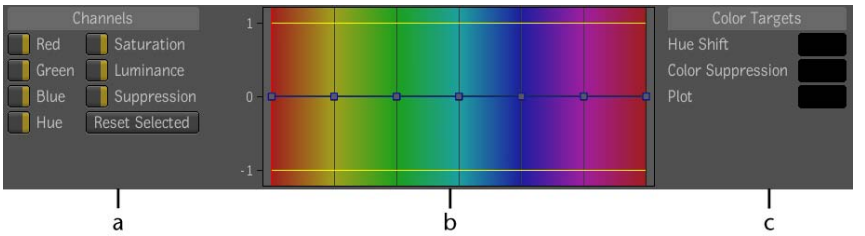
Color Curves lets you remove color spill by either suppressing a sampled color, by modifying the red, green, blue, or hue, by modifying the saturation or by modifying the luminance of the selected color, or by shifting the hue of a color range to a sampled hue shift target. Each curve in Color Curves is a hue gradient. When you change the shape of a curve, the colors along the curve's gradient change to reflect the result. For example, if your hue shift target is magenta, as you raise the green portion of the default hue shift curve, it gradually becomes magenta, becoming fully magenta when you reach the full value (75%).

Because the Color Curves affects the RGB and is applied to the output of a keyer tool, your process tree should include as a minimum, a source and a keyer.

While working on RGB, you can:

- Shift the hue of a color range to a sampled hue shift target.
- Suppress a sampled color.
- Modify R, G, B, saturation, and luma.

The Color Curves UI is divided into three areas.



(a) Channels (b) Curves window (c) Color Targets

Use:	To:
Channels controls	Select which color curve you want to manipulate.
Curves window	Adjust the curve by manipulating its control points and tangents. For more precise work, add and delete control points by using the + and - hotkeys when your cursor is in the Curves window.
Color Targets color pots	Plot the target color for a hue shift or a color suppression. Use the Plot color pot to reference any color in the image.

NOTE Color Target color pots are only used with the Hue and Suppression channels.

Shifting the Hue of a Color Range to a Sampled Hue Shift Target

You can use Hue Shift to compensate for colors in an image that are too hot or too cool, or to correct undesired tones present in the image.

Shift the Hue of a Color Range to a Sampled Hue Shift Target by using the Hue toggle button in the Channels area to activate the hue shift curve. Use the Hue Shift color pot in the Color Targets area to set a hue shift target color using the color picker.

With the hue shift target set, modify the shape of the hue shift curve along the hue range that you want to shift by selecting and manipulating the curve's control points and tangents—see Hotkeys on page 817.

Suppressing a Sampled Color

Unwanted color can be caused by factors such as inconsistent lighting conditions during a shoot. This can result in images that contain unnatural looking colors or one predominant color, which gives the image an unwanted color cast. You can suppress the unwanted color using the Suppression feature in the Color Curves.

Use the Suppress toggle button to activate the color suppression curve. Use the Color Suppression color pot next to the Suppress button to set a color suppression target using the color picker.

With the color suppression target set, modify the shape of the color suppression curve along the hue range that you want to suppress.

Modifying the R, G, B, Saturation, and Luma

To increase the probability of pulling a good key, you can modify the levels of the RGB channels, saturation and luma.

Toggle the Red, Green, Blue, Saturation, and Luma buttons to activate their corresponding curves. Manipulate the shape of each curve over the source hue range that you want to affect. The value of the parameter that corresponds with the curve changes relative to the height of the point along the curve in the curve editing area.

Each curve is a hue gradient and as the shape of the curve changes, the colors along the curve's gradient change to reflect the result. For example:

- Raising the luma curve over the green hue range, the green curve along that range becomes brighter—the luma value increases (RGB values increase together).
- Lowering the blue curve over the magenta hue range, the blue curve over that range becomes red—the blue (B channel only) value decreases.

Modifying Luma

While working on luma, you can modify the R, G, B, saturation, and luma along the luma range.

You can toggle the Red, Green, Blue, Saturation, and Luma buttons to activate their corresponding curves. Manipulate the shape of each curve over the luma range that you want to affect. The value of the parameter that corresponds

with the curve changes relative to the height of the point on the curve in the curve editing area.

Each curve is a hue gradient and as the shape of the curve changes, the colors along the curve's gradient change to reflect the result. For example:

- Raising the blue curve over the darker luma range, the curve along that range becomes bluer (for all pixels with a lower mean luma value, the blue channel value increases—the result in the image is that blue is added to the shadows)
- Lowering the saturation curve over the lighter luma range, the saturation curve over that range desaturates (for all pixels with a higher mean-luma value, the saturation value decreases—the result in the image is that highlights wash out).

Keyer Super Tool

The Keyer super tool provides you with an envelope to perform all keying tasks. The Keyer super tool also provides increased flexibility with the option of adding more tools (or removing existing ones) from the super tool schematic.

About the Keyer Super Tool

The Keyer is a super tool that combines a Screen Degrain, Master Keyer matte generator, Cleanup Alpha tool, Edge tool, Garbage Mask, and a Spill & Blend operator in one tool.

Using the Master Keyer matte generator, you can make parts of an image transparent by selecting and isolating regions of color. This process creates an alpha channel matte, which is then used to composite the image over a new background.

With the Master Keyer matte generator, you can refine the key by gesturally modifying the matte, removing color spill, blending edges, applying patches, and removing grain.

The following are examples of typical uses for the Keyer supertool:

- Pulling a rough matte
- Refining a matte

- Refining the edges of a matte
- Degraining a key in image
- Spill suppression
- Improving the blend between the front and back images.
- Compositing using a matte

Supported Features

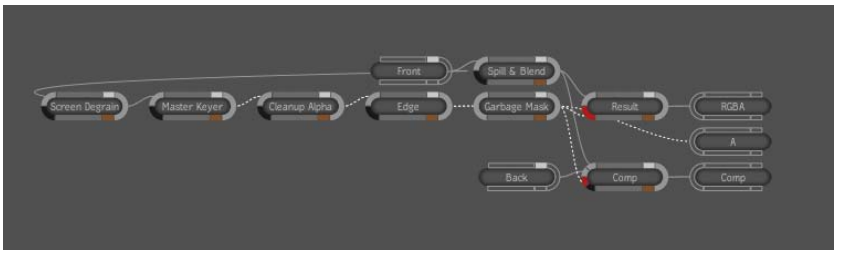
The Keyer supertool supports the following features and functionality:

- Quick selection of Keyer nodes
- Adding and changing tools in the Keyer schematic
- Master Keyer matte generator with on-screen manipulation
- Unlimited patches
- Screen Degrain
- Spill suppression and Blend
- Compositing of foreground and background via generated matte

Inputs and Outputs

The Keyer super tool has three image inputs and three outputs.

Double-click the Keyer super tool node, or right-click the node and select Edit Group to expand the Keyer.



The Front input image is despilled by the Spill & Blend tool node. The result is sent to the Result node (an instance of the Set Alpha tool) to become the RGB channels of the RGBA Output Image output.

The Front input image is processed first by the Screen Degrain tool node, then by the Master Keyer matte generator. The resulting matte is then sent to the Cleanup Alpha node, then to the Edge node, and that result to the Garbage Mask node. The output of the Garbage Mask node is sent to the input of the Result node to become the alpha channel of the RGBA Output Image output, to the Comp tool as the matte input, and is also outputted directly as the A Output Image.

The Back input image is sent to the Back input of the Comp tool node (an instance of the Blend & Comp tool). The Comp tool node provides a composite of the spill-suppressed front over the back, through the processed matte, both as a user convenience, and also as output on the Comp Output Image output.

Workflow

The following sequence of actions is a recommended workflow for keying an image with the Keyer supertool. You may not need to complete all the procedures. You may also revisit procedures as you develop the key.

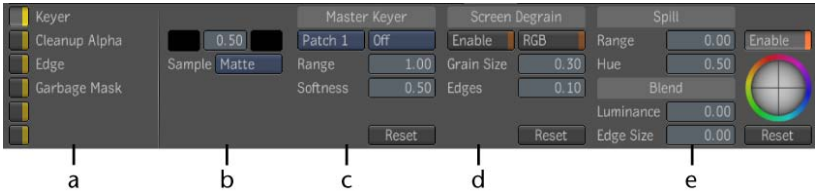
- Generate and refine a matte using the Master Keyer matte generator controls.
- Create any necessary patches to remove unwanted gray areas from the matte.
- Refine the matte using the Edge and Cleanup Alpha tools.
- If the image is grainy, apply the Screen Degrain algorithm.
- Remove any color spill.
- Improve the blend between the front and back images.

In the most common use case, the key color you sample will be the one used for screen degraining and spill and blend adjustments. To provide quick workflow, the Master Keyer, Screen Degrain and Spill & Blend initially created by the Keyer super-tool all share the primary and secondary key colors, and mix values of these samples.

If you don't like this key color sharing, the Master Keyer, Screen Degrain and Spill & Blend are all available as separate tools, and there you can set colors separately for all three.

Keyer Supertool UI

The Keyer super tool UI is composed of five areas:



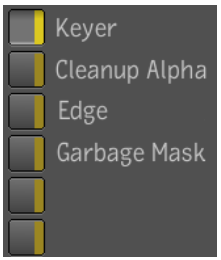
(a) Quick Selection of Keyer nodes (b) Master Keyer matte generator controls and Sample menu (c) Patch controls (d) Screen Degrain controls (e) Spill and Blend controls

To reset the Keyer tool:

➤ Click on the Reset button in the Tool Options area located on the far right of the Tool UI.

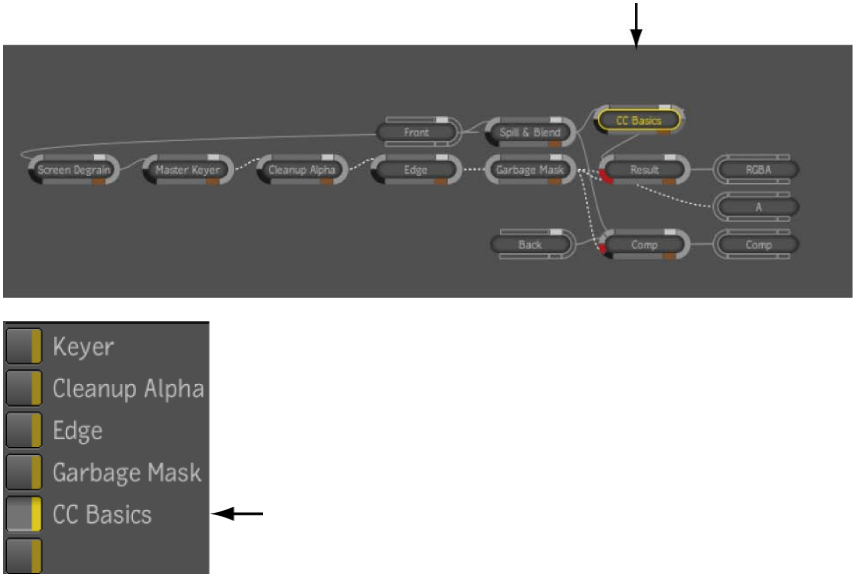
Node Select

A set of user-definable radio buttons is presented at the left of the Keyer UI.



The first four buttons are pre-defined to select the Keyer itself, the Clean Up Alpha, the Garbage Mask, or the Edge tool respectively. Selecting the node button will display the UI of the given tool, as well as set the display of the Animation editor, and the composition browser. All buttons except the first can be defined or redefined to select other nodes (and their UIs) in the Keyer schematic by clicking them with the Control key pressed. This will assign the button to the currently-selected node.

For example, if you selected the CC Basics node, you would then CTRL-click the node select button you wish to assign to it.

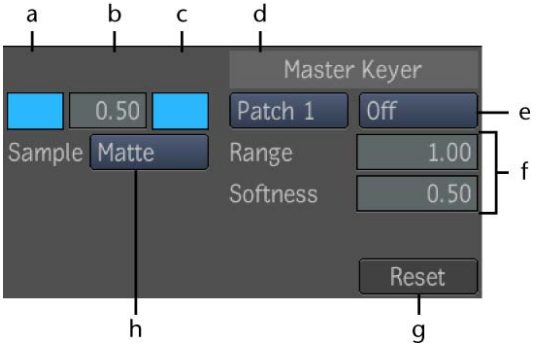


Generating a Matte with the Master Keyer

Using the Master Keyer controls, you can generate and refine a matte for your chroma key.

To generate and refine a matte with the Master Keyer:

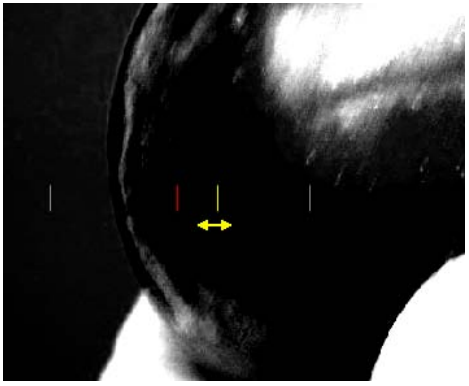
1. Set a Player view to Tool Output and, with the selection on the Keyer super tool node, cycle to the Comp Output of the Keyer—see Cycling Through Inputs and Outputs on page 71.
2. Select Matte from the Sample menu or press **M**.



(a) Primary sample color pot (b) Mix field (c) Secondary sample color pot (d) Patch list (e) Mode menu (f) Range/Softness sliders (g) Reset button (h) Sample menu

3. Click the Primary Sample color pot and sample the image. Click the image to sample a single pixel, or use **Ctrl** + drag to sample a larger area.
4. Click the Secondary Sample color pot and then sample the image. Sample an area where you do not want any softness in the matte.
5. Drag in the Mix field to adjust the mix between the primary and secondary sample. Drag right to include more of the secondary sample or left to include less.
6. Gesturally refine the matte. In the image window, click an area of the matte that you want to refine; only those parameters that pertain to the area you click appear. Parameters are displayed in order of importance, from top to bottom (those that are brightest and at the top have the greatest effect on the image). You can then modify a parameter by dragging its highlighted slider.

- To add softness, drag a slider to the right.
- To remove softness, drag a slider to the left. The red indicator shows the original value and the yellow indicator shows the current value.



7 To modify more than one parameter, move the mouse between the parameters until the cursor changes to a double arrow, or drag vertically. When you highlight the parameter you want to adjust, drag the slider.

8 When you are finished modifying the displayed parameters, click another area of the image without highlighting a parameter to hide them. Alternatively, you can press **Esc**. The parameters are no longer displayed.

9 Repeat steps 6 to 8 in other parts of the matte to further refine it. Only the parameters that apply to the problem area will appear.

10 To scroll through the image and display the pertinent parameters, use **Alt**+ drag the image without clicking it. The parameters update as you

drag. You can then modify the displayed parameters by clicking the image and dragging the highlighted slider.

NOTE To reset Matte parameters, click the Reset button under the Master Keyer UI. All matte parameters are reset, except the Mix field and the key colors.

Removing Unwanted Grays Using Patches

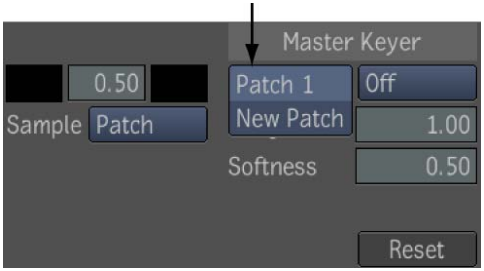
If you have unwanted gray areas in the matte, you can use patches to isolate a range of colors to be included in, or excluded from, the key.

NOTE Patch parameters are animatable.

To remove unwanted grays using patches:

1. Set a Player view to Tool Output and, with the selection on Keyer super tool node, cycle to the alpha output—see Cycling Through Inputs and Outputs on page 71.
2. Select Patch from the Sample menu or press **Shift** + the number associated with the patch. For example, **Shift + 1** for Patch 1.

3. Select Patch 1 or create a new patch from the Patch menu.



NOTE You can create an arbitrary number of matte generator patches. The number of patches starts at one, and new patches can be created. Although, in theory, there is no limit to the number of patches, in practice, there are diminishing returns in terms of matte quality when adding new patches.

4 Sample the image where you want to apply the patch. To sample a single pixel, click the image. To sample an area of the image, **Ctrl** + drag a selection box.

When you **Ctrl** + drag to sample, the Patch button is selected and the appropriate patch mode appears in the Patch box and is applied to the image.

Selected Patch:	Is applied to:
Black	Areas of the image to be included in the black part of the matte.
White	Areas of the image to be included in the white part of the matte.
Analysis	Areas of the image that are along the edge of the key. Edge Analysis is useful when there is a specific edge you want to erode but cannot do so with the Matte parameters. You can then increase or decrease the softness of this patch using the Softness slider.

5 To add more color to the patch, resample the image.

6 To use the same patch but start with a new sample, press **Ctrl** + **Alt** + drag the image.

7 To manually select a patch type, select it from the Patch box.

NOTE If you want to reset the patch, so you can automatically select the patch type, you must disable the patch and set the patch type to Off. You can then resample an area in the image and generate a patch type.

8 To improve the patch, use the Patch controls.

Drag:	To:
Range	Increase or decrease the color range that is included in the patch.
Softness	Soften the edge.

NOTE To reset Patch parameters, click the Reset button under the Master Keyer UI.

Degraining the Key In Image

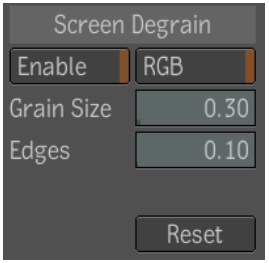
Graininess can make it difficult to pull a clean and effective key. With the Screen Degrain controls you can remove film grain, modify the size of the grain, and desharpen the edges of the key. The front image is not affected when you apply Screen Degrain to the Front input image.

NOTE Screen Degrain parameters are animatable.

To remove grain from the key in image using Screen Degrain:

1 Set a Player view to Tool Output and, with the selection on the Keyer super tool node, cycle to the A or Comp Output—see Cycling Through Inputs and Outputs on page 71.

2 Select Degrain from the Sample menu, or press **D**.



3 Sample a grainy area of the image. To sample a single pixel, click the image. To sample an area of the image, **Ctrl** + drag a selection box. Once the image is sampled, Screen Degrain is automatically enabled and grain is removed from the image.

4 Drag in the Screen Degrain fields to modify the grain size and restore edge sharpness.

Use:	To:
Enable	Enable the degrain algorithm.
RGB	Enable RGB to degrain on all channels. The degrain algorithm is optimized for blue screen degraining.
Grain Size	Estimate the size of the grain in the image.
Edges	Unsharpen the edge of the image. By default, Degrain sharpens the edges. Use the Edges field to restore the natural look of the edges in the image.

5 If you are not satisfied with the result, you can start over with a new sample, and then adjust the Screen Degrain fields. To resample an area of the image, **Ctrl** + **Alt** + drag the image.

NOTE You can also reset Screen Degrain parameters. To reset Screen Degrain parameters, click the Reset button under the Screen Degrain UI.

6 If resampling the image and adjusting the Screen Degrain fields does not sufficiently remove grain, select RGB to increase the overall effect of Screen Degrain.

Suppressing Spill

After you create a key and key out any trouble areas, some of the background color may have spilled over the edge of the key. Use the Spill controls to suppress or disguise color spill in the front image.

To gesturally remove color spill:

1. Set a Player view to Tool Output and, with the selection on the Keyer super tool node, cycle to Comp Output—see Cycling Through Inputs and Outputs on page 71.
2. Select Spill from the Sampling box or press **S**.
3. Remove any color spill. Click anywhere in the Player, and modify the parameters that appear by dragging the sliders; you do not need to click a specific area.

Drag:	To:
Range	Set the range for the removal of color spill along the edges of the key. Drag right to soften the edge and remove color spill further into the key. Drag left to harden, or create a thinner, edge.
Hue	Modify and suppress colors that are adjacent to the primary sample.

NOTE To reset Spill and Blend parameters, click the Reset button, below the Edge Balance trackball.

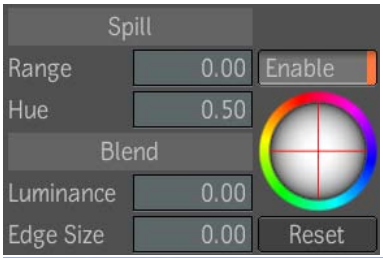
Modifying Blend

Using the Blend parameters you can gesturally modify the luminance at the edge of the key, so that it blends with the luminance in the background image. For example, when the front image is darker than the back image, you can use the Blend parameters to lighten the edge of the key.

With the Blend Luminance controls, you can adjust the edge blending and maintain the edge detail. Adjusting the Edge Size amount uses pixel analysis to determine the extent of the adjustment.

To blend the front and back images:

1. Set a Player view to Tool Output and, with the selection on the Keyer super tool node, cycle to Comp Output—see Cycling Through Inputs and Outputs on page 71.
2. Select Blend from the Sample menu or press **Shift** + **B**.
3. Blend the front and back image. Click anywhere in the image window, and modify the parameters that appear by dragging the sliders; you do not need to click a specific area.



Drag:	To:
Luminance	Darken or lighten the edge of the key. Luminance only affects the luma of the edge.
Edge Size	Set the range for the blend. Drag right to soften the edge and blend further into the key. Drag left to harden, or create a thinner edge.

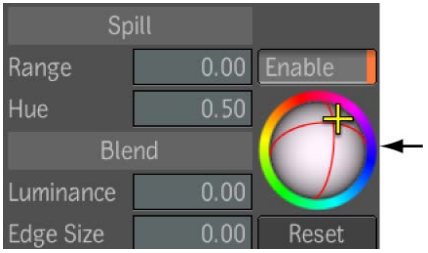
4 To modify both parameters, move the mouse between them to highlight a parameter, or drag the pen vertically. When you highlight the parameter you want to adjust, drag the slider.

NOTE If you do not like the result, you can click Undo to reset parameters after you complete an operation.

5 When you are finished modifying the displayed parameters, click another area of the image without highlighting a parameter to hide them or press the **Esc** key.

6 To add a cast to the edge of the key and improve the overall look by matching the edge with a color cast in the back image, drag the Edge Balance trackball toward the color you want to add. The trackball only affects the chroma of the edge.

Modifications made using the trackball are cumulative; each movement of the trackball is added to the previous one.



NOTE To reset Spill and Blend parameters, click the Reset button, below the Edge Balance trackball.

Masking

23

Topics in this chapter:

About Masking

Masking is the process of hiding a region of an image. You can use masks to remove the area outside the mask shape and keep the area inside the mask shape, or you can use masks to remove the area inside the mask shape and keep the area outside the mask shape.

Pixel Masking lets you temporarily isolate specific areas of the footage. You can apply effects to the selected area of an image without affecting the rest.

Masking Concepts

You can use garbage masks to key undesired elements in an image and to do rotoscope work. A garbage mask affects the image's alpha channel, which is where the transparency information is stored. You can use a mask to create the alpha channel, or you can use a mask to add to an alpha channel.

Garbage matte A matte that isolates unwanted elements from the primary element in an image. Garbage mattes are also referred to as garbage masks.

Auto Tangent When selected, the mask is created with tangents.

Intensity Defines the alpha value defined by the mask.

Opacity Defines the transparency of the mask.

Invert The area outside of the mask is covered by the garbage mask settings.

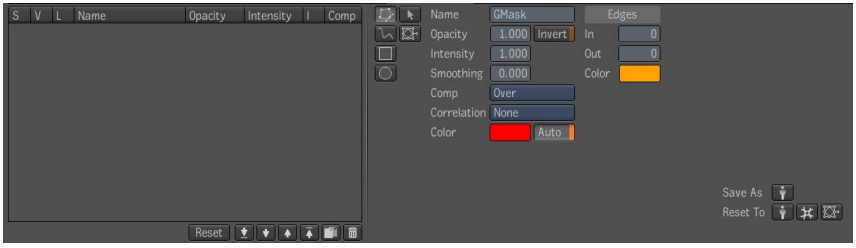
Edge Distance The inner or outer offset from the edge of the mask.

Edge Position The position of the edge handle from the first control point of the mask.

Edge In/Out The inner/outer offsets from the edge of the mask.

Garbage Mask Tool Tabs

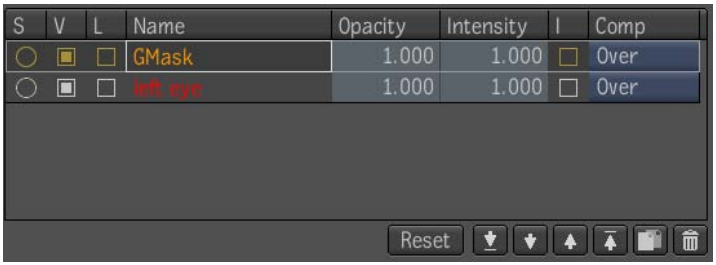
The Garbage Mask is a three-tabbed tool consisting of Garbage mask controls—see Creating Masks on page 540, Global Composite Output controls—see Output Tab Settings on page 558, and the Options tab—see Options Tab Settings on page 559.



Garbage Mask Controls

The Garbage Mask tab consists of the following controls and parameter settings:

Mask List The Mask List displays all the masks that have been created for the current composition. It also displays some of the masking parameters and navigation buttons.



Use:	To:
S	Activate the Solo flag to isolate a mask. Only one mask can be isolated at a time.
V	Activate the Visible flag.
L	Activate the Lock flag. Locking a mask locks the points, shape, edges, tangents, and composite parameters.
Name	Change the name of a mask. With the cursor in the text field, press **F2**, type in a new name and press **Enter**.
Opacity	Set the opacity of the mask.
Intensity	Set the intensity of the mask.
I	Invert the mask.
Comp	Select a compositing operator. The default is the Over mode—see Compositing Operators on page 563.

Use:	To:
Reset	Reset the Comp operator, Opacity, Intensity and the mask Invert toggle of selected masks.
	Move selected masks to the bottom of the Mask list.
	Move selected masks down one place in the Mask list.
	Move selected masks up one place in the Mask list.
	Move masks to the top of the Mask list.
	Duplicate selected masks. The duplicate appears over the selected mask in the Player.
	Delete selected masks.

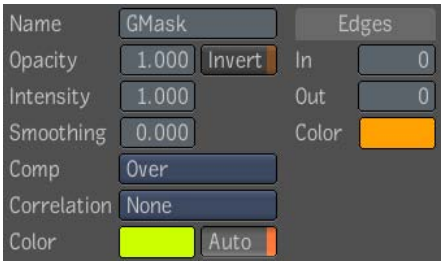
Creation Tools Use the following Creation tools to draw garbage masks.

Use:	To:
	Create freeform Bézier spline shapes.
	Create freehand shape.
	Create rectangular shapes. These shapes can be a square, fixed size, proportional, or unconstrained.
	Create elliptical shapes. These shapes can be a circle, fixed size, proportional, or unconstrained.

Creating Masks

When you select a Creation tool, default values for future masks are made available for modification. These default values are initially set to the previously saved values. You can reset default values in each Garbage Mask tab by selecting

the Factory Default button—see User Settings on page 555. You can change the following parameters once a Creation tool is selected:



- Garbage mask Name
- Opacity (animatable)
- Intensity (animatable)
- Smoothing
- Invert (animatable)
- Compositing Operator—see Compositing Operators on page 563.
- Correlation. Specify how input mattes are correlated. This can be used to improve the quality of the composite in special cases. For example, if you composite two mattes that share a good portion of their outline, you should indicate if they are Adjacent or Superposed. By default, the correlation mode is None, assuming that normally, the input mattes are not correlated.
- Mask Color and Auto Mask Color toggle
- Initial In/Out Edge distance
- Edge Color

To create a shape using the Freeform tool:

1. Select the Freeform tool button or press **D**.
2. Set initial mask parameters if necessary.
3. Click and drag on the image in the Player to place the first control point and adjust the tangent (if not in Auto Tangent), or (if in Auto Tangent) to move the point.
4. Move the cursor to the location where you want the next control point to be and click.

5 Continue to click in the image to add more control points.

6 To close the shape, click on the first control point you created, or press **Enter**. If you hold **Ctrl + Alt**, the shape will be closed in Auto Tangent mode.

Once the first control point has been created using the Freeform tool, the garbage mask is automatically set to edit mode.

NOTE If you are not in Auto Tangent mode and want to create several points in Auto Tangent, hold **Ctrl + Alt** when creating the points.

To create a rectangular mask:

1 Select the Rectangle tool button or press **S**.

2 Set initial mask parameters if necessary.

3 Select Rectangle Options settings if necessary. These options include:

Use:	To:
From Center	Click and drag the cursor to draw the mask from the center of the first mouse/pen click.
Unconstrained	Create an unconstrained mask. Click and drag to draw and change the shape of the mask.
Square	Click and drag to draw a perfect square.
Fixed Size	Create a rectangular mask based on width and height values you set. Once the values are set, click the location in the image you want the shape to appear.
Proportional	Create a rectangular mask based on width and height proportions you set. Once the values are set, click and drag to create the shape.

To create an elliptical mask:

1. Select the Elliptical tool button or press **W**.
2. Set initial mask parameters if necessary.
3. Select Ellipse Options settings if necessary. These options include:

Use:	To:
From Center	Click and drag the cursor to draw the mask from the center of the first mouse/pen click.
Unconstrained	Create an unconstrained mask. Click and drag to draw and change the shape of the mask.
Square	Click and drag to draw a perfect circle.
Fixed Size	Create an elliptical mask based on width and height values you set. Once the values are set, click the location in the image you want the shape to appear.
Proportional	Create an elliptical mask based on width and height proportions you set. Once the values are set, click and drag to create the shape.

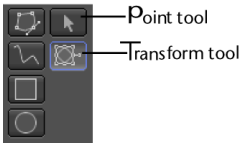
Editing Masks

Once you create a mask, you can use different tools to modify it. As the portion of the image that you are masking changes over time, you will need to adjust the points of the mask to exactly match the image. The mask editing options can be applied to a single point, a group of points, a single shape, or a group of shapes by:

- Selecting a point or group of points in the Player.

- Selecting a shape or group of shapes from the mask list.
- Selecting a shape or group of shapes in the Player.

There are two mask editing tools in the Garbage Mask UI; the Point tool and the Transform tool.



Use the Point tool to edit mask points. Depending on what hot key is pressed and on the location of the mouse cursor, the mouse cursor updates and displays a visual cue for the current edit mode.

You can nudge the position of selected mask control points, selected shapes, and edge handles (in increments of one pixel) by holding down the **Shift** key and pressing the arrow keys on the keyboard. To nudge the position of control points, selected shapes, and edge handles in increments of 10 pixels, hold **Ctrl** + **Shift** and press the arrow keys on the keyboard.

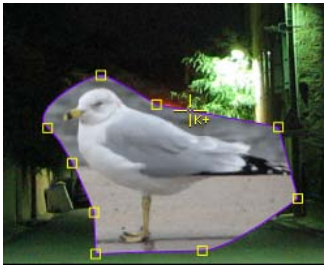
The following mask point parameters (for each point) can be animated. They are accessible from the animation timeline and expressions can be assigned to all of them:

- X
- Y
- Tangent In angle
- Tangent In Distance
- Tangent Out angle
- Tangent Out Distance

NOTE Tracking information can be assigned to point position—see Tracking and Stabilizing on page 753.

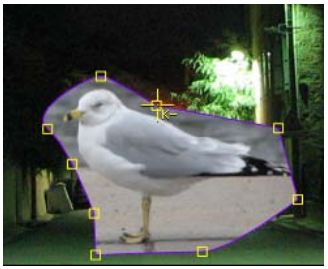
To insert a control point:

➤ Press and hold the **Alt** key and click on the curve dragging the point to move it.



To delete control points from a mask:

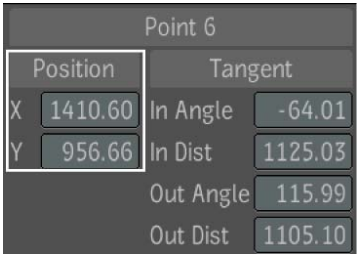
➤ Press and hold the **Alt** key and click on the control point you want to delete or, select points and press the **Delete** key.



To move control points:

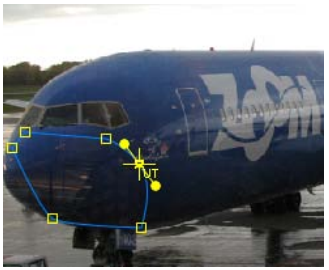
➤ Do one of the following:

- Click and drag a control point. Hold **Ctrl** + **Alt** while moving the point to move in Auto Tangent mode.
- Click or drag a bounding box around a point to select the point. Type the position values into the X and Y value editor under the Position label and press Enter.
- Click or drag a bounding box around a point to select the point. Click + drag inside the value editor.



To add or delete tangent handles:

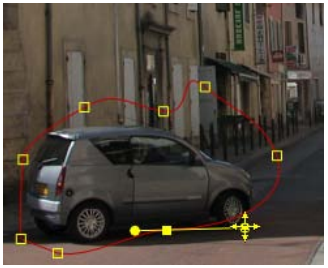
➤ Press and hold the **Ctrl** key and click on the control points to go from a sharp curve to a smooth curve with tangent handles and drag to adjust the tangents. Press and hold the **Ctrl** key and click on the control point a second time to delete them.

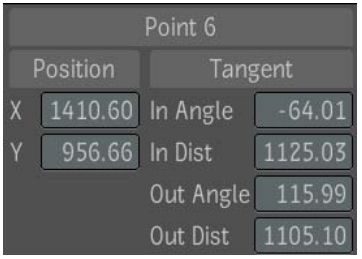


To adjust the length of a single unbroken tangent handle:

➤ Do one of the following:

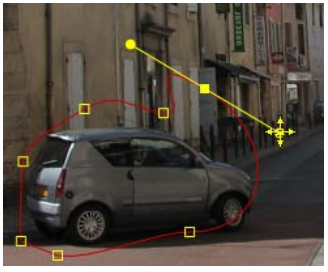
- Click and drag the handle.
- Type the distance value into the In Dist or Out Dist value editor under the Tangent label and press **Enter**.
- Click + drag inside the value editor.





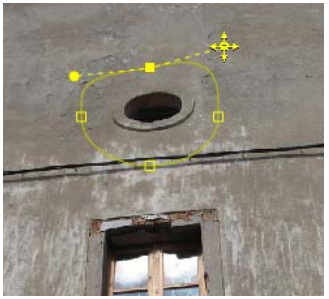
To adjust the length of both sides of an unbroken tangent handle:

➤ Hold the **Alt** key and drag the handle. When you press **Alt** and drag an unbroken tangent handle, both tangent handles are automatically given the same length.



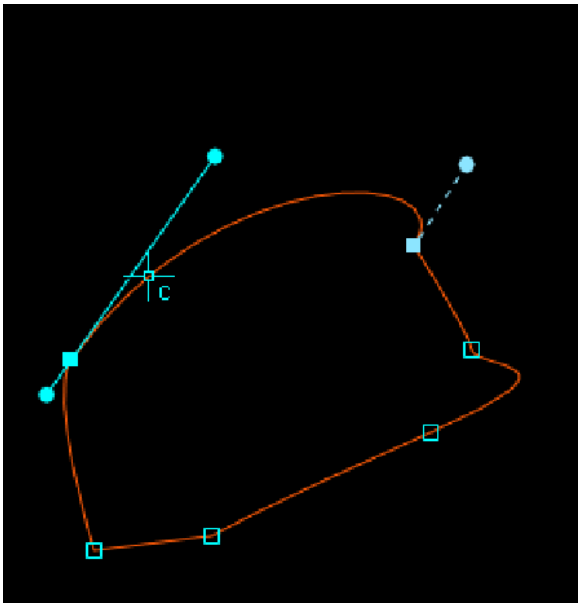
To break a tangent handle:

➤ Press **Ctrl** then click and drag the handle. Press **Ctrl** then click and drag the handle again to make the handle continuous.



To move a curve between points:

➤ Hold **Ctrl + Alt** over the location on the curve you want to move then drag the curve. The curve and tangents adjust accordingly based on the location of the cursor.



Transforming Masks

Once your mask is created, use the Transform tool to fine tune its position, shape, and size. You can manipulate the mask inside the player. These settings

can also be animated. Once you select the Translate tool, the following parameters are displayed:

The following mask shape Transform parameters can be animated. They are accessible from the animation timeline and expressions can be assigned to all of them:

- Translation X
- Translation Y
- Rotation R
- Scale X
- Scale Y
- Pivot X
- Pivot Y
- Pivot R

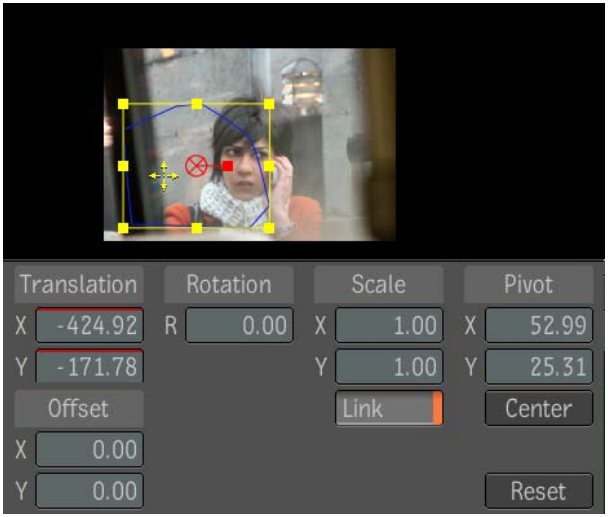
NOTE Tracking information can be assigned to shape transformation—see Tracking and Stabilizing on page 753.

Use:	To:
Translation X,Y	Translate the mask in the X and Y direction.
Offset X,Y	Add an offset to the translation of a garbage mask.
Rotation	Rotate the mask.
Scale	Change the mask's X or Y scale (or both with Link enabled).
Pivot	Adjust the X and Y location of the mask's center of rotation/scaling. Use the Center button to set the pivot point to the center of the mask's bounding box.
Reset	Reset all the transform parameters to their initial values.

To translate the mask:

➤ Do one of the following:

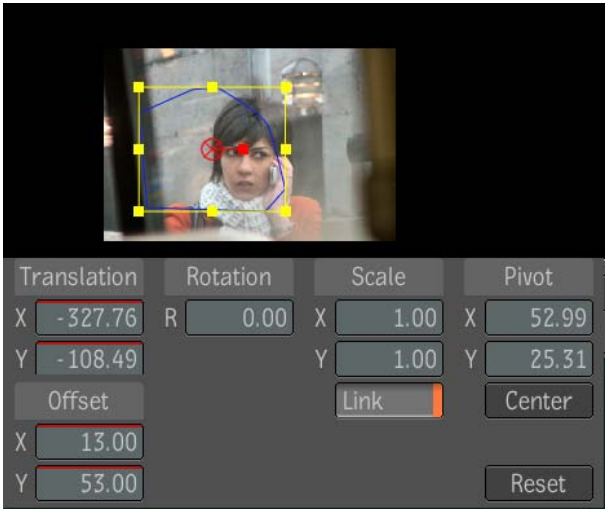
- Place the cursor inside the mask's bounding box, and click + hold and then drag the mask to translate in any direction.
- Type the position into the X, Y value editors and press **Enter**.
- Click + drag inside the value editor.



To add an offset to the translation of a mask:

➤ Do one of the following:

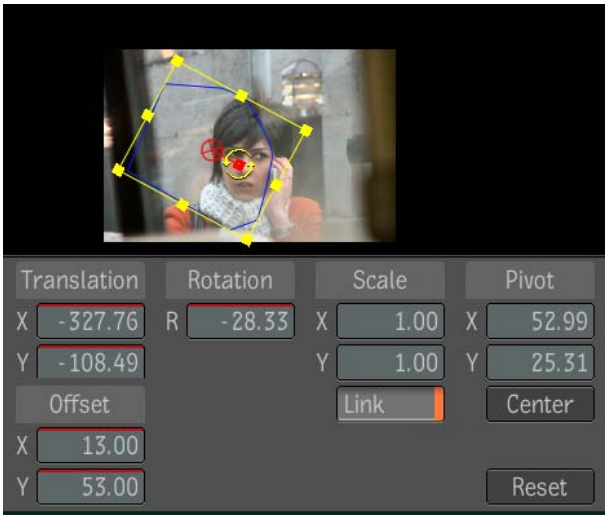
- Type the offset position into the X, Y value editors and press **Enter**.
- Click + drag inside the value editor.



To rotate a mask:

➤ Do one of the following:

- With the cursor over the rotation handle, rotate the mask.
- Type the angle of rotation (in degrees) into the Rotation value editor and press **Enter**.
- Click + drag inside the value editor.



To scale a mask:

➤ Do one of the following:

- With the cursor over the one of the mask's resize handles, click + drag the mask.
- Type the X and Y scale values into the X and Y Scale value editors and press **Enter**.
- Click + drag inside the value editor.

To adjust the X and Y location of the mask's center of rotation (pivot point):

➤ Do one of the following:

- With the cursor over the mask's pivot handle, click + drag the mask.
- Type the X and Y pivot position values into the X and Y Pivot value editors and press **Enter**.

- Click + drag inside the value editor.

Creating and Editing Edges

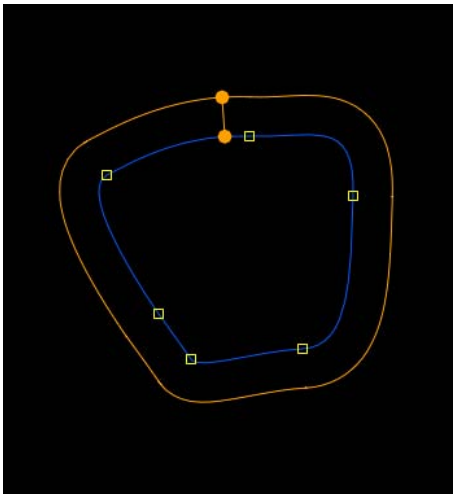
If you have not set initial edge gradient parameters before creating a mask shape. You can do so while in edit mode.

The following mask In and Out Edge point parameters (for each edge point) can be animated. They are accessible from the animation timeline and expressions can be assigned to all of them:

- Position
- Distance

To create an edge point:

➤ Hold down **Ctrl** and click on the mask curve and drag the point. An edge point with two handles is created.



To change the edge distance or position:

1 Do one of the following:

- Drag the outer edge handle towards the outside of the mask to create an outer edge gradient, or towards the center of the mask to create an inner edge gradient.
- Type the distance (in pixels) into the Distance value editor and position (relative to the first control point generated for the mask) then press **Enter**.
- Click + drag inside the value editor.

NOTE You can also multi-select several edge handles using the **Shift** key and change the distance or position of more than one edge by the same value.

2 Press **Alt** to lock the position of the edge while changing its distance.

To change the color of an edge handle:

1 Select any shape.

2 Click the color pot under Edge Settings.

3 Select a new color from the Color Picker UI and click Set.

NOTE This operation changes the color of every edge handle in the mask.

To set the edge handle visibility to not visible:

➤ Click the Visible (default) button under Edge Settings.

User Settings

Each tab has a set of User Setting buttons you can use to store initial mask parameters that can be reused during the rotoscoping process. By default, the first time you create a mask without changing any initial settings, the mask will have the following default settings.

Any subsequent mask you create will inherit the same settings except for the mask color (which by default is on Auto and picks a random color), and the mask name instance (the next mask would be named GMask(2)).

If you change the parameters for a second mask, for example, its name, color and toggle off the Auto Color, all masks you create afterward will retain those parameters.

If you need to create masks with a particular set of parameters that you will need from time to time during your rotoscoping process, use the User Settings buttons.

Use:	To:
Save As	Save initial mask parameters as a user favorite.
Reset To	Reset mask parameters to a previously saved user favorite.
Reset To	Reset mask parameters to factory default settings.
Reset To	Reset mask parameters to those of a selected mask in the Mask List.

Example: User Settings Workflow

The following example procedure demonstrates the User Settings workflow where an artist needs to create the following series of masks:

- Default settings freeform mask.
- Red ellipse of fixed size with Auto Color turned off and saved as a user favorite.
- Green rectangular mask drawn from center.
- Freeform mask reset to default (second instance).
- Second instance of the red ellipse (reset to user favorite).

- New rectangular mask (second instance) reset to the green square selected from the Mask List.

To create a series of garbage masks working with User Settings:

1. Select the Freeform creation tool

   (or press **D**) and draw a freeform mask using default values.

2. Select the Ellipse tool.

   (or press **W**).

3. Change the name to Red and press **Enter**.
4. Select a red color and turn Auto Color off.
5. Set the fixed size dimensions.
6. Select Save As User Favorite.

7. Click once in the Player to create the mask.
8. Select the Rectangle tool.

   (or press **S**).

9. Change the name to Green and press **Enter**.
10. Change the color to a green.
11. Toggle From Center
12. Click and drag in the image to create the mask.
13. Select the Freeform tool

    (or press **D**).

14. Select Reset To Default.

15 Create the mask in the image.

16 Select the Ellipse tool.

(or press **W**).

17 Select Reset To User Favorite.

18 Click once in the Player to create the mask.

19 Select the Mask named Green from the Mask List.

20 Select the Rectangle tool

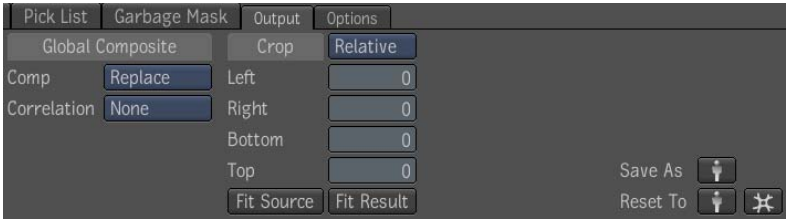
(or press **S**).

21 Select Reset To Selected.

22 Click and drag in the image to create the mask.

Output Tab Settings

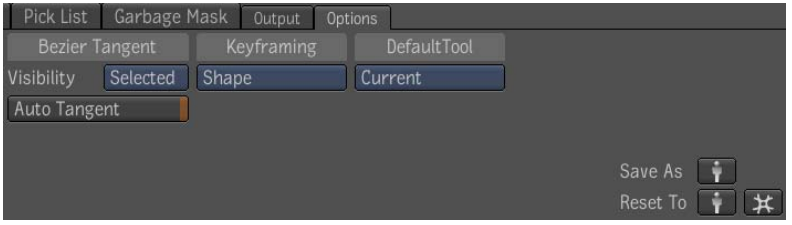
Use the Global Composite settings to specify how all masks, combined together using their individual modes, affect the alpha channel for the selected input source image.



Select:	To:
Comp	Choose which compositing mode will be used —see Global Composite Operators on page 564.
Correlation	Specify how input mattes are correlated. This can be used to improve the quality of the composite in special cases. For example, if you composite two mattes that share a good portion of their outline, you should indicate if they are Adjacent or Superposed. By default, the correlation mode is None, assuming that normally, the input mattes are not correlated.
Crop	Change the size of an image—see Crop Tool on page 410.
User Settings	Store initial mask parameters that can be reused during the rotoscoping process—see User Settings on page 555.

Options Tab Settings

Use the Options tab to set default settings for creating and editing masks.

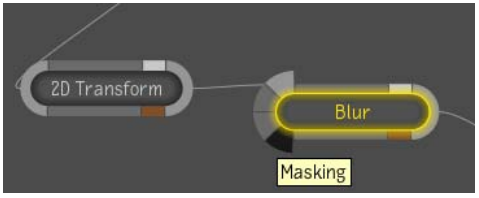


Use:	To:
Bezier Tangent Visibility	Set the visibility mode of Bezier tangents. Choose from only those tangents Selected, All, or None
Auto Tangent	Toggle the Bezier tangent creation mode from Off to Auto Tangent.

Use:	To:
Keyframing	Set how mask points will be keyframed. Choose from Selection, Point, or Shape. If a mask control point is moved (animated) the following attributes will be keyframed based on the Keyframing mode. ■ **Shape (default)** X, Y positions, Tangent Angles In and Out, and Tangent Lengths In and Out for all points of the shape. ■ **Point** X, Y position and its Tangent Angle In and Out, and Tangent Length In and Out ■ **Selection** X and Y position.
Default Tool	Set how the tool behaves once a shape is closed. When a shape is closed and Create has been selected from the list, the tool will revert back to the Creation tool (Freeform, Rectangle, or Ellipse) that created the shape. If Edit is selected, the tool will revert to edit mode. If Current is selected, the tool will revert back to the current setting.
Options tab User Settings	Store initial mask parameters that can be reused during the rotoscoping process—see User Settings on page 555.

Pixel Masking

Pixel Masking lets you limit the effect of an image modifier to a subset of the input pixels (those covered by the mask). All tools that support masking have a Mask image input.



The actual masking is implemented by blending the original input with the tool output using the pixel values in the mask as the control weights in the blend.

Masking Parameters

The masking parameters are in a separate tab in the Tool UI. This tab is immediately to the right of the tool tab the mask will originate from.

Select:	To:
Create	Create an instance of a masking tool selected from the pull-down list. The tool is created in the current composition and its output is connected to the Mask input of the current tools. The list is editable in the user settings tab under Masking Menu Tools.
Edit	Return to the tool UI to change its parameters.
Active	Determine whether masking is enabled or disabled. By default, masking is on. **NOTE** If no image is connected to the Mask input, the value of Active has no effect. That is, no masking is applied in any case.
Invert	Invert the mask prior to using it in the blend operation. By default, Invert is off.
Channel	Select one of R, G, B, A. This is the channel of the Mask image to be used for masking. By default, Channel is A.
Intensity	Scale the mask (after optional inversion) prior to using it in the blend operation. By default, Intensity is 1 (full intensity).

The list of tools in the masking tool menu contains all plug-in tools that describe they can act as masks. The most recently used masking tool becomes the default selection in the menu, for all maskable tools. This most recently used masking tool is stored as a user setting.

When you add a masking tool from the masking tool input menu, it is added to the dependency graph and its output is connected to the masked tool's

input. If there is already a tool connected to the masking input, there are two behaviors:

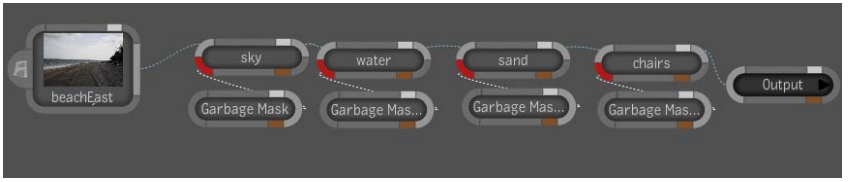
- If the new masking tool has an image input, the current masking tool output is disconnected from the masked tool, and connected to the new masking tool's primary image input.
- If the new masking tool does not have an image input, the current masking tool output is simply disconnected from the masked tool.

NOTE A masking tool with no image input, such as a mask generator, takes its output format and resolution from the composition settings.

The Active, Invert, Channel, and Intensity controls in the masking tab correspond to the Masking structured input. The Image text field shows the name of the node connected to the masking input (if any).

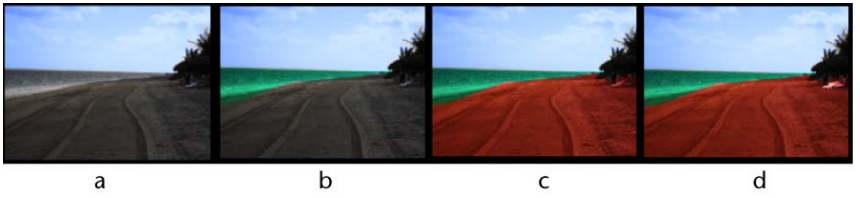
Pixel Masking Example

The following example illustrates how you can use several masking inputs to perform simple color corrections (using CC Basics) on different parts of an image.



By using the masking inputs, four color corrections (sky, water, sand, and chairs) were quickly applied.

(a) Original Image



(a) Sky corrected (b) Water corrected (c) Sand corrected (d) Chairs corrected

Compositing Operators

The following table lists the compositing operators applicable between generated masks and previously composited masks:

Operator	Result:
Over	Composites the generated mask (front) over the previously composited masks (back). The output will cover any area covered by either the front or the back. Where the front and back overlap, the output will show the front.
Replace	Completely replaces the previously composited masks with the generated mask.
Atop	Similar to the Over operator, but the output will cover the same area covered by the previously composited masks.
Inside	Composites the generated mask (front) over the previously composited masks (back), but the output will cover only the area covered by both front and back.

Operator	Result:
Outside	The output will be equal to the generated mask (front) except that the part of the front overlapping the previously composited masks (back) will be missing. No part of the back will be visible in the output in any case.
Cutout	The output will be equal to the previously composited mask (back) except that the part of the back covered by the generated masks (front) will be missing. No part of the front will be visible in the output in any case. Basically this operator is equal to the Outside operator with the front and back roles reversed.
Xor	Both generated mask (front) and previously composited masks (back) are copied to the output except for the part where they overlap.

Global Composite Operators

The following table lists the compositing operators applicable between the input image and previously composited masks:

Operator	Result:
Over	Composites the previously composited masks (front) over the input image (back). The output will cover any area covered by either the front or the back. Where the front and back overlap, the output will show the front.
Replace	Completely replaces the input image alpha channel with the previously composited masks.
Atop	Similar to the Over operator, but the output will cover the same area covered by the input image alpha channel.
Inside	Composites the previously composited masks (front) over the input image alpha channel (back), but the output will cover only the area covered by both front and back.
Outside	The output will be equal to the previously composited masks (front) except that the part of the front overlapping the input image alpha channel (back) will be missing. No part of the back will be visible in the output in any case.
Cutout	The output will be equal to the input image alpha channel (back) except that the part of the back covered by the previously composited masks (front) will be missing. No part of the front will be visible in the output in

Operator	Result:
	any case. Basically this operator is equal to the Outside operator with the front and back roles reversed.
Xor	Both previously composited masks (front) and input image alpha channel (back) are copied to the output except for the part where they overlap.

Raster Paint

24

Topics in this chapter:

About Raster Paint

The raster paint module in Composite is represented by a supertool node. The Paint node lets you work at any resolution, with any number of layers, and bit depths.

The paint node contains one or many layers of raster images and the compositing operators combine them to produce a single output image.

All operations, including painting, can be carried out while viewing the whole composited layer stack. Painting can be applied to any layer in the stack and

the composite view is constantly maintained. Individual layers can also be muted selectively.

Committing and Undoing Operations

You can undo multiple paint strokes in layer-based Paint. In the context of undo, a stroke is whatever you paint on the canvas until you commit.

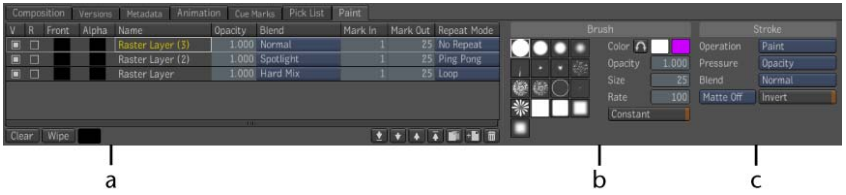
Do any of the following to commit paint strokes to the undo stack:

- Press any key.
- Move the cursor out of the current viewport.

For example, if you use a 20-pixel brush to apply several strokes to the canvas and then click **Ctrl + Z** (for Windows and Linux) or **Cmd + Z** (for Mac OS), all of the strokes created with this brush are erased. On the other hand, if you press **Spacebar** between each stroke and then click **Ctrl + Z** (for Windows and Linux) or **Cmd + Z** (for Mac OS), the strokes will be erased one at a time.

Paint UI

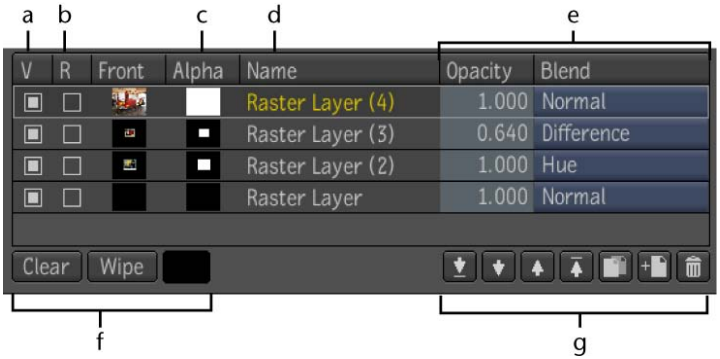
The Paint tool UI consists of three areas:



(a) Layer editor (b) Paint brush controls (c) Stroke controls

Layer Editor

The Layer Editor lets you create layers and reorder them. For each layer, you can apply a blending mode and set the opacity to define how it is blended with the layer below it. You can also mute layers or use them in reveal operations. The current layer indicates which layer is being drawn into. From the current layer, you can also select either the front or the alpha as the target.



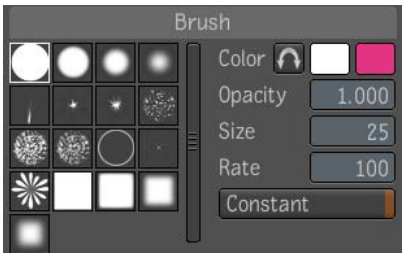
(a) View checkbox (b) Reveal checkbox (c) Alpha (d) Name (e) Opacity/Blend controls (f) Canvas controls (g) Layer controls

Use:	To:
V (view)	Mute or view a layer.
R (reveal)	Conduct reveal operations with this layer.
Alpha	Select the alpha channel to paint on.
Name	Use Name to give the layer a name.
Opacity	Set the layer opacity.
Blend Mode	Set the Blend mode. Click the Blend button select a mode—see Blend Modes on page 345.
Layer controls	Navigate through raster layers, and copy, add, and delete layers.
Layer Canvas controls	Clear the last committed stroke from the canvas or wipe a new color onto the current canvas. To select a color for the wipe operation, click the color pot, select a color from the color picker, and click Set.

Paint Brush Controls and Attributes

The brushes define the look of the paint strokes. You can customize brushes to define the look of a stroke.

The paint brush controls have the following parameters:

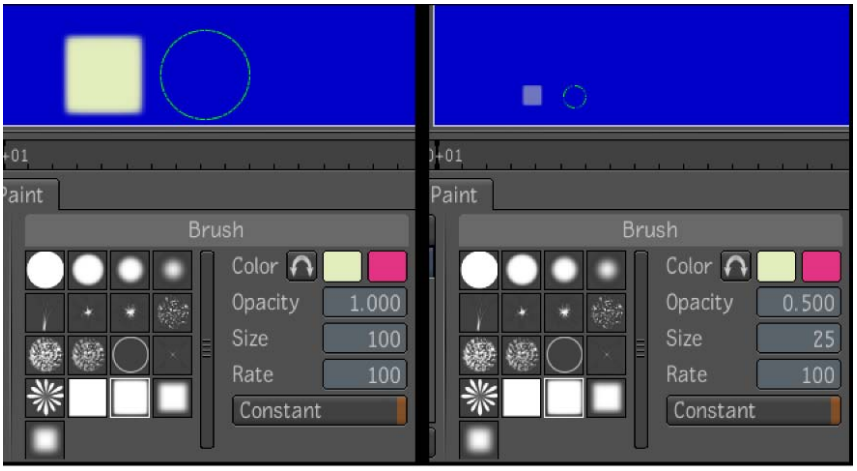


- **Stamping Patterns** The brush stamping pattern is a bitmap image that is used as a pattern to create a stroke. They can be resized from 1 pixel to 512x512 pixels. Composite comes with a set of predefined brushes and they are stored as an image file in the Composite installation directory under \Resources\Brushes. You can create your own brushes using third-party software and add them to the stamping pattern list by putting them in the \Brush folder. Brushes can be any pixel size, but every brush will be reformatted to 512 x 512. For best results, it is best to create your stamp patterns as 512 x 512 grayscale.tiff images, but any file format and size supported by Composite will be accepted.

- **Color** Define the color of the paint stroke. Click a color pot and select a color from the color picker. You can store two colors and use the toggle button to reverse them or press **X**.

- **Opacity** Set the opacity (transparency) of the brush. A value of 100% applies a fully opaque color. Use a lower value to apply a more transparent color.

- **Size** Set the brush size. The brush size is indicated by the diameter of the white circle surrounding the brush. To increase the brush size, hold down the **S** key and drag the brush to the right on the canvas. To decrease it, hold down the **S** key and drag to the left. You can also type the brush size in the Size field or drag the value editor in the Size field.



(a) Brush size 100, opacity 1.000 (b) Brush size 25, opacity.500

- **Rate** The brush rate is the rate at which brush strokes are applied to the canvas. A high value produces a smooth continuous stroke, and a low value produces a less continuous stroke with larger gaps between brush images.

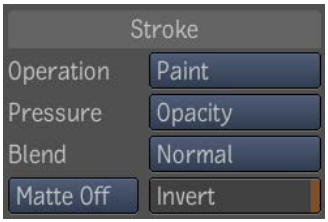
- **Constant** The stamping process can be configured by specifying the stamping rate and forcing constant distance between stamps.

(a) Rate value = 100 (b) Rate value =20 (c) Rate value = 20 + constant

Paint Stroke Controls

Strokes are rendered based on the brush attributes and the stencil. The stroke shape is formed by stamping the brush shape along the path created by the pen (or mouse) movement.

The paint stroke controls have the following parameters:

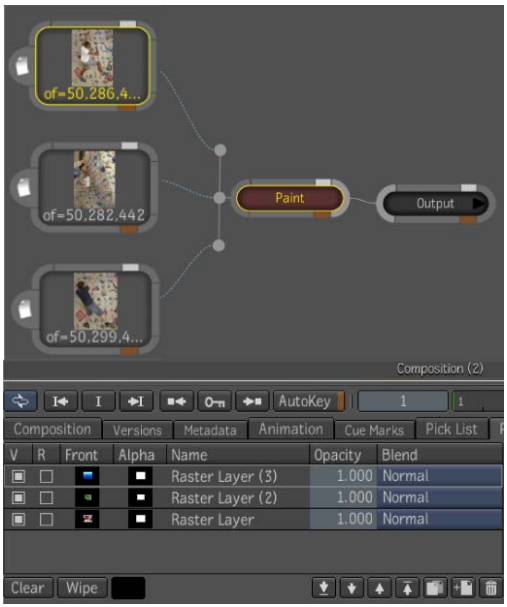


- **Operation** Paint materials define the material used to render the brush. Materials include: Paint (default, solid color), Reveal, Clone and Erase.
- **Pressure** When drawing with a tablet, the pressure can be mapped to the size, opacity, or both (or can be set to Off) of the stamping pattern while drawing.
- **Blend** The blending modes define how the stroke is blended with the current drawing—see Blend Modes on page 345.
- **Matte Off & Invert** You can use stencils to apply paint strokes to an image—see Using Stencils on page 592.

Working with Layers

A raster paint layer is a canvas into which you draw strokes. The strokes are merged together as they are drawn to produce the resulting raster image. Each layer has a front and an alpha. You can select the one on which you want to draw by selecting its proxy image in the Layer Editor.

For example, the following graphic shows that the layer named “University” is selected and its front image selected for painting (it has a white frame around its proxy image).



Raster Layer Format

When you create a new layer in Raster Paint, you can change the layer's format as long as you haven't committed any paint strokes to the layer. To change the layer's format, double click the paint node, then click to highlight the raster layer node. You can adjust the settings in the Raster Layer tab. However, once a paint stroke is committed to the layer, the following settings cannot be changed:

- Format
- Width and height
- Image aspect ratio
- Rate
- Channels
- Depth
- Repeat mode

If you want to change the format settings of a layer, you must create a new layer and start over.

Adding, Deleting, Clearing, and Wiping Layers

All layers appear in the Layers list. The Layers list allows you to view and rearrange layer ordering in the Paint node, as well as select layers to be hidden or revealed.

Layers can be internal or external to the Paint node. An internal layer is a transparent layer that can only be added from the Paint node. It can be used to create, for example, an alpha.

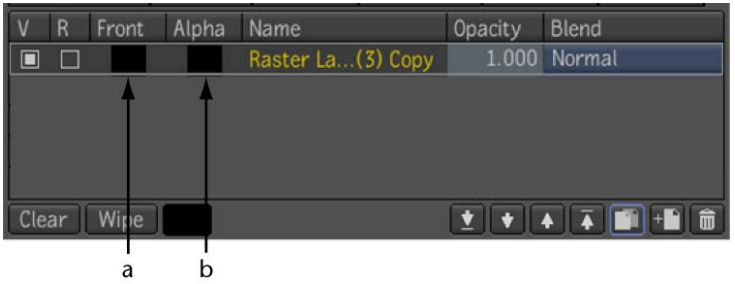
An external layer does not originate from the Paint node. It can be an image and can include a front and/or alpha. An external layer is connected to the Paint node by means of a Layer node, whereas an internal layer needs no such connection since it originates with the Paint node. A common reason for adding an external layer is to touch up an image's front or alpha. When you paint on an external layer, you are compositing strokes onto the incoming image. This makes it possible to change the incoming image without losing your strokes.

When you add either an internal or external layer, it goes to the top of the Layers list. The order in the Layers list reflects the layer order on the canvas. The topmost layer covers the image data of the underlying layers. Layers can be hidden or rearranged within the Layers list.

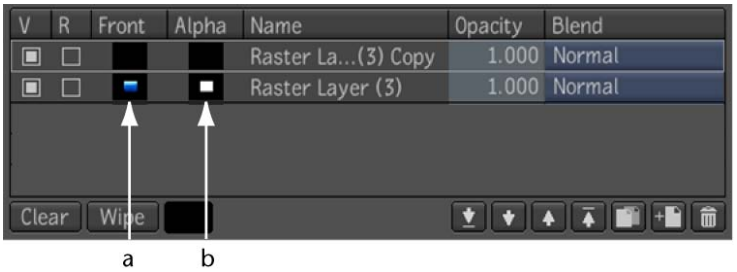
When you delete a layer, it is removed from the Layers list. When you clear a layer, it remains in the Layers list but all strokes are removed.

Adding a Layer

When you add an internal layer, it includes both a layer front and a layer alpha. When you add an external layer that only includes the front, a black layer alpha will automatically be created.



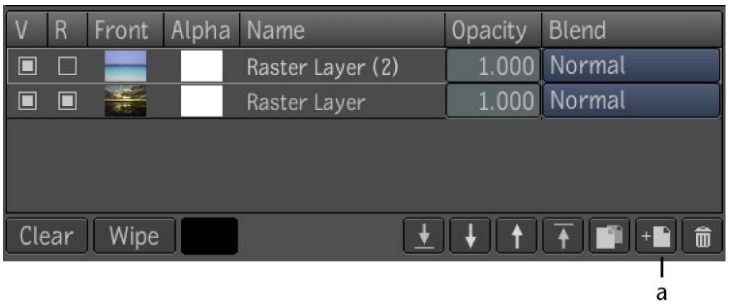
(a) Internal layer - front (b) Internal layer - alpha



(a) External layer - front (b) External layer with its alpha

To add an internal layer:

➤ Click the Add Layer button at the bottom of the Layer Editor.

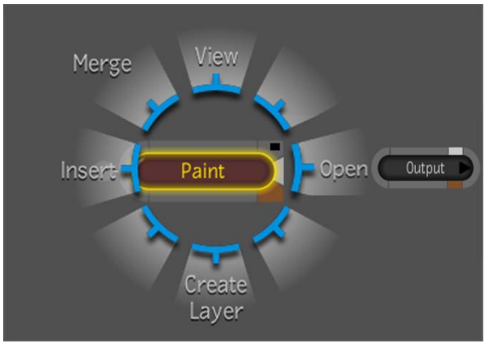


(a) Add Layer button

To add an external layer:

➤ Do one of the following:

- Select the Add Layer button at the bottom of the Layer Editor then connect an input image to the new layer node's input.
- Drop a composition (or compositions) containing a rendered result onto the Paint node and select Create Layer from the drop gate.



NOTE The newly added layer has the same format as the linked rendered result. The drop gate is also activated when you drop a tool or files that can be imported. In that case, the new layer is created using the composition format.

NOTE To rename a layer node, double-click the Paint tool node. Once inside, select the raster layer node to rename. In the Tool Options of the node's UI, type the new name in the Name field and press **Enter**.

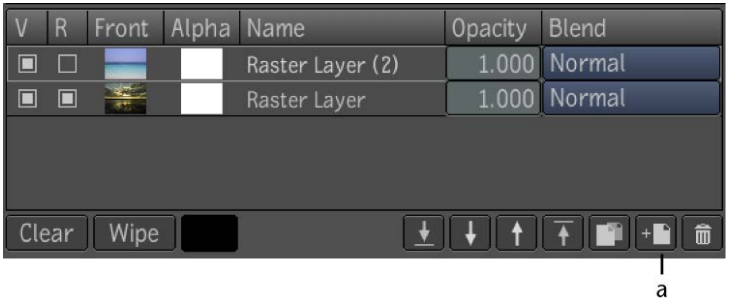
Deleting Layers

You can delete the layers you no longer need.

NOTE You cannot use the **Delete** key to delete a layer node from the Layer Editor.

To delete a layer from the Layer Editor:

1. Highlight the layer you want to delete.
2. Select the Delete Layer button at the bottom of the Layer Editor.



(a) Delete Layer button

The layer is deleted.

Clearing a Layer

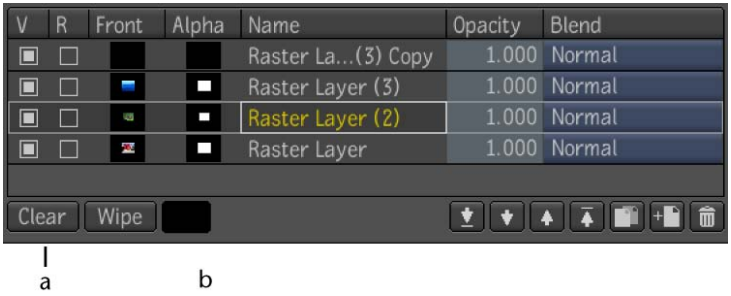
Clearing a layer is the same regardless of the layer type. When an entire layer (not just the front or alpha) is selected for clearing, the front input is set to be cleared by default. You must specify which input of the layer you want to clear by selecting its proxy.

You cannot clear more than one layer at a time. If you select two layers to clear and then select the Clear button, the front input of the first layer you selected will be cleared.

The act of clearing only clears strokes for the current frame. It does not clear the incoming front and alpha of an external layer.

To clear a layer:

1. Select either the front or alpha input from the layer you want to clear (a white box should appear around its proxy).
2. Select the Clear button at the bottom of the Layer Editor.



(a) Clear Layer button (b) Selected layer to clear

The layer is deleted.

Wiping Layers

Wipe a layer when you want to apply a uniform color. For example, you can wipe the color of the bottom layer in the Layers list to change the background color.

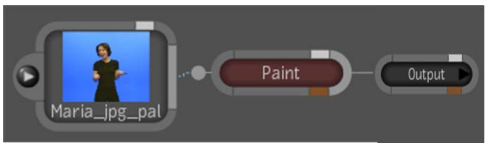
NOTE Any strokes that have been applied will not be maintained when you apply a wipe.

To wipe a layer with a color:

1. Select the layer from the Layers list. Select either the Front or the Alpha input to wipe.
2. Pick the wipe color by clicking the color pot next to the Wipe button—see Color Picker on page 154.
3. Select the Wipe button to apply the color to the layer Front or layer Alpha, depending on which one is currently selected.

Layer Behavior

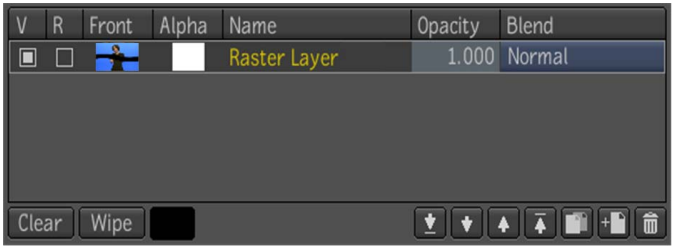
Raster layer behavior is constrained based on the alpha of the layer being manipulated. The dependency graph shown in following image displays a Paint tool node consisting of one external raster layer named Raster Layer.



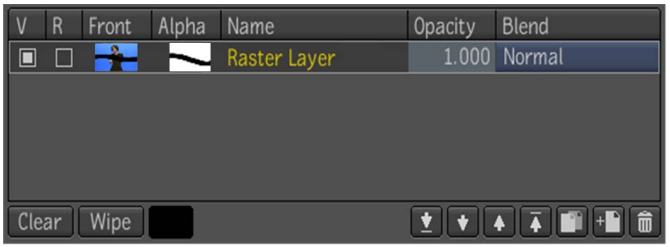
The image doesn't yet have its own alpha channel. By default, the Paint tool creates one for it and is shown as a proxy beside that of the input image illustrated in the following graphic of the Layer Editor.

The following graphic displays the result of painting a black stroke across the front input image.

Note that the alpha channel is not affected (as represented in the proxy image in the Layer Editor) because its input was not selected for a paint operation.

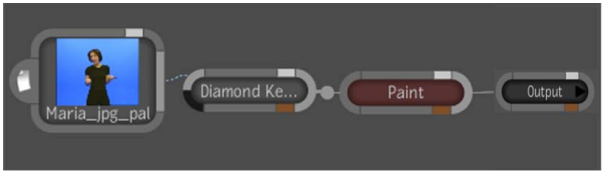


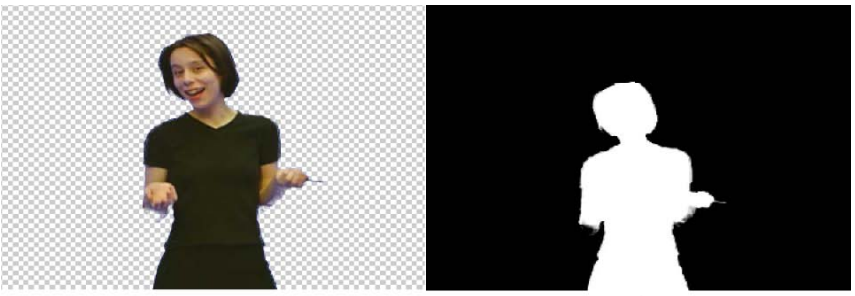
The following images show the result of selecting then painting the alpha of the same image.



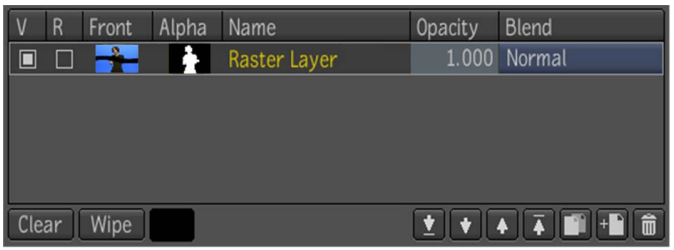
In this case, only the alpha portion of the image was manipulated, however, since a black stroke was used, part of the alpha has become transparent. If a white stroke were used there would be no change to the alpha. If a color other than black or white were used, varying transparencies would occur on the front image and varying shades of gray would result on the alpha.

The next example illustrates layer behavior when the front input image is combined with its own alpha channel. In this case, a Diamond Keyer is added to the dependency graph and the blue screen is keyed out of the image.





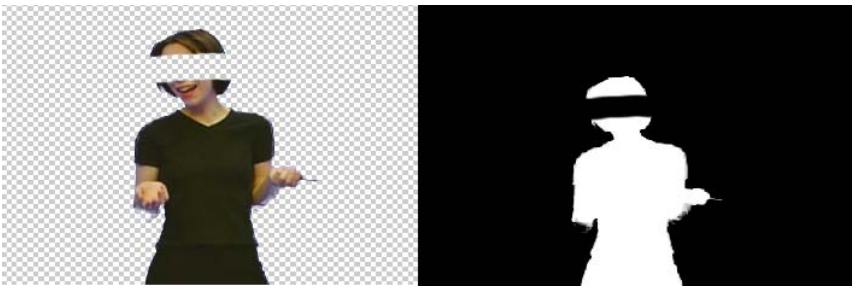
(a) Composition output view (b) Tool output view (Keyer)



When a stroke is added to the front image (in this case using black), the stroke is constrained to the white part of the alpha, and since the alpha was not selected, no change to the alpha was made.

(a) Black stroke contained within boundaries set by alpha

When the alpha is painted with a black stroke, the resulting output image is also altered; more of it becomes transparent.



If a white stroke is used, the transparency of the alpha is decreased.

Stroke Operations

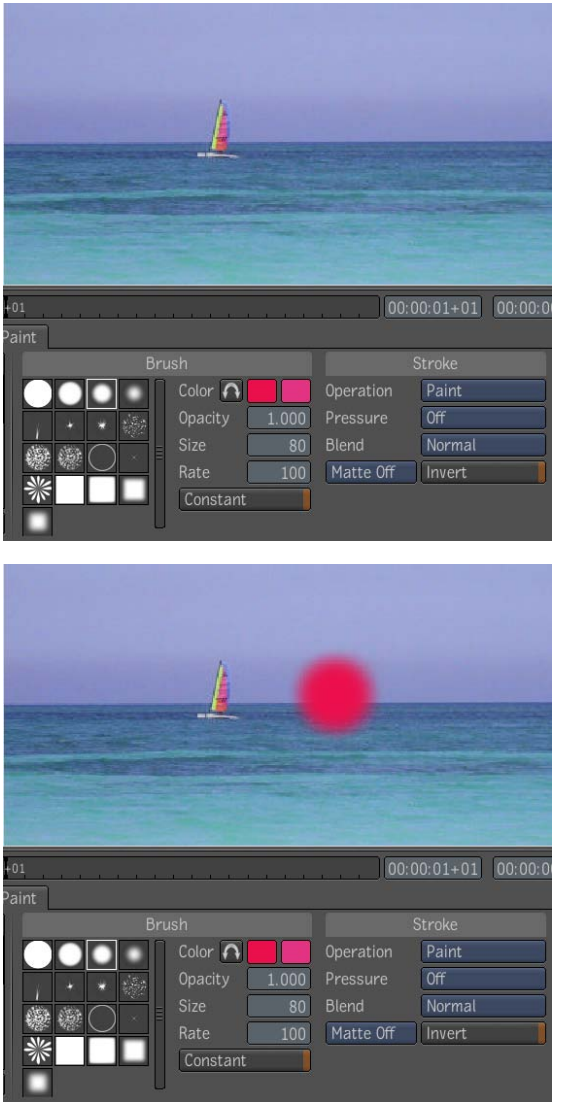
There are four paint stroke operations available in the Paint tool.

Operation	Description
Paint	Lets you paint on a selected layer's front or alpha—see Paint on page 582.
Reveal	Lets you use the contents of one or more layers as the paint source for a different layer—see Revealing Parts of an Image on page 586.
Clone	Lets you copy a region of a layer and paint it on a destination area—see Cloning a Region of an Image on page 589.
Erase	Lets you erase paint strokes—see Erasing Paint Strokes on page 592.

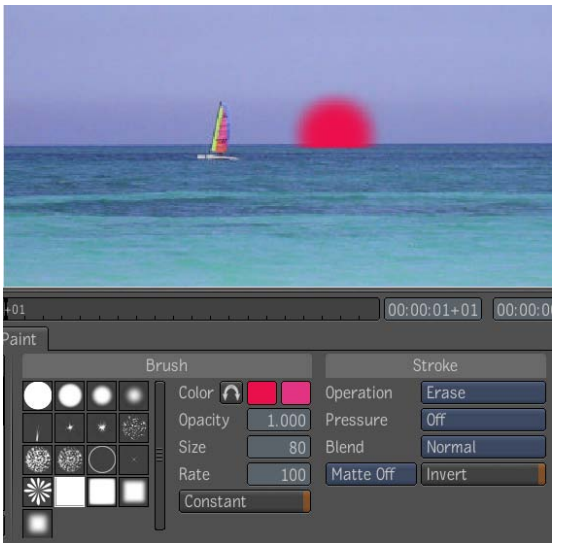
Paint

The Paint stroke operation lets you paint on a selected layer's front or alpha. Each raster layer has a front image and an alpha which you can apply strokes to.

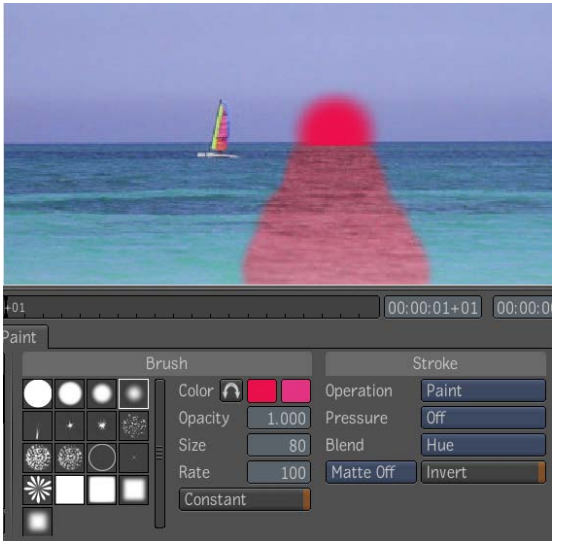
In the following example, a red sunset is added to the image of the ocean. In the first image a round brush with a slight blur around its edge is used to create the sun.



To hide the lower half of the sun so that it lines up with the horizon, a sharp-edged square stroke was used, and the stroke operation was changed from Paint to Erase. The sun was then painted out.



The last step is to add the sun's reflection to the water.



Drawing a Straight Line

To draw a straight line:

1. Click the location in the image where you want the line to begin.
2. Press **Shift** and click the location in the image where you want the line to end.

   The line is drawn between the two points.

Painting the Alpha

You can retouch mattes with the paint tool.

A new internal layer has an empty, black layer matte. An external layer's alpha contains the matte information of the incoming image.

Painting on the alpha of an internal layer is accomplished in one of two ways: implicitly or explicitly. Implicit alpha strokes are added when you paint on the layer front. For every stroke added to the layer front, a corresponding stroke is added to the layer alpha. Explicit alpha strokes are added when painting directly on the layer alpha.

Painting on the alpha of an external layer can only be accomplished explicitly. When you add strokes to the layer front, corresponding strokes are not added to the alpha.Strokes added explicitly to the layer alpha will be visible on the layer front (nor in its proxy) in composition view when you commit the stroke(s).

Pen to be removed from image shown in the alpha and the comp.

Unwanted areas can be removed from the image by painting on the alpha of the image.

Revealing Parts of an Image

The Reveal operation lets you use the contents of one or more layers as the paint source for a different layer. You can include the contents of as many layers as you like. For example, if one layer has a tree that you want to include on a different layer, you can select the layer with the tree and then paint it onto the other layer.

You can also offset the layer to be revealed. For example, if a bicycle appears at the left on one layer, but you want to paint it onto the right side of another layer, you can offset the reveal destination.

To reveal one or more layers:

1. Under Stroke, select the Reveal operation.

   The brush changes from white to red.

2. Set Pressure and Blend attributes—see Blend Modes on page 345.

3. Set the brush attributes—see Paint Brush Controls and Attributes on page 569.

4. For each layer you want to reveal, select the R (reveal) option in the Layer Editor. If you selected the R (reveal) option for more than one layer at once, paint strokes will reveal the layer at the bottom of the list.

   NOTE You can reveal layers that are marked as hidden in the Layers list.

5 To reveal the contents of a different frame than the one you are currently viewing, enter its frame number in the Slip field.

6 In the Layer Editor, click the layer that you want to paint on.

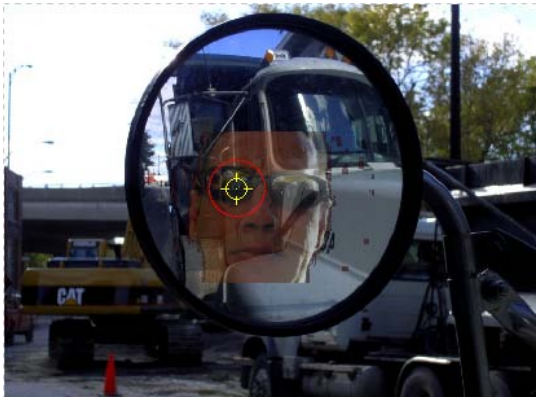
a b

(a) Layer to paint (b) Layer to reveal

7 To see the layer or layers selected for reveal, select the V (view) option, and adjust the opacity.

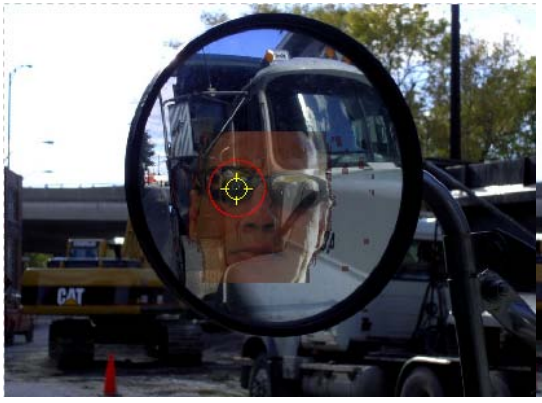
The Preview option offers a more intuitive way of previewing the effects of the reveal. When you enable Preview, the layer is superimposed over the layer or layers selected for reveal, and the Opacity field is activated to allow for transparency adjustments. This lets you see exactly what your brush strokes will reveal.

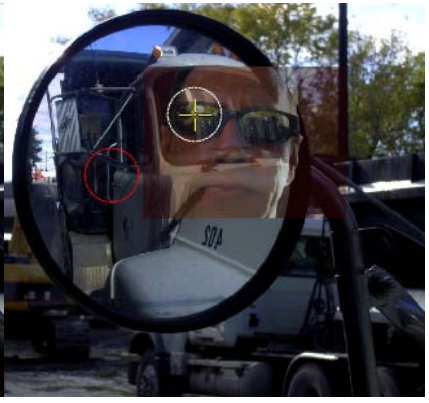
8 Click and drag on the canvas to apply strokes that reveal the contents of the selected layers.



To reveal and offset a layer:

1. Perform steps 1 to 6 from the previous procedure, “To reveal one or more layers”.
2. To set the source area, press **Ctrl** and click the canvas. When you click the canvas, the red brush is anchored and the green brush reappears.





Setting source area

Setting destination

3. Position the green brush over the destination area and click the canvas. The offsets are made and immediately shown in the X and Y Offset fields.

   The green and red brushes are now locked into positions relative to each other and move in tandem.

4 To set precise coordinates for the reveal offset, use the X and Y Offset fields.

5 Click and drag on the canvas to apply offset strokes that reveal the contents of the selected layers.

Revealed portion of layer using Average blend method.

Cloning a Region of an Image

The Clone tool lets you copy a region of the layer and paint it on a destination area. For example, if your clone source includes a sailboat, when you paint on the destination area, the sailboat will be replicated. The clone source and

destination occupy the same layer. To clone part of one layer on a different layer, use the Reveal tool—see Revealing Parts of an Image on page 586.

To clone part of a layer and paint it on the same layer:

1. Under Stroke, select the Clone operation.

   The brush changes from green to red.

2. Set Pressure and Blend attributes—see Blend Modes on page 345.

3. Set the brush attributes—see Paint Brush Controls and Attributes on page 569.

4. To clone the contents of a different frame than the one you are currently viewing, enter its frame number in the Slip field.

   In this example, the sailboat will be cloned to a location further to the right.

5. Set the source area of the clone and the target destination for the clone by pressing **Ctrl** and clicking once. When you click the canvas, the red brush is anchored and the green brush reappears.

NOTE The Preview option offers a more intuitive way of previewing the effects of the reveal. When you enable Preview, the layer is superimposed over itself, and the Opacity field is activated to allow for transparency adjustments. This allows you to see exactly what your brush strokes will clone.

6 Position the green brush over the destination area, then click + drag the canvas to create the clone stroke. The offsets are made and immediately shown in the X and Y Offset fields.

The green and red brushes are now locked into positions relative to each other and move in tandem.

7 To set precise coordinates for the reveal offset, use the X and Y Offset fields.

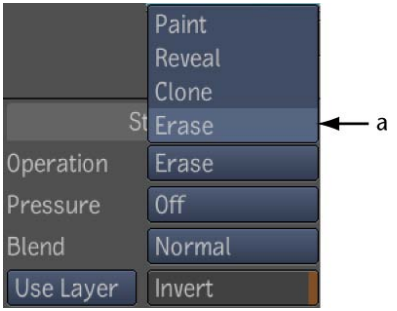
Cloning of a sailboat in preview mode.

Erasing Paint Strokes

When you erase paint strokes, you can select the brush size and type for different effects.

To erase paint strokes:

1 Under Strokes, select the Erase operation.



2 From the Layer Editor, select the layer on which you want to erase strokes.

3 Set a brush type and size—see Paint Brush Controls and Attributes on page 569.

4 Click and drag on the canvas to erase.

Using Stencils

You can enable/disable a stencil to paint without or through a stencil on a per stroke basis. The stencil is applied transiently, only the part of the strokes that pass through it are kept.

There are three stencil modes:

Stencil:	**Result:**
Matte Off	Default setting, no stencils are used.
Use Layer	Strokes will be constrained by the matte inputted into the layer.
Use Stroke	Forms a stencil from what is currently available in that layer.
Invert	Inverts the stencil.

Vector Paint

25

Topics in this chapter:

About Vector Paint

Composite's Vector Paint tool can be used to draw, to rotoscope, to clone parts of an image in a different location, or to reveal a completely different image.

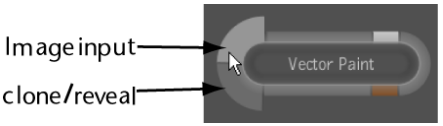
In the Vector Paint tool, you draw by creating shapes on the canvas. Contrary to Composite's Raster Paint, once a shape is created in Vector Paint, you can edit its parameters to change the shape's location and orientation, for example using a tracker to animate the shape. If the shape is a Bézier, you can edit individual points, changing their opacity or their tangents, for example. You can do this using the same key frame animation techniques you use in other Composite tools. All the shapes drawn in a given Vector Paint tool are composited back to front over an optional background image to produce the Vector Paint output image.

Additionally, Vector Paint has strong similarities with other Composite shape-based tools, such as the Garbage Mask, Remove Dust, and Warp 2D tools.

Vector Paint Image Inputs

You can optionally connect images to the Vector Paint image inputs. The Vector Paint node in the Schematic has two image input tabs. The top-most tab is the background image input. Connect an image to this input to use as your canvas background. If left unconnected, your paint shapes will be applied over a black background.

The bottom-most tab is for clone/reveal image sources. You can connect as many images as you wish to this tab, and select any one of these sources to be the clone/reveal source for a paint shape drawn in clone or reveal mode. If left completely unconnected, the only possible clone/reveal source will be the background (or black, if the background is also unconnected).



Shape Properties

When you draw using the Vector Paint tool, you are creating shapes on the canvas. All shapes share basic properties, such as:

- Translation
- Pivot Point
- Rotation
- Scale
- Shear

Other shared properties control how the shape is drawn on the canvas, and are described in the following sections. These properties can be edited and animated over time. Unless otherwise noted, all shape properties are animatable.

In Vector Paint, shapes are of two types: Bézier or line strip.

Line strip shapes are also called freehand shapes. These shapes are created using the freehand tool (if Auto Make Bézier is off). Line strips can have large numbers of individual points with better performance than a Bézier curve with the same number of points. The drawback is that individual points in

line strips cannot be edited. To edit them, make a Bézier curve out of the line strip.

Shapes in Vector Paint have a duration: they appear at their Mark In frame, and are removed at their Mark Out frame.

Vector Paint UI

The Vector Paint UI is composed of five tabs: the main (Vector Paint) tab, the Transform tab, the Brush tab, the Output tab, and the Settings tab. Each tab has two panes, with a divider between them that can be used to adjust the relative size of the two panes.

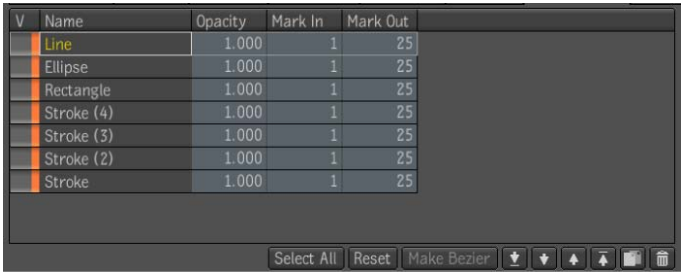
The left pane, the shape and tools pane, is identical in all five tabs. It is composed of the shape browser on the left, and on the right, a column with shape creation and editing tools, Bézier creation controls,and pen pressure controls.

All controls in the Vector Paint UI feature tool tips to provide information about their use.

Shape Browser

The shape browser displays all shapes created in the current Vector Paint tool. By default, the shape browser presents shapes in bottom to top compositing order, with the bottom-most shape at the bottom of the shape browser, and the top-most shape at the top. The name column is the only column on which sorting can be done. The sort order can be inverted by clicking on the name title to provide front to back ordering (back stroke at the top of the browser).

The following columns are shown in the shape browser by default:



Use:	To:
V	Control the visibility of the shape (mute or view).
Name	Provide a meaningful name for the shape.
Opacity	Set the shape opacity.
Mark In	Set the frame at which the shape becomes visible.
Mark Out	Set the frame at which the shape disappears.

The following columns are optional, and can be added to the shape browser display by right-clicking on the title row of any existing column.

Use:	To:
L	Lock. When enabled, prevents any change to the shape through the browser or in the player.
Blend	Blend mode of the shape with the composited shapes and/or background image behind it.
Stream	Set stream in which the shape will be drawn, for multi-stream or stereo painting.
D Mode	Draw Mode for the shape, stroked or filled.
C	Open/Closed state.
Oper	Operation, paint, clone, or reveal.

Use:	To:
Channels	Which channels will be affected by the shape.
AA	Anti-Aliasing. Filled shapes (without a gradient) will have sub-pixel antialiased rendering, and stroked shapes will have sub-pixel positioning of strokes.

In general, all columns (except Lock) have their equivalent in the main tab's main section.

At the bottom of the shape browser are controls for operating on shapes.

Use:	To:
Select All	Select all shapes.
Reset	Reset selected shape(s).
Make Bézier	Convert selected line strip shape(s) into shapes.
Move to Back	Move selected shape(s) to the back of the compositing order.
Move Down	Move selected shape(s) one position down towards the back of the compositing order
Move Up	Move selected shape(s) one position up towards the front of the compositing order.
Move to Front	Move selected shape(s) to the front of the compositing order.
Duplicate	Duplicate selected shape(s).
Trash	Delete selected shape(s).

Shape Creation and Editing Tools

At the right of the shape and tools pane are the shape creation and editing tools, as well as Bézier creation controls and pen pressure controls. The creation

tools form the first row of the shape creation and editing tools, and the editing tools form the second.

Use:	To:
Freehand	Create a freehand line strip shape.
Bézier	Create a Bézier shape point by point.
Rectangle	Create a rectangular (or square) Bézier shape.
Ellipse	Create an elliptical (circular) Bézier shape.
Line	Create a line Bézier shape.
Transform	Edit a shape's transform.
Edit	Edit points in a Bézier shape.
Clone/Reveal Transform	Edit the clone/reveal transform of a shape.

NOTE Notice that the Rectangle, Ellipse, Line and Bézier shape tools all create Bézier shapes.

Immediately below the shape creation and editing tools are the Erase controls. Contrary to many other drawing packages, Vector Paint does not have dedicated erasing tools. Instead, you choose the erase mode you want (erase to background, to transparent, or to background color),and use the standard creation tools to create new shapes that erase. These shapes can be edited, tracked, and changed just as other shapes.

Make Bézier Controls

Use:	To:
Auto	Automatically convert new freehand shape.

Use:	To:
Tolerance	Set the pixel tolerance when converting freehand shapes to Bézier.

Pen Pressure Controls

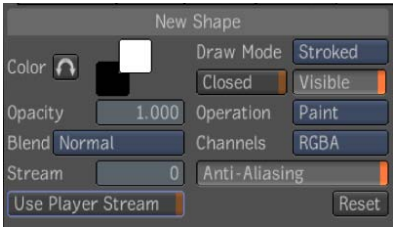
Use:	To:
Size	Have pen pressure control brush size.
Opacity	Have pen pressure control opacity.

Main Tab

Vector Paint's main tab contains the most common settings that affect a shape.

Main Shape Attributes

The leftmost section in the main tab contains the basic shape controls. Its title shows the currently selected shape's name, or "New Shape" if a creation tool is selected.



Use:	To:
Color	Set the two colors (named "front" and "back") used by shapes drawn with the Paint operation (see Operation, below). The front color is used for drawing, and the back color is used for erasing. Another use for the back color is to toggle quickly between two colors, either using the swap button in the UI or by using the hotkey (default: X)

Use:	To:
Opacity	Control shape opacity.
Blend	Control the blending mode of the shape. See Blend Modes in the Blend Alpha on page 344 section.
Stream	Set the stream in which the shape will be drawn. Note that this parameter is not animatable. See Multi-stream Compositing on page 371.
Use Player Stream	This button is only visible when a creation tool is selected. When creating a shape in a player, this mode will set the new shape's stream to be that of the player. In a stereo workflow, this allows you to draw in one stream (e.g. left stream), change the player's stream, and draw in the other stream.
Draw Mode	Set the shape's draw mode. Stroked shapes will stamp the brush image along the shape's path. Filled shapes will fill the polygon defined by the shape.
Closed	Set whether the shape is considered to be an open or closed shape.
Visible	Enable rendering for this shape (whether or not it is visible).
Operation	Set the shape to Paint, Clone, or Reveal. Paint will draw using the solid foreground color. Clone and Reveal will use an image source to draw.
Channels	Set which channels of the image will be affected by the shape. Alpha modes use the current color's luminance.
Anti-Aliasing	Set rendering quality, depending on the draw mode. For Stroked shapes, this controls whether sub-pixel positioning of brush stamps will be used. For Filled shapes, this

Use:	To:
	controls whether smoothing will be applied to pixels that are only partially covered by the shape.
Reset	Reset all parameters described in this section.

Clone/Reveal Attributes

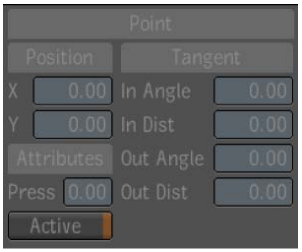
This section contains attributes relevant to shapes that use an image source as their color content, when the shape Operation is Clone or Reveal, as opposed to a solid color, when the shape Operation is Paint.

Use:	To:
Source	Select the image source that will be cloned or revealed into the shape. The menu offers the images that are connected to the node's "Clone/Reveal Sources" input, as well as the background image, labeled as [Background].
Translation X/Y	Set the X, Y translation for the clone/reveal image source.
Rotation (R)	Set the rotation for the clone/reveal image source.
Frame	Select which frame to use as the clone/reveal source, according to the Absolute toggle.
Absolute	Control the interpretation of the Frame value. Enable to have the Frame value be

Use:	To:
	an absolute frame index, disable to interpret the Frame value as an offset relative to the current frame.
Edges	Control how pixels outside the clone/ reveal image source are generated. Transparent sets them as transparent, Edge extends the edge pixel of the clone/reveal source, Repeat repeats the clone/reveal image source, and Mirror mirrors the clone/reveal image source.
Filtering	Control how pixels read from the clone/ reveal image source will be filtered before being applied by the stroke. Nearest is the fastest, but lowest quality. Bilinear provides very good quality at excellent speed. Other modes provide excellent quality, at far slower speeds. See Cropping and Filtering the Output on page 482.
Preview	Control drawing the clone/reveal image source overlay, to ease placement of the source. This can be very useful in combination with the clone/reveal transform tool.
Preview Opacity	Control the opacity of the clone/reveal image source overlay.
Reset	Reset all parameters described in this section.

Point and Gradient Attributes

This section shows attributes for the currently selected shape point or edge gradient point. These will only be enabled when the Edit tool is selected. Please note that only Bezier shapes have editable points. Line strips must first be converted to Bezier shapes if you need to edit individual points. For more information about Point attributes, see the Editing Masks on page 543 in the Masking chapter.



Use:	To:
Position X,Y	Controls the point's position.
Presssure	Controls the point's pressure. This can be used to change brush size or shape opacity.
Active	Controls whether the point is used for drawing the shape. Since this parameter is animatable (as almost all other Vector Paint parameters), deactivating a point over a range of frames may be useful if you want to keep points part of the shape, but temporarily don't need it.
Tangent	Control the incoming and outgoing tangent angle (in degrees) and length.

Gradients can be added to any Bézier shape, but not to line strips. Gradients will only be rendered on filled shapes, not on stroked shapes. Shapes don't need to be closed for gradient to render. This is a good way to have a stroke with detailed falloff control.

Edge gradients can be added by adding a gradient point on the curve with Ctrl-Click on Windows and Linux, Cmd-Click on Mac OS. For more information about Edge gradient attributes, see Creating and Editing Edges on page 554 in the Masking chapter.

Use:	To:
Distance:	Control edge gradient point distance from the shape. Negative means inside the shape if the shape is closed.
Position:	Control gradient point position on the shape relative to the shape's point indices. For example, 2.5 represents a gradient point midway betweeen points 2 and 3, while 2.75 is 3/4 of the way from point 2 to 3.

Transform Tab

This tab displays controls for two separate transforms that affect the shape: Shape Transform and Clone/Reveal transform.

The Shape Transform controls the shape's position and aspect. This section is only enabled when an editing tool is selected. For more information about the shape transform, see Transforming Masks on page 548 in the Masking chapter.

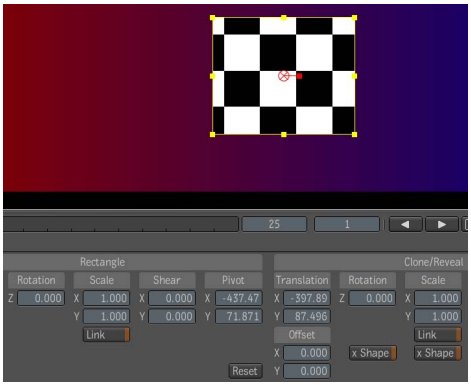
The Clone/Reveal Transform controls the transform of images used in clone/reveal shapes. These controls only have an effect if the shape's current operation is Clone or Reveal. The Paint operation does not make use of this transform.

In Clone mode, by default, only the translation is composed with that of the shape, which provides the expected result when dealing only with translations. Using the "x Shape" (by shape) toggle controls, other parts of the

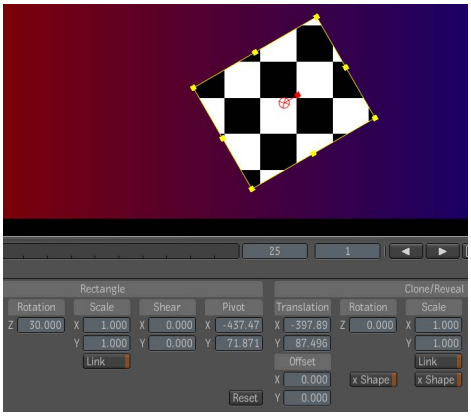
transformation will be composed by the shape transform. If the "x Shape" toggle is off, they will be absolute.

In Reveal mode, all parts of the transform are absolute, with scale, rotation and shear with respect to pivot, as expected. The "x Shape" toggle controls are not used.

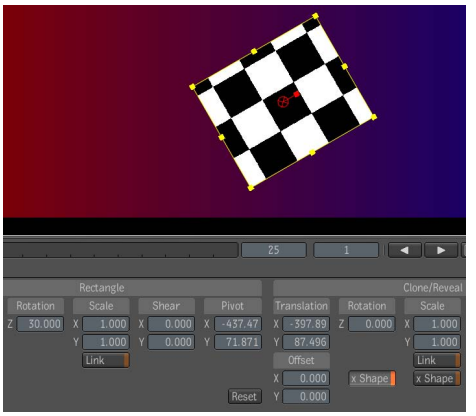
The examples below display the rotation attribute for rectangle and clone/reveal areas.



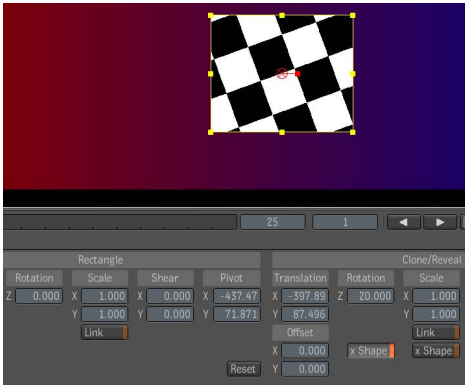
Default rotation position



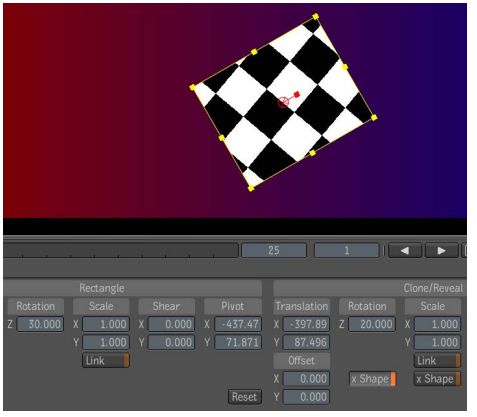
Rectangle rotation at 30 .



Rectangle rotation at 30, x Shape button enabled.



Clone/Reveal rotation at 20, x Shape button enabled.



Rectangle rotation at 30, Clone/Reveal rotation at 20, x Shape button enabled.

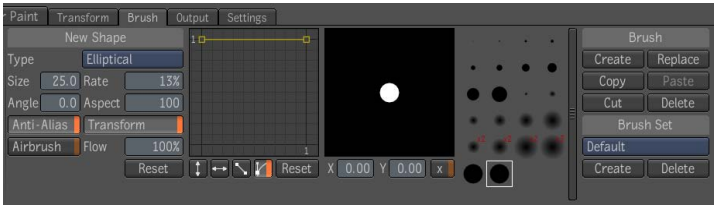
NOTE Clone/reveal transform controls are also found in the main tab for translation x, y and rotation.

Brush Tab

The brush tab has controls that affect drawing of stroked shapes. As for other tabs, it has the shape and tools pane at the left. In the right pane, from left to right, are the following columns:

- Brush Attributes
- Brush Profile Editor
- Brush Preview
- Brush Palette
- Brush Palette Controls

Brush attributes control how new or existing stroked shapes will be rendered.



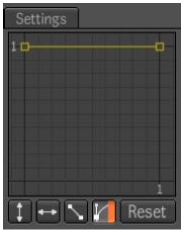
Use:	To:
Type	Control brush base shape, elliptical or rectangular.
Size	Control brush size in nominal pixels. Result is independent of pixel aspect ratio.
Rate	Control spacing between stamps, in percentage of brush size.
Angle	Set the brush shape rotation in degrees.
Aspect	Control the aspect ratio of the brush shape.

Use:	To:
Anti-Alias	Enable/disable anti-aliased brush rendering.
Transform	Enable/disable whether brush transform is affected by shape transform.
Airbrush	Toggle to create stamps when no pointer movement is detected
Flow	Factor applied to the brush profile when Airbrush is enabled.

Brush Profile Editor

The Brush Profile Editor allows you to specify a radial profile for your brush. You can use this to create a smooth falloff, or add control points to create a complex profile for artistic effect. Click profile control points to select them, and click and drag control points or tangent points to change the profile. Alt-click will add a profile control point. Use Ctrl-Alt-Click to break tangents.

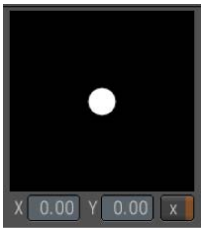
At the bottom of the Brush Profile Editor are controls to modify your profile.



Use:	To:
Flip Vertical	Flip brush profile vertically, along with its animation.
Flip Horizontal	Flip brush profile horizontally, along with its animation.
Straighten	Straighten selected brush profile segments.
Edit Tangents	Enable/disable tangent editing.
Reset	Reset the brush profile to its default.

Brush Preview

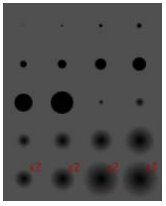
The brush preview displays a rendered preview of the brush, with all parameters applied, including its profile. At the bottom of the brush preview are brush center controls.



Use:	To:
X	Set brush center x coordinate, relative to the brush size. -1 is left, 1 is right.
Y	Set brush center y coordinate, relative to the brush size. -1 is bottom, 1 is top.
x	Display brush center indicator in preview.

Brush Palette and Brush Palette Controls

The brush palette displays a preview of all brushes in the current brush set. Click on a brush to use it with the current shape. Shift-click to use only the brush profile. Click in an empty location to create a brush.



Once a brush is selected, you can use the brush palette controls to replace, copy, paste, cut, or delete the brush selection. You can also select a different brush set, delete the current one, or create a new brush set.

Output Tab

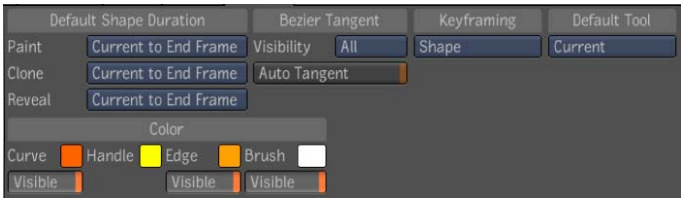
The Output tab provides controls that are used when Vector Paint does not have an image connected to its background image input. In such a case, Vector Paint behaves as an image generator.

For more information about image generator properties and formats, see Setting Image Generation Properties and Formats on page 420 section in the Image Processing and Tools chapter.

Settings Tab

The Settings tab provides controls to change default shape durations, colors for symbols drawn in the player, Bézier tangent settings, keyframing settings, and post shape creation tool selection.

For Bézier tangent settings, Keyframing settings, and Default Tool settings, see Options Tab Settings on page 559 in the Masking chapter.



Use:	To:
Default Shape Duration	Set the default duration for newly-created-shapes using the Paint, Clone, and Reveal-operations, respectively.
Curve Color	Set the curve color in the player.
Curve Visible	Enable curve visibility when the edit tool is selected.

Use:	To:
Handle Color	Set the Bézier handle color in the player.
Edge Color	Set the edge gradient curve color in the player.
Edge Visible	Enable edge gradient curve visibility in the player.
Brush Color	Brush outline color in the player.
Brush Visible	Brush outline visibility in the player.

Hotkeys

For a detailed list of Vector Paint hotkeys, open the Hotkey Editor, and go to the "Tools --> Vector Paint and Tools --> Shape Editor" category. For more information about the Hotkey Editor, see Hotkey Editor on page 163 in the Getting Familiar with your Workspace chapter.

Some hotkeys are not listed in the hotkey editor, as they are fixed and cannot be changed. These are:

- Alt for creating lines constrained to multiples of 45 degree angles.
- Alt for constraining shape rotation to multiples of 45 degree angles.
- Alt for creating centered ellipses/rectangles.
- Shift for keeping 1:1 proportions when creating ellipses and rectangles, thus creating circles and squares.
- Ctrl-alt-click in the brush profile editor, to break tangents.
- Shift-click in the brush library to use pictured profile without changing the current brush parameters (size, rate, angle, ...).
- Alt-click on the "Replace" button in the brush library, to replace the current brush without confirmation.
- Alt-click on the "Delete" button in the brush library: delete the selected brush without confirmation.

Performance Notes

Here are some notes on the following topics: Rendering, Clone/Reveal Filtering, Mulit-Shape Editing and Manipulators.

Rendering

Vector Paint uses the normal Composite high performance computation engine when rendering images used by the nodes connected to its output (downstream nodes). This provides the best overall performance and throughput, but can introduce noticeable lag (or latency) when drawing shapes in a player.

For this reason, in player Tool Output display mode, Vector Paint uses a low-latency renderer that ensures all interactions are fluid and images are produced with minimal lag.

You can use this to your advantage depending on the type of interaction you need. For example, if you need maximal performance and are not creating strokes interactively, you can connect a Pass-Through tool node to Vector Paint's output and set a context point on the Pass Through, to view Vector Paint's output. When rendering large numbers of strokes on large images, this will produce the highest throughput. The quality and pixel accuracy of both rendering methods is identical.

Clone/Reveal Filtering

Bilinear filtering has an optimization for situations where only translations are used that will make it notably faster than the other filters.

Using clone/reveal sources or a background with non-square pixels disables filtering performance optimizations.

Multi-Shape Editing

When multiple shapes are selected, parameter changes made in the UI will apply to all selected shapes. The following actions, however, do not follow this convention and only apply to the primary selected shape:

- Changing the Front and Back Color in the Main Tab.
- Changing shape and edge gradient points in the Main Tab.

- Changing the brush profile, in the Brush Tab.
- Resetting parameters through ctrl-click or right-click.

Manipulators

As for the other shape-based tools, many aspects of the the shapes' appearance and positioning can be modified by manipulating them directly in the player.

One notable exception to this is the shear parameters which, for both shape and clone/reveal transforms, can only be modified through the Transform tab.

Shape Transform Manipulators

When the Transform tool is active, the selected shapes' pivot and bounding box are drawn on top of the rendered shape. Handles on these symbols allow the translation, rotation, scale and pivot to be manipulated. Please refer to the Transforming Masks on page 548 section for detailed information about shape transform manipulation.

Point Editing Manipulators

When the Edit tool is active, the selected shape's curve, points, edge gradient (if any) are drawn on top of the rendered shape.

The color used to draw these symbols can be changed in the Settings tab, and applies to all the shapes in the currently selected Vector Paint node.

The curve and edge gradient symbols can be turned off, if needed, to avoid clutter. See the Settings Tab for these parameters, and consult the Hotkey Editor for the hotkeys associated with toggling the curve and edge gradient visibility

Please refer to the Editing Masks on page 543 section for detailed information about point and edge gradient manipulation.

Clone/Reveal Transform Manipulators

Shapes using the Clone or Reveal Draw mode can have their source image transform adjusted directly in the player using the Clone/Reveal Transform

tool. This manipulation tool is available for single-shape selections, and its manipulators will only be drawn if the selected shape's Operation is set to Clone or Reveal.

The manipulators appear in the player at the pivot position for the image being used. This means that the manipulators will not necessarily be close to the actual shape using the image. The shape's name is indicated beside the manipulator to identify which shape's clone/reveal transform is being edited.

This tool has two editing modes: one for editing the translation/rotation/scale, and one for editing the pivot and its rotation. You can toggle between the two using the "Set Clone|Reveal Pivot" hotkey, which defaults to "Ins". When the pivot is being edited, the text "(Pivot)" appears after the shape's name close to the manipulator.

This manipulation tool is the same as the one found in the 2D Transform tool. Please refer to the 2D Transform Tool on page 432 for further details.

Please note that the clone/reveal preview can be enabled when manipulating to have a better idea of which section of the image is being used.

Color Correction

26

Topics in this chapter:

About Color Correcting

Color correction describes any process that modifies the perceived color of an image. The CC Basics tool options include a complete selection of color manipulation tools that give you precise control over color values. You can modify luma ranges in an image (shadows, midtones, and highlights), sample colors, and set the color balance. You can also apply simulated camera exposure and film development effects to images using the Photo Lab tool.

NOTE All color correction tools have a masking input that allows you to limit the effect of the color correcting tool to a specific range of pixels—see Pixel Masking on page 560.

Applying LUTs

Film projects generally use digitized film images originating from a datacine conversion process. These high-resolution scans of film frames preserve much of a film's resolution, color depth, and grain. Digital film scan formats consist of logarithmic data, which allows for more subtlety in the lower luma ranges of an image. Linear data maps the luma range in equal intervals from black to white. Because the algorithms are optimized for linear data, a look-up table (LUT) can be used to convert the logarithmic data for a linear data environment.

A LUT is an ASCII file that describes specific pixel values used for image data conversion both on import and export. For example, you can use a LUT to convert a 10-bit logarithmic film image to a 16-bit linear image on import. Typically, you would apply the inverse LUT on export to restore the 10-bit logarithmic data, ensuring that the result film prints match the original shots. You can use two kinds of LUTs: 1D LUTs and 3D LUTs.

Calibrating your Monitor

Before using LUTs, you should calibrate your monitor. Monitor LUTs affect the way images are displayed on the monitor without altering the image data. If you import logarithmic film scans without conversion, you need to use a monitor LUT to display the images correctly. Even when you apply a LUT to film scans on import, you may still require a monitor LUT to reproduce the gamma qualities of film—see *Chapter 4, Lustre Color Calibrator* in the *Lustre Color Management User's Guide* on the Composite Web Portal.

Working with 1D LUTs

A 1D LUT is generated from one measure of gamma (white, gray, and black) or a series of measures for each color channel. You typically work with a pair of 1D LUTs: the first converts logarithmic data to linear data, and the second converts the linear data back to logarithmic data to print to film.

1D LUTs can be used as import and export LUTs, and as monitor LUTs. Import and export LUTs are mainly used to convert between the logarithmic data contained in film scans and the linear data used within the application. Monitor LUTs ensure the image displayed on the monitor resembles the final output without the actual image data being modified.

You can apply a 1D LUT when importing a film clip, and then convert the image data back to logarithmic data when exporting the final output. For example, you can use a 1D LUT on import to convert 10-bit to 8-bit, 16-bit, 32-bit, or linear images for manipulation. Typically, you would then apply the inverse LUT on export to restore the logarithmic data, ensuring that the modified film print exactly matches the original shot.

NOTE You can use 1D LUTs created outside the environment. However, to be recognized, the 1D or 3D LUT file must be in the correct format and be named correctly. Comment lines are permitted.

Using 1D LUTs Created Using Third Party Software

You can use 1D LUTs created in other applications as long as they are in the recognized format. All 1D LUT files must have the *.lut* extension. You should save all 1D LUTs in the project's LUT folder for ease of use, and so they can be archived with the setups of the project.

The 1D LUT Format

The 1D LUT file begins with a declaration of the number of tables and the number of entries per table:

```
LUT: <number of tables> <number of entries per table>
```

The number of tables in the file depends on the channel resolution required. For a LUT in which all three RGB channels are converted using the same values, just one table is required. To convert the three channels using different values, use three tables. For three channels plus alpha, use four tables. The number of entries in each table corresponds to the bit depth of the source file: 256 entries for 8-bit resolution, 1024 for 10-bit and 4096 for 12-bit. For example, a LUT operating on three 8-bit channels independently would have the following declaration:

```
LUT: 3 256
```

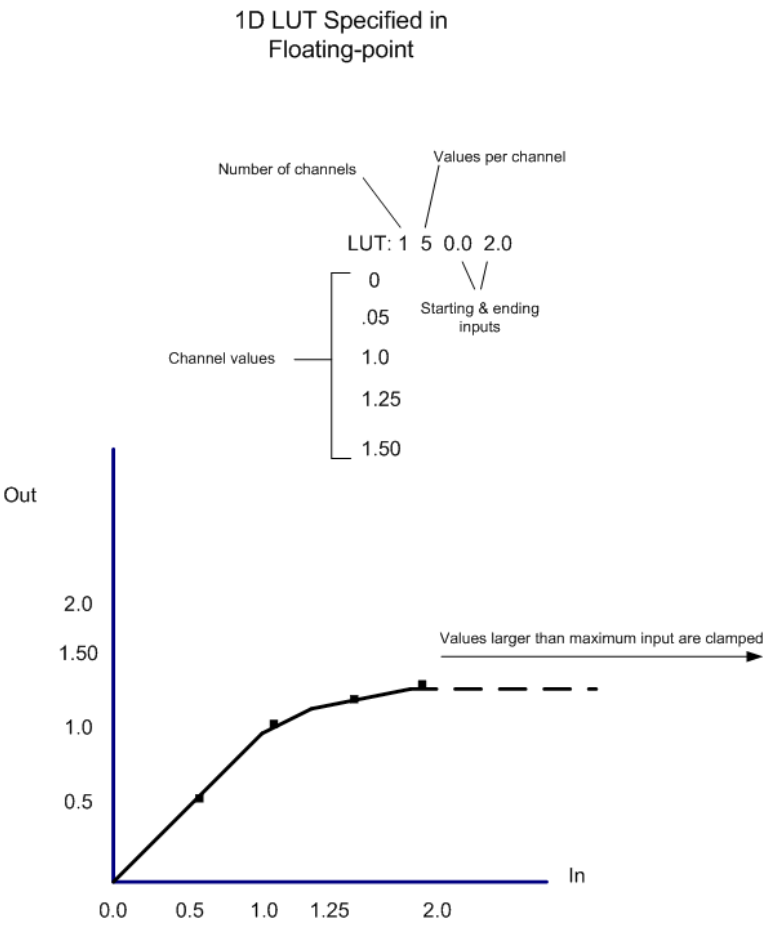
Each line following the header contains a single entry indicating the value to which the source is converted. For example, a table converting 10-bit logarithmic values to 8-bit linear would contain 1024 entries, corresponding to the 0–1023 intensity range of pixels in the source file. Each of these entries would be in the range 0–255, corresponding to the intensity range in the destination.

Blank lines and comment lines (starting with a number sign [#]) are ignored. Comment lines are useful for indicating the end of one table and the beginning of another, or for describing how the script or program works.

Floating Point 1D LUT File Format

Floating point LUTs are supported and are reversely compatible in most cases. You can specify your own floating-point 1D LUT using an ASCII editor as long as it is in the correct format and is named correctly.

The following illustration represents a 1D floating-point LUT that consists of one channel of five values that fall between the range of 0.0 and 2.0.



Applying a 1D LUT

LUTs are typically stored in a folder within the application or on a network, you should store LUTs created externally in the same folder for ease of use.

Three formats are supported:

- 8-bit unsigned integer
- 16-bit signed float
- 32-bit signed float

To apply a 1D LUT to an image:

1. From the Tools tab, drag the 1D LUT tool from the Color Correction folder to the dependency graph in the Schematic view.
2. In the 1D LUT tab, click the LUT Browser button. Locate the 1D LUT you want to use and drag it to the dependency graph in the Schematic view.
3. Select the Input Depth.

4. Set the Exposure if required.

   The 1D LUT is applied.

NOTE Exposure adjustment is an animatable attribute—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Modifying a 1D LUT

You can modify a 1D LUT by changing values in its ASCII file without having to reset the 1D LUT field.

To modify a 1D LUT:

1. Open the ASCII file of the 1D LUT and make the necessary modifications.
2. Save the LUT.
3. Click Refresh.

   The image is refreshed with the modified parameters.

NOTE If you want to modify a 1D LUT that is provided with the 1D LUT tool, save the modified 1D LUT under a different file name to preserve the original LUT as is.

Working with the Lustre Color 3D LUTs

Lustre Color LUTs provide a wide range of generic, out-of-the-box display and conversion capabilities. There are two main types of LUTs: display and conversion.

It is important to note that the LUTs were built for generic use in production, meaning that, while delivering an accurate print film rendition, it may not match that of a particular film lab. This is expected behavior, because each lab processes film differently. However, the LUTs use 3D cubes in order to ensure the best possible match between the film output and your display while taking into accounts its physical limitations. For example, a CRT monitor cannot physically represent all of the colors available for film.

LUTs are typically stored in a folder within the application or on a network. You should store LUTs created externally in the same folder for ease of use.

For complete information on the Lustre Color LUTs, refer to the *Lustre Color Management User's Guide* on the Composite Web Portal.

Applying a 3D Display LUT

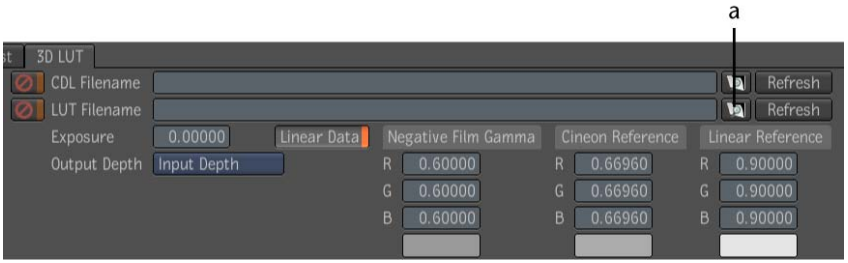
Display LUTs provide a simulation of the final output for a given film stock and display device without rendering. All display LUTs have an accompanying gamut warning LUT, which displays in magenta all out-of-gamut colors. That is, those colors that are not reproducible on the display device—see Applying a LUT or Color Conversion Tool to the Player on page 210.

Applying a 3D Conversion LUT

Conversion LUTs render in effects for a given film stock and display device in order to create a new clip for use in a subsequent step in the DI process. There are several types of conversion LUTs, and many of these offer a selection of mapping methods, so you can choose the type best suited to your footage.

To apply a 3D LUT:

1. From the Tools tab, drag the 3D LUT tool from the Color Correction folder to the dependency graph in the Schematic.
2. In the 3D LUT tab, click the LUT Browser button.



(a) LUT Browser button

The LUT Browser appears, displaying the Lustre_Color_3DLUTs folder.

3 Open the folder and select a LUT from the Conversion folder. Drag it to the dependency graph in the Schematic.

4 In the 3D LUT tab, set the Exposure, Negative Film Gamma, Cineon Reference, and Linear Reference—see Modifying Negative Film Gamma, Cineon Reference, or Linear Reference Levels on page 628.

The 3D LUT is applied.

NOTE Exposure is an animatable attribute—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Working with ASC CDLs

Composite supports the American Society of Cinematographers' Color Decision List (ASC CDL). The ASC CDL was developed by the American Society of Cinematographers in a unique collaboration between manufacturers, cinematographers, and colorists. The ASC CDL was developed to ensure that basic color grading adjustments can be shared amongst different manufacturer's systems, from on-set through post-production.

The American Society of Cinematographers CDL color specification defines a common language for primary color correction. A CDL is expressed by nine numbers (three RGBtriplets) representing predefined color functions: offset, slope, and power. The ASC has defined a number of formats in which a CDL may be carried. Composite supports the form where a single set of CDL parameters is stored in an XML file. If you are working on a project in which the cinematographer is making use of CDLs, you can use the 3D LUT tool as a display modifier to preview your work in the context of the cinematographer's

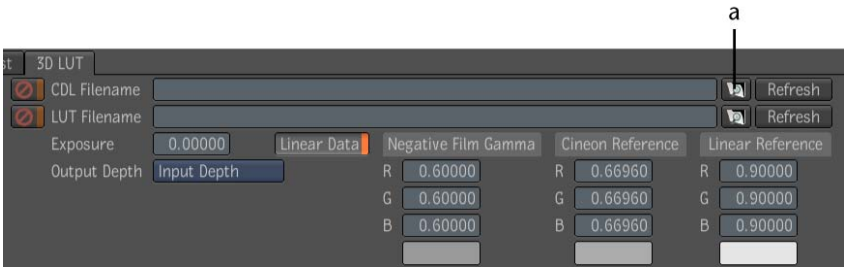
intended look—see Applying a LUT or Color Conversion Tool to the Player on page 210.

Within the3D LUT tool, the CDL is applied after the (optional) linear to log conversion but before the 3D LUT. Note that you may use the CDL even if you don't select a 3D LUT (and vice-versa).

The ASC CDL defines the color grading metadata, but it does not specify the color management that should be used with it. In order to get the best results, you should check with the provider of the CDLs to understand what color management was being used when they were created. Currently, most CDLs are used in the context of log images being previewed through a print-film simulation transform. If a custom 3D LUT is not provided to you, one of the 3D LUTs from the Lustre Color collection may get you close.

To apply a CDL:

1. From the Tools tab, drag the 3D LUT tool from the Color Correction folder to the dependency graph in the Schematic.
2. In the 3D LUT tab, click the CDL Browser button.



(a) CDL Browser button

3. In the Browser, open the folder that contains the CDL you want to use and select the CDL.

   The CDL is applied to your composition.

Broadcast Safe Tool

When working with material that is intended for television broadcast, the Broadcast Safe Colors tool ensures the images in your composites translate well (and safely) to a television display.

About Broadcast Safe Colors

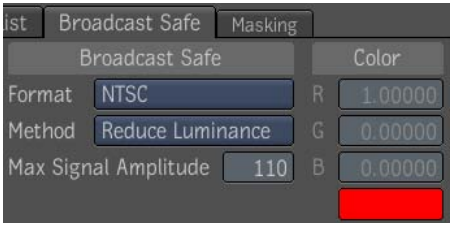
Since most computer monitors and television displays have different color ranges, the same image on a computer display can appear very different on an NTSC or PAL monitor.

Colors appear much brighter on a television display. This is because video uses a different gamma curve for luminance than a computer monitor. Also, most computer monitors use RGB color space, and television displays use YUV color space.

Some pure colors, such as red and yellow, can cause bloomed (very saturated, fuzzy) images. In some cases, colors can cause interference with broadcasting, which is referred to as NTSC illegal colors.

Broadcast Safe Color Controls

The Broadcast Safe Colors tool automatically reduces the luminance or saturation of portions of the image, or isolates unsafe or safe colors that you can then adjust manually.



Select:	To:
Format	Set the unsafe colors as NTSC or PAL.
Method	Determine how the unsafe colors will be fixed, or identify the colors so you can fix them.
Max Signal Amplitude	Adjust the video signal amplitude, where 120 is maximum, and 90 is the minimum. The default amplitude is 110.
Replacement Color	Click to select the replacement color from the color picker. Then click Set.

To use the Broadcast Safe tool:

1. From the Tools tab, drag the Broadcast Safe tool from the Color Correction folder to the dependency graph in the Schematic view.
2. From the Format list, select NTSC or PAL (your video format).
3. From the Method list, select one of the following:

Select:	To:
Reduce Luminance	Automatically reduce the luminance of the offending portions of the image to broadcast legal. Use Maximum Signal Amplitude to adjust the limits of the luminance. Note: The Maximum Signal Amplitude should be limited to 110.
Reduce Saturation	Automatically make any overly saturated portions of the image broadcast legal. Use the Maximum Signal Amplitude to adjust the limits of the saturation.
Key Out Unsafe	Cut out the unsafe colors from the image, leaving only the safe color regions.
Key Out Safe	Cut out the safe colors from the image, leaving only the unsafe color regions.
Replace Color	Fill the unsafe portion of the image with the selected color.

NOTE Use the Key Out Unsafe and Key Out Safe options to see any problem colors in your image immediately. For example, you can fix the problems in the Color Corrector by adding the output of the Broadcast Safe tool as the masking input to the Color Corrector.

Transforming Color Space with the Log and Delog Tools

Logarithmic data is automatically converted to linear data on import by applying a 1D LUT to the imported image. However, if you need to work in log space, you can modify both the display and image data by using the Log and Delog tools.

The Log and Delog tools let you transform between linear and log color space. The Log tool transforms from linear to log space and the Delog tool transforms from log to linear space.

The log space in Composite is very similar to that used by Cineon encoding, but it does not clamp high dynamic range values, and it is scaled with respect to the Cineon codes. That is, a Cineon code of 1023 corresponds to a value of 1.0.

The transformations can be controlled with the negative film gamma, the Cineon reference, and the linear reference. These reference points are expressed in units of the log space described above. You can display their value either as raw or as Cineon codes.

Modifying Negative Film Gamma, Cineon Reference, or Linear Reference Levels

The transformation between log and linear color spaces is controlled by three parameters: negative film gamma, Cineon reference, and linear reference. Gamma controls the contrast, while Cineon reference and linear reference establish a correspondence between points in the two color spaces. The Cineon code, specified as the Cineon reference, is transformed to the value specified as the linear reference. For example, you may want to base the transformation on the 90% white card and specify a Cineon reference code of 685 and a corresponding linear reference value of 90.

NOTE All Delog and Log parameters are animatable —see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

- The white reference is mapped to 0.9 in linear space. Cineon codes above reference white are mapped to values above 0.9, however, these values can be greater than 1. This tool does not clamp to 1, so there is no loss of information.
- All incoming pixels with values between 1 and 95 produce black pixels.

NOTE All channels are linked by default.

To uniformly modify values with the Log or Delog tool:

➤ Do one of the following:

- Drag the red (R), green (G), or blue (B) channel fields to the right to increase, or to the left to decrease, values uniformly on all three channels. Alternatively, you can also drag the luminance (L) field to achieve the same result.

- Select a single channel that you want to modify by deselecting the Lock button.
  Drag the red channel field to the right to increase, or to the left to decrease, the level of the red channel.

  The green and blue channels are not affected. However, the overall luminance field updates to reflect the change.

NOTE You may also type values into the channel fields and press **Enter**. In the case of modifying all channels uniformly, when values are typed into any field, the other fields update. The luminance field will update regardless of which method is used to modify the values.

Using the Trackball to Modify Negative Film Gamma, Cineon Reference and Linear Reference

The trackball allows you to modify levels with a more freestyle, intuitive approach. The trackball emulates a typical 360-degree color wheel that can be dynamically changed by dragging the center of the color wheel. Consequently, the effect on the red, blue, and green channels is predictable.

For example, dragging the trackball towards the red portion of the color wheel increases the value of the red channel, but decreases the values of both the blue and green channels, and adds blue and green to the shadows and midtones of the image.

Modifications made using the trackball are cumulative; each movement of the trackball is added to the previous one. The trackball changes color to reflect the degree of change.

To modify levels with the trackball:

➤ Drag the trackball to modify levels in the image.

NOTE To reset the trackball, channel, and luminance levels, select the Reset button below the trackball or the Reset button in Tool Options area on the right side of the UI (this button resets the entire tool).

Color Correcting with CC Basics

The CC Basics UI contains the controls used to perform color correction. CC Basics lets you:

- Color correct shadows, midtones, and highlights separately or together
- Rewire the red, green, or blue channels of an image
- Set the hue, saturation, or contrast of an image
- Set the color balance
- Set the red, green, blue, or RGB channels by modifying the gamma, gain, and offset values for the channel
- Match colors
- Perform color suppression

NOTE Many of the CC Basics fields are animatable attributes—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Determining the Sequence of Corrections

After you have analyzed the image, it is important to determine what sequence to use to make the color corrections. Some corrections may interfere with previous adjustments that you have made, or change results that were satisfactory. In general, you should not set saturation before applying modifications to the hue, as hue shift affects the saturation value.

For predictable results, perform color corrections in the following sequence:

- Correct the hue color balance
- Correct the light and dark areas of the image using gamma, gain, offset, and contrast

■ Correct the saturation

Color Correcting Shadows, Midtones, and Highlights

When you modify saturation, gamma, gain, offset, and contrast, you can color correct the shadows, midtones, and highlights ranges in the image individually or all together.

To color correct shadows, midtones, and highlights:

➤ Under Ranges, select a luma range to modify.

Click:	To Modify:
Master	The entire image. Modifications made in the Master range are applied after individual range modifications.
Shadows	Dark areas in the image.
Midtones	Midlevel areas in the image.
Highlights	Light areas in the image.

Rewiring Controls

Rewiring color channels redirects the information contained in one channel to another, and in some cases, makes adjustments to that information.

The following modes are available from the Rewire list.

Select:	To:
Normal	Return the footage to its original values.
Mono	Create a monochrome (black and white or grayscale) version of the image.
Negative	Create a negative version of the image.
Custom	Redirect information from one color channel to another.

The following rewiring options are available.

Select:	To:
R → R	Use color values for the red channel.
R → G	Replace color values of the red channel with those of the green channel.
R → B	Replace color values of the red channel with those of the blue channel.
R → Y	Replace color values of the red channel with the luma of all channels.
R → 1 - R	Replace color values of the red channel with its inverse. For example, 1 corresponds to the maximum value of the given color channel. In 8 bits, this value is 255.

NOTE The green and blue channels are rewired in the same way as the red channel (described in the previous table).

To rewire a color channel:

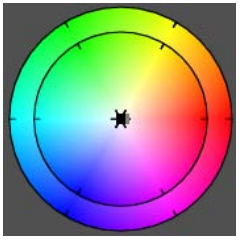
1. Select a rewire mode from the Rewire list.
2. Select a rewiring option.

   You can redirect the information from one channel into any of the other channels. You can also direct a pixel's luma value into a component, or make a negative of a channel's information.

The Color Wheel

The color wheel acts as a reference and monitoring guide or template. You can see how the colors that are 180 degrees apart are complements, and how the RGB and CMYK points are distributed around the wheel.

Because RGB is an additive color model, magenta, cyan, and yellow are expressed as combinations of two other colors.



The default display of the color wheel is the following:

- Red is at 3 o'clock.
- Blue is at 7 o'clock.
- Green is at 11 o'clock.
- Magenta (red plus blue) is at 5 o'clock.
- Cyan (green plus blue) is at 9 o'clock.
- Yellow (green plus red) is at 1 o'clock.

Setting Hue Shift

Shifting the hue changes the colors in the image.

You can use hue shift to compensate for material that is too hot or too cool, or simply to correct undesired tones present in the images. Note that in a monochrome or very low saturation image, a hue shift produces no results, since there is no hue to begin with.

Since you are working with a circle, a hue shift value of 360 returns to the original color or image. A hue shift of 180 is equivalent to a negative, as the colors are at their complements. The default for the Hue parameter is 0.

To apply a hue shift:

➤ Select the range that you want to modify, then do one of the following:

- Set the Hue Shift field by either dragging inside the field or typing in a value.
- Place your mouse on the outer ring of the color wheel and drag in a circular motion.

Setting Saturation

Saturation indicates how concentrated the color is: a saturated color has very little gray, while a desaturated one is dampened with more gray. For example, a pastel blue has a very low saturation level, while a blue screen is heavily saturated. A scarlet red has a high saturation, while a pale pink has a low one.

To set the saturation:

➤ Set the Saturation field by either dragging inside the field or typing in a value.

Setting Contrast

Contrast indicates the difference between the light and dark colors in an image. A high-contrast image is mostly black and white, with very little gray. A low-contrast image is very flat, foggy, and composed mostly of midtones.

To set the contrast:

➤ Set the Contrast field by either dragging inside the field or typing in a value.

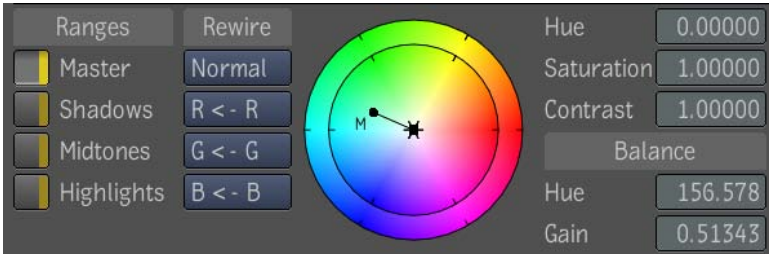
Color Balancing with Hue and Gain

The two parameters used with color balancing are hue and gain. The value of hue determines the color to add and the value of gain specifies the amount of color to use.

To color balance with hue and gain:

➤ Do one of the following:

- Set the Hue and Gain fields by either dragging inside the field or typing in a value.
- Drag the center of the wheel. Repeat to set a different range (shadows, midtones, or highlights) of colors.



Pure red is the 0-degree point for hue on the color wheel. When you increase the hue value, you move counterclockwise on the color wheel. When you decrease the hue, you move clockwise on the color wheel.

The center of the color wheel represents 0 gain. As you increase the value of gain, you move towards the edge of the color wheel and add more of the selected color to the image.

NOTE If saturation is set to 0, no color balancing is performed, regardless of the gain value.

Quick Gain

The Quick Gain controls are used to set the Gain in three specific ways:

NOTE The main Gain value is applied after the other Gain fields, so changes to main Gain do not change the separate channel Gain fields or the Quick Gain fields.

Modify:	To:
Temp	Set gain using a scale from warm (-100) to cool (100), which results in orange to cyan. It affects the Red and Blue Gain fields.
M/G	Set gain using a scale from green (-100) to magenta (100). It affects the Red, Green, and Blue Gain fields.
Value	Set gain using a scale from dark (0) to bright (200). It affects the Red, Green, and Blue Gain fields.

To modify the image using the Quick Gain controls:

➤ Drag the field to set the gain. You may also type values into the field and press **Enter**.

NOTE You can reset all fields simultaneously by clicking Reset in the Tool Options area located on the right side of the UI.

Setting the Gamma

You can set the gray values of an image by setting the gamma to brighten or darken an image without greatly affecting the shadows or highlights.

To set the gamma of the image:

1 Select the tonal range that you want to modify.

2 Drag the Gamma field to change the value (you can also type values into the field). Lower the gamma value to increase the contrast or raise the gamma value to decrease the contrast.

Setting the Gain and Offset

You can boost the colors in an image by increasing the gain and offset values. Gain and Offset controls multiply the pixel color values by the gain and then adds the offset value. The resulting color values are clipped at the maximum color value of 255 in 8-bit mode, or 1 in floating point mode.

You can also reduce the colors in the image by decreasing the gain and offset values. The resulting color values are clipped at the minimum value of 0.

Gain is expressed as a percentage value. The default value of 100% has no effect on the image since the color values are multiplied by 1.

NOTE You can set the gain for individual ranges, or across all ranges simultaneously (by using the Gain fields in the global controls).

To boost or reduce the colors in the image:

1 Select the range that you want to modify.

2 Drag the Gain field for the color channel that you want to modify (you may also type values into the field).

3 Drag the Offset field for the color channel that you want to modify (you may also type values into the field).

Suppressing Colors

Unwanted color can be caused by factors such as inconsistent lighting conditions during a shoot. This can result in images that contain unnatural

looking colors or one predominant color, which gives the image an unwanted color cast. You can dampen the channel information for one or more channels. It is available for all channels.

To suppress a color or channel:

➤ Under Suppress, disable the color or channel by selecting it.

NOTE Suppressing blue may result in a subtle overall pale yellow cast, since yellow is the spectral complement of blue. The effect of suppressing any color will unsuppress its complement.

Matching Colors

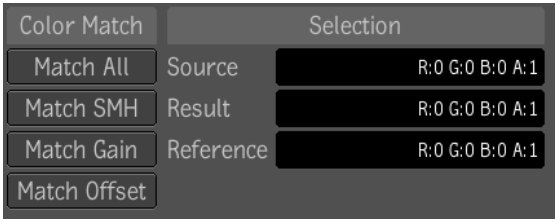
There are four options available for transferring the color value settings from one image to another. One color match option will almost certainly work better than all others for each particular shot, but it may take trial and error to determine which option works best.

Not all color match options use all sample values. Match All and Match SMH (Shadows, Midtones, Highlights) use all of the four pairs, but Match Gain and Match Offset only match the Master pair of colors. In this second case, the gamma, gain, and offset of the Master range is changed to perform the match.

To match a source image to a reference image:

1. Under Color Match, click a tonal range.
2. **Ctrl** + click in the Source field and use the eyedropper to sample a pixel in the Player for that tonal range.

   The sampled color is displayed in the Source field and stored as the source value for the selected tonal range.



3 **Ctrl** + click in the Reference field to invoke the eyedropper and sample a pixel in the player for that tonal range.

The sampled color is displayed in the Reference field and stored as the value for the selected tonal range.

4 To match other tonal ranges, repeat the preceding steps.

You may want to match the Master, or you may want to match just the Shadows, Midtones, the Highlights, or any combination thereof.

5 Select one of the following Color Match options.

Select:	To:
Match All	Match RGB values; match gamma, gain, and offset; and match shadows, midtones, and highlights from source to reference.
Match SMH	Match shadows, midtones, and highlights from source to reference.
Match Gain	Apply gain to the source to match it to the reference of the master color sample pair.
Match Offset	Apply Offset to the source to match it to the reference of the Master sample pair.

The Result field updates to reflect the changes made to match the color.

CC Histogram

The Histogram controls help you visualize how the individual pixel RGB values are distributed across the image.

The histogram is a visual representation of all the color values contained in the image as they are distributed over the range of pixel values (0–255). The color values depend on the bit depth (255 is the highest value for 8-bit).

You can view histograms for the red, green, or blue channels, or you can view the histogram for the entire image (RGB).

NOTE If vertical lines are too small to be viewed, you can zoom in by placing the cursor in the histogram viewer and then using the Up arrow and Down arrow keys.

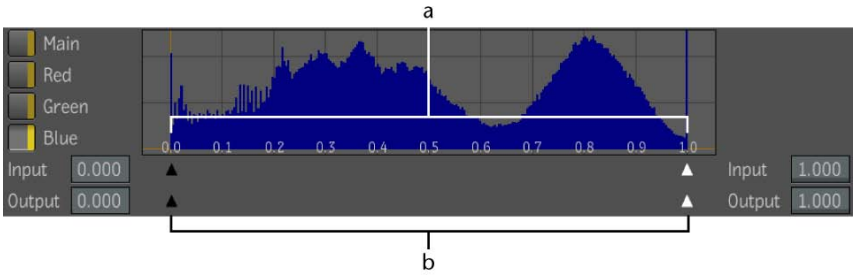
CC Histogram Controls

The Color Correct Histogram UI displays the input and output level adjustments for the selected tonal range and channel, as well as the histogram for the selected channel. The Ranges tab also displays the histogram for the selected channel, as well as the Shadows, Midtones, and Highlights curves.

The four Tonal Ranges selectors select the tonal range to operate on, namely Master (all pixels), Shadows, Midtones, and Highlights.

The four channel selectors select the channel to operate on, namely luminance labelled Main, Red, Green, and Blue.

There is one set of input and output levels adjustments for each channel of each tonal range for a total of sixteen sets of input and output levels adjustments.



(a) Input (b) Output

NOTE The CC Histogram's input and output level controls' fields are animatable attributes—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Input Sliders

The Input sliders below the histogram viewer are used to control the range of input color values in the image. The white slider on the right sets the maximum value for the range. The black slider on the left sets the minimum value for the range.

The histogram shown in the main tab is that of the selected channel creating a total of four possible histograms. The histogram background color matches that of the selected channel: gray for luminance, red, green, or blue.

All main tab values are shown in the range [0 to 1].

The input slider controls the values that are clamped to 0 (below the minimum) and to 1 (above the maximum). Values in between are scaled from 0 to 1. You can also use this to increase contrast.

You can set the maximum and minimum limits for the color range by entering the values in the Input fields on either side of the histogram.

NOTE Input levels increase contrast (remap more grays to blacks and whites), while output levels decrease contrast (remap more blacks and whites to grays). If you have an image that requires some softening of color or tone, the output levels re-introduce some midtones to the image.

To increase contrast with the input sliders:

1. From the Tools tab, drag the CC Histogram tool from the Color Correction folder to the dependency graph in the Schematic view.

2 Select Master, Shadows, Midtones, or Highlights to select the areas of the image that you want to modify.

3 Select Main, Red, Green, or Blue to select the channel that you want to work with.

4 Do one of the following:

- Position the cursor on the black slider below the histogram. Set the level until the value that you want to use as the lower limit for the color range appears in the Minimum field.
  This darkens the shadow areas.

- Position the cursor on the white slider below the histogram. Set the level until the value that you want to use as the upper limit for the color range appears in the Maximum field.
  The brightness in the highlight areas increases.

5 To cancel the changes, select Reset in Tools Options.

Output Sliders

The sliders on the Output levels bar below the histogram contain the Output level controls. Use these controls to set the range of colors in the image.

The white slider on the right sets the maximum value for the range. The black slider on the left sets the minimum value for the range.

The Output slider remaps pixel values produced by the input level. Values at 0 are remapped to the minimum, values at 1 are remapped to the maximum, and values in between are linearly interpolated. The output slider background is a linear ramp of the same color as the selected channel: gray, red, green, or blue.

You can also set the maximum and minimum limits for the color range by entering the values in the Output fields on either side of the histogram.

To reduce contrast with the Output sliders:

1 From the Tools tab, drag the CC Histogram from the Color Correction folder to the dependency graph in the Schematic view.

2 Select Master, Shadows, Midtones, or Highlights to select the areas of the image you want to modify.

3 Select Main, Red, Green, or Blue to select the channel that you want to work with.

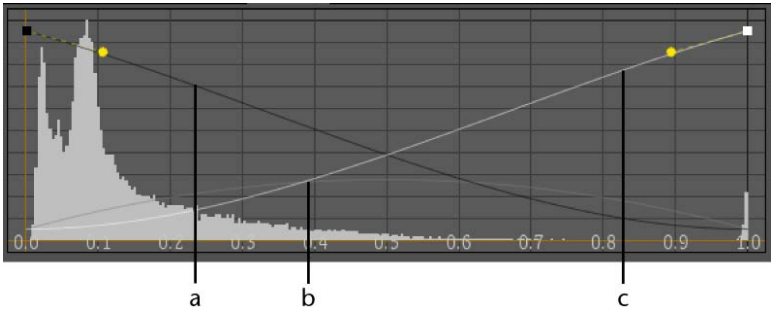
4 Do one of the following:

- Position the cursor on the black slider below the Output levels bar. Set the level until the value that you want to use as the lower limit for the color range appears in the Minimum field.
  The shadow areas are lightened.
- Position the cursor on the white slider below the Output levels bar. Set the level until the value that you want to use as the upper limit for the color range appears in the Maximum field.
  The brightness in the highlight areas is decreased.

5 To cancel the changes, select Reset in Tools Options.

Ranges Controls

When using a CC Basics or CC Histo tool, you can redefine what constitutes a shadow, midtone, or highlight.



(a) Shadows curve (b) Midtones curve (c) Highlights curve

To display the curves for the three tonal ranges:

➤ Select the Ranges tab.

To redefine shadows, highlights, or midtones:

1 Select a Display parameter from the drop-down menu.

2 Drag the tangent handle connected to the curve you have selected to modify.

To see the effects of the curves on color balance:

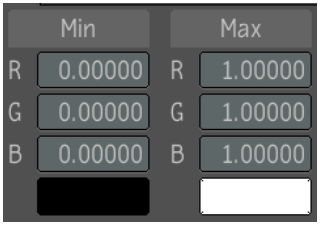
1. Open the CC Basics UI.
2. Under Balance, adjust the Hue and Gain to set the color balance for each of the Shadow, Midtone, and Highlight ranges.
3. Go back to the Ranges controls and set the curves.
4. Go back to the CC Basics controls. Without changing the color balance setup, note that the resulting image is different from that in step 1.

   The difference is the result of the changes that were made to the curves of the shadows, midtones, and highlights.

Clamp Color Tool

The Clamp Color tool lets you clamp colors that are outside a given color gamut. This is useful when you want to clamp an HDR image before using it with certain esoteric blend modes in a composite or when you want to clamp negative color components before using other color correction tools. Most of the time, you will want to clamp colors against the conventional [0,1] range, so this is the default behavior of the tool. This tool is an image modifier; it can be masked and muted and can only affect the RGB channels.

The Clamp color tool has the following parameters:



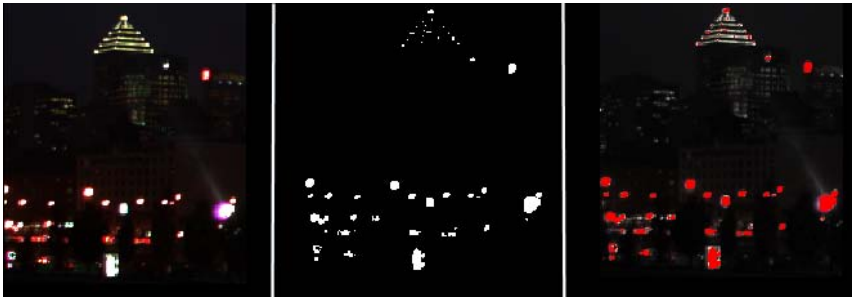
Use:	To:
Min	Set the minimum color values in the image to be clamped.
Max	Set maximum color values in the image to be clamped.

As an aid in visualizing which pixels are affected by its operation, this tool has two secondary outputs: It generates a one-channel image (a mask) where all out of range pixels are set to one and the rest are set to zero. It generates a pseudo-color image (a map) where all pixels that are below the range are

represented in blue, all the ones above the range in red, and the ones within the range are desaturated, almost turned to grayscale, to provide context and better locate the out of range pixels with respect to image features.

To view the Clamped Mask or Clamped Map:

➤ Normally, you will not connect the secondary outputs to other tools, but you may want to look at them in the Player by cycling through the tool outputs by pressing the **6** hotkey.



Original image

NOTE Secondary outputs can be neither masked nor muted.

Color Space Tool

The Color Space tool lets you convert to and from a variety of common color spaces, as well as invert the color space conversion without having to change selections in the source and destination space menus.

The Color Space tool covers all transformations between the following color spaces:

- RGB (REC 709)

- CMY
- HSV
- HLS
- Y'CbCr (YUV) 709
- Y'CbCr (YUV) 601
- YIQ
- CIE XYZ
- CIE Yxy
- CIE L*a*b*
- CIE L*u*v*

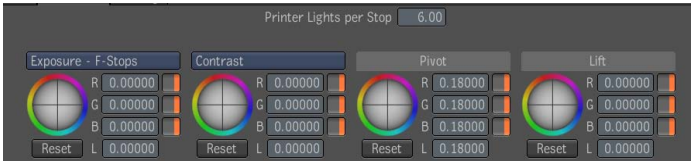
Processing Images with Photo Lab

The Photo Lab tool lets you simulate camera exposure and photo development changes in images. You can change the exposure to brighten or darken an image in incremental steps, providing perceptually relative uniform changes in luminance. Photo development adjustments can produce images with different color distribution.

Photo Lab Processing

With the Photo Lab tool, you can set the exposure, gamma, pivot, and lift of each color channel independently and in a variety of units, such as F-stops or printer lights for exposure.

The input image is first subjected to a gain adjustment, then to a gamma correction around a pivot point, and finally to a lift. Each step is optional.



NOTE Many of the Photo Lab tool's fields are animatable attributes—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Setting Exposure, Contrast, Pivot Point, and Lift of an Image

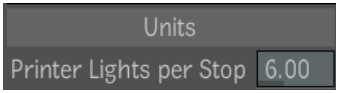
- **Exposure** Exposure can be expressed in terms of F-stops, printer lights, or gain. As with cameras, increasing the exposure by one F-stop results in an image with the luminance value that is double that of the original. Printer light units work very much like F-stops, but it usually takes six printer light units (default setting) to get the same effect that you get if you set exposure by one F-stop. If you are trying to simulate a calibrated printing device for which you have accurate measurements, Photo Lab allows you to specify how many printer lights there are in one F-stop. Increasing the exposure by one F-stop also increases the gain by a factor of two.
- **Contrast and Pivot** Gamma correction is applied about a pivot value. In other words, pixel values equal to the pivot value are left unchanged. This is useful when you want to use gamma correction to change the contrast of an image, but do not want to affect a particular luminance level. The amount of gamma correction can be expressed in terms of contrast or in terms of the usual gamma exponent. Increasing the gamma exponent decreases contrast.
- **Lift** The lift is simply a uniform offset added to all the pixel values (different offsets for different color components). Lift is usually applied as the last step of the process and can be used to control the overall brightness of the image.

NOTE This tool simulates the physical workings of camera exposures and printing devices as long as the media is encoded in a linear color space. All channels are linked by default.

Use the following procedures to set values for exposure, contrast, pivot point and lift. Note that menus are available for selecting exposure and contrast units.

To uniformly modify the levels on all channels or on a single channel of an image:

1. From the Tools tab, drag a Photo Lab tool from the Color Correction folder to the dependency graph in the Schematic view.
2. Select exposure and contrast units.
3. Set Printer Lights per F-stop by dragging the field to the right or to the left.



4. Do one of the following:
   - Drag any of the red (R), green (G), or blue (B) channel fields to the right to increase or to the left to decrease levels uniformly on all three channels. Alternatively you can also drag the overall luminance (L) field to achieve the same result.
   - Enter a value into any channel's field and press **Enter**.
   - Select a single channel that you want to modify by deselecting Lock, then drag the channel field to the right to increase, or to the left to decrease, the values of the channel. You may also enter a value into any channel's field and press **Enter**.
     The other two channel fields and the overall luminance field update to reflect the change.

Using the Trackball to Modify the Levels of an Image

The trackball allows you to modify values with a more freestyle, intuitive approach. The trackball emulates a typical 360-degree color wheel that can be dynamically changed by dragging the center of the color wheel. Consequently, the effect on the red, blue, and green channels is predictable.

For example, dragging the trackball towards the red portion of the color wheel increases the value of the red channel, but decreases the values of both the blue and green channels, and adds blue and green to the shadows and midtones of the image.

Modifications made using the trackball are cumulative; each movement of the trackball is added to the previous one. The trackball changes color to reflect the degree of change.

To modify the levels of an image using the trackball:

➤ Drag the trackball to modify levels in the image.

NOTE To reset the trackball, channel, and luminance levels, select the Reset button below the trackball or the Reset button in Tool Options area on the right side of the UI (the latter resets everything).

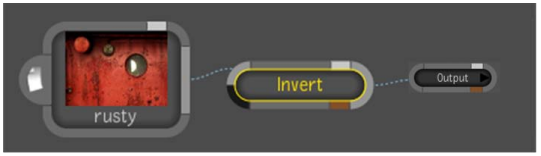
Inverting an Image

You can convert an image into a negative by applying the Invert tool.

NOTE You can invert just RGB, just Alpha, or RGB and Alpha, see the Affects menu in Tool Options.

To invert an image:

➤ From the Tools tab, drag the Invert tool from the Color Correction folder to the dependency graph in the Schematic view.



The image is inverted.

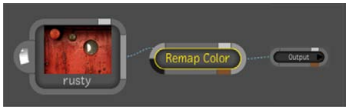
Remap Color

The Remap Color tool lets you view High Dynamic Range (HDR) images on a limited dynamic range display, such as a computer monitor. You can also use the Remap Color tool as a 1D LUT tool to remap an image's color output using interactive histogram and color curve tools.

For more information on 1D LUTs, see Applying LUTs on page 618.

To remap the color of an image:

1 From the Tools tab, drag the Remap Color tool from the Color Correction folder to the dependency graph in the Schematic view.



2 From the Remap Color UI, select a mode in which to remap the color:

- Linear: Maps a given range of the input values to the output interval. You can specify which luminance value in the input maps to black and which one maps to white. In this mode, you can click the Clamp button to have the curve defined by constant output values outside the level range. Otherwise, the straight line between black and white is interpolated for the entire range.
- EXRDisplay: Popular among Open EXR users, compresses a large portion of the high dynamic range into the gamut of the display without introducing clamping artifacts.

3 Under Output Depth, set the depth at which you want to output the image.

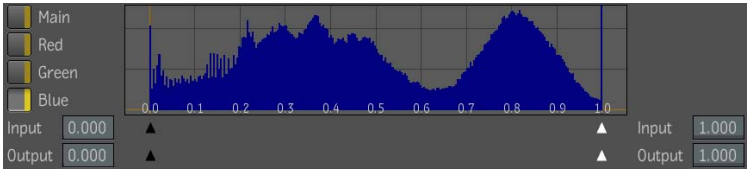
4 In the Remap Color UI, adjust the parameters for the mode you selected:

Mode:	Use:	To:
Linear	In Min and In Max	Set the minimum and maximum input color values in the image.
Out Min and Out Max	Set the minimum and maximum output color values in the image.	
Gamma	Set the gray values. Increase to brighten and	

Mode:	Use:	To:
	decrease to darken.	
EXRD-isplay	Exp	Set the exposure of the image. You can lighten or darken the image to reveal detail in the high or low end.
Defog	Compensate for fogging due to stray light in the recording device.	
Knee Low and Knee Hi	Set the white level of the image, which determines which value is mapped to the maximum intensity of the display.	
Gamma	Set the gray values. Increase to brighten and decrease to darken.	

Working with the Histogram

Controls on the left side of the histogram/curve viewer let you control how you want to view the histogram. The source histogram is white and the result histogram is black. By default both the source and result histogram are displayed in gray. You can view the source or result histogram, or a combination of the two at the current time.



NOTE The histogram is not updated during playback.

To frame the histogram:

➤ Under Histo View, click the Frame button.

To view the source or result histogram:

➤ Under Histo View, click the button beneath the Frame button and select Source or Result.

To zoom the histogram:

➤ Under Histo View, enter a value in the Zoom box.

To zoom the histogram/curve viewer:

➤ Do one of the following:

- To zoom in, press the Up Arrow key.
- To zoom out, press the Down Arrow key.
- To zoom in/out, press **Ctrl** + **Spacebar** and drag (for Windows and Linux) or Cmd + Spacebar (for Mac OS).

To pan the histogram/curve viewer:

➤ Press the Spacebar and drag.

Setting the Source Color

If you have a specific color in mind, you can set the source color by picking it from the remap color input image. Once you select a source color, the result color is automatically generated, and the orange color bar in the histogram/curve viewer moves to indicate where in the range the color exists. Then you can adjust the curve until you're satisfied. While you're doing this,

a dashed color bar indicates the changes you've made to the color, relative to the source color you set.

To set the source color:

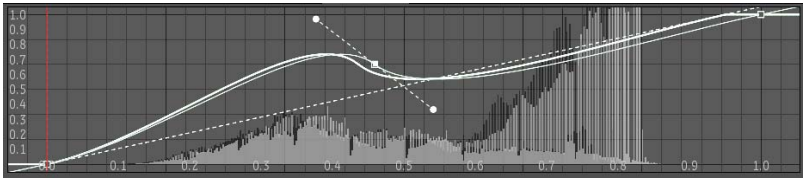
➤ Click the Source color pot and select a color from the image in the Player.

The result color is automatically generated.

Working with Curves

You can view and set values for the red, green, or blue histogram channels individually or for the curve representing the entire image (RGB).

When using the Remap Color tool, there are three curves displayed in the histogram/curve viewer: the basic curve, adjusted curve, and final curve. The basic curve is the result of the mode you selected and is displayed as a dashed line. The adjusted curve is the result of any changes you made to the curve itself; this is the only curve you can directly manipulate. It is displayed as a white line. The final curve is a combination of the basic and adjusted curves, and is displayed as a heavy white line. By default, the final curve is always displayed, but you can view the basic and adjusted curves individually.



To view individual R, G, or B curves:

➤ Under Curves, click the channel selection button (second button) and select Red, Green, or Blue.

To frame the curve:

➤ Under Curves, click the Frame button.

To adjust the curve:

1 Under Curves, click the curve selection button (third button) and select Adjust.

The Adjust curve is displayed.

2 Do any of the following:

- Click the control point in the middle of the curve and adjust the tangent handles.
- Add control points by pressing the Plus (+) sign and clicking the curve. Press the **Esc** or zero (**0**) key when you're done.
- Delete control points by pressing the Minus (-) sign and clicking the points to delete. Press the Esc or zero (0) key when you're done.

3 Click the Adjust button to mute the adjusted curve.

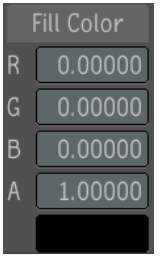
4 To reset the curve, click the Reset Adj. button.

TIP To reset the Remap Color tool, click the Reset button in the Tool UI Area.

Set Fill Color

This tool lets you set the fill color (RGBA) of an image without affecting any of the other characteristics of the image. This tool in an image modifier; since it only affects the fill color and does not change the pixels of the input image, this tool cannot be masked; it can be muted; it affects the color and alpha of the fill color only. This tool does not modify the pixel format or image resolution of its input.

The Set Fill Color tool has the following parameters:

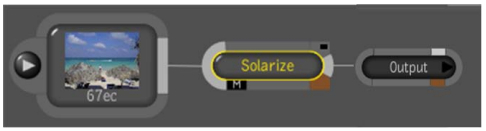


Solarizing an Image

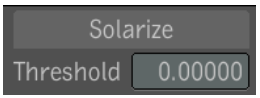
You can set the inversion threshold of an image to create solarizing effects.

To solarize an image:

1 From the Tools tab, drag the Solarize tool from the Color Correction folder to the dependency graph in the Schematic view.



The Solarize tool UI appears with the Threshold set to zero and the image inverted (by default).



2 Adjust the Threshold to change the amount of solarization.

Things to Remember

- You can reset the threshold field by selecting Reset in Tool Options.

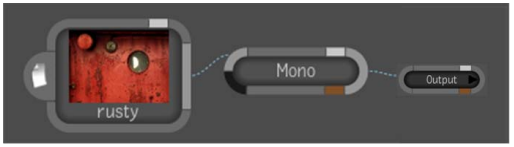
NOTE The Solarizing tool's Threshold field is an animatable attribute—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Creating a Monochrome Image

You can convert a color image into a monochrome image by using the Mono tool.

To create a monochrome image:

➤ From the Tools tab, drag the Mono tool from the Color Correction folder to the dependency graph in the Schematic view.



The image becomes monochrome.

Modifying a Display With the Pass Through Tool

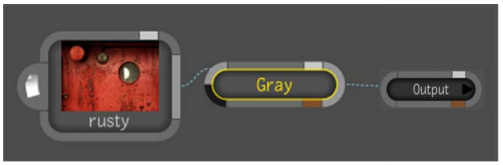
See Applying a LUT or Color Conversion Tool to the Player on page 210.

Setting the Amount of Gray in an Image

You can set the amount of gray in a color image.

To set the amount of gray in an image:

1 From the Tools tab, drag the Gray tool from the Color Correction folder to the dependency graph in the Schematic view.



The Gray tool UI appears with the Gray Amount field at its default of 100%.

2 Adjust the Amount field to change the amount of gray in the image.

Things to Remember

- You can reset the Amount field by selecting Reset in Tool Options.

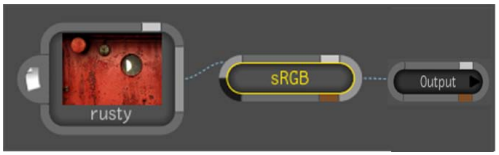
NOTE The Gray tool's Amount field is an animatable attribute—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Converting an Image to sRGB

Use the sRGB tool to convert an image into sRGB color space.

To convert an image into sRGB color space:

1 From the Tools tab, drag the sRGB tool from the Color Correction folder to the dependency graph in the Schematic view.



2 Select the output depth and set the exposure.

NOTE The sRGB tool's Exposure field is an animatable attribute—see Setting Keys Manually on page 676 and Validating and Applying the Expression String on page 729.

Animation

27

Topics in this chapter:

Animation Concepts

Simply defined, animation is a simulation of movement created by displaying a series of pictures, or frames. From one frame to the next certain values are changed. Almost all values can be animated. A value can be anything from the position, rotation, scaling, or transparency of an object, to the gamma, gain, or offset in a color correction.

Keyframe Animation

Keyframing is the simplest form of animating an object. It is based on the notion that an object has a beginning state, or condition, and changes over time in position, form, color, luminosity, or any other property to some different, final state. Keyframing takes the stance that we only need to show the keyframes or conditions that describe the transformation of the object, and that all other intermediate positions can be figured out from these—see Keyframing and Interpolation on page 660 and Setting Keyframes on page 660.

Keyframing and Interpolation

When you keyframe, you determine what an object looks like at specific points in time, while algorithms fill the frames in between the keyframes. This technique is called in-betweening. The intermediate values between the keyframes are computed by interpolation.

Extrapolation is used to determine the behavior of a channel before the first or after the last keyframe—see Modifying Interpolation on page 687.

Setting Keyframes

You can set keyframes for just about anything that has a value, including an object transform, visual attribute, as well as any tool attribute. When you set a keyframe to animate a particular parameter, a function curve is created. The curve is a graph that represents the animation of that parameter over time. You can edit the animation by editing its curve in Animation Editor or by modifying the attribute values in the Tool UI. You can set keyframes manually or automatically—see Setting Keys Manually on page 676 and Setting Keyframes Automatically on page 680.

The Animation Tab

The Animation tab is where you can edit animations in the tool UI. It is composed of three areas:

Use:	To:
Composition Browser	View tool parameters and set keyframes in the current animation—see Composition Browser on page 661.
Animation Editor	Display the animation curves and tracks for a composition—see Animation Editor on page 149.
Property Area	Set animation properties, views, and functions—see The Animation Property Area on page 666.

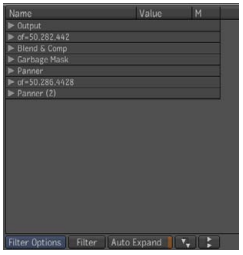
To access the Animation tab

➤ Select the Animation tab in the tool UI.

The Animation tab UI appears.

Composition Browser

The Composition Browser is the area where you select one or more specific curves to be displayed and edited in the timeline. It is located on the left of the Animation tab. It also exists as a standalone view, which can be accessed by swiping through the east gate (right) of the Gate UI.



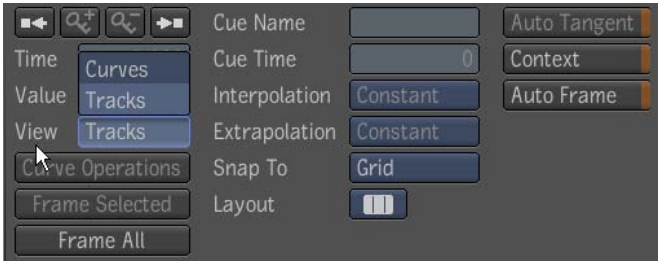
By default, the Composition Browser shows tools in the dependency graph.

You can also select other filter types from the Filter list.

Animation Editor

The Animation Editor allows you to work in either a curve view or a track view. Click the View button in the Animation Tab Property Area to choose one or the other. For further information on other options for the Animation Editor, see The Animation Property Area on page 666.

NOTE You can access the same options available in the Animation Property Area from the Animation tab in the Tool UI.

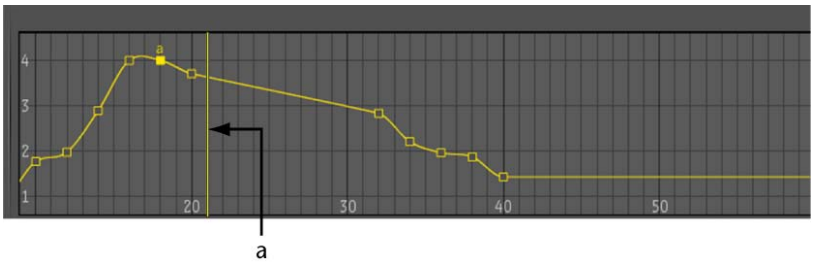


Curves View

Animation curves provide an accurate representation of a tool's relative behavior over time.

You can work with animation curves either from the Animation tab UI or in the Animation browser, which can be accessed by swiping east through the The Gate UI on page 12 .

The following graphic shows the Animation tab UI curves view.



(a) Time cursor

The time cursor is displayed as a vertical yellow line and can be moved along the time range by dragging it. The time cursor position represents the current frame. You can move to any frame by dragging the yellow current frame cursor to the new frame number.

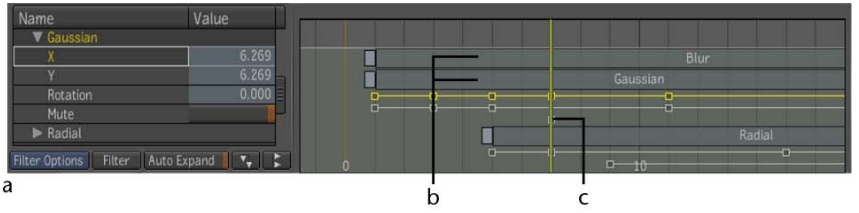
Tracks View

The tracks view allows you to see your work in overview mode. You can move, add, and delete keys but not edit their value.

Tracks are used to edit the attribute values using the value editor, which is located on the right side of the Tracks view (you must select the keyframe to edit).

In track mode, each channel displays all of its keyframe positions. Each channel group and each tool displays a track that corresponds to the start and end of all included channels.

For example, the following graphic shows the tracks view displaying a composition that includes the Blur tool. Five keyframes have been set on the Gaussian's X track and the track is selected.



(a) X channel of Gaussian Blur attribute in Composition Browser (b) Blur, Gaussian, and Radial tracks (c) Time cursor

You can move the contents of the Animation Editor using the following keys while the cursor is focused on the Animation Editor:

To pan:

➤ **Spacebar** + drag.

To zoom:

➤ Do one of the following:

- **Ctrl** + **Spacebar** + drag (Windows and Linux) or Cmd + Spacebar (Mac OS)
- **Up Arrow** to zoom in and **Down Arrow** to zoom out.

To resize the Animation Editor:

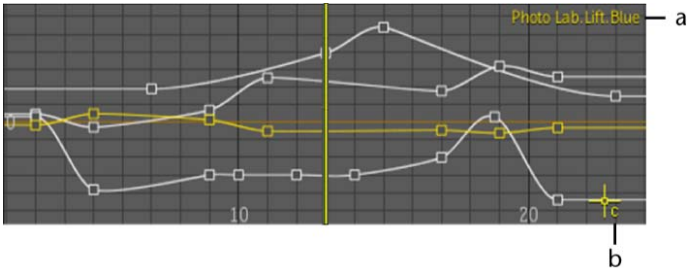
1. Select and drag the horizontal separator located just above the tool UI up or down.
2. Select and drag the separator located between the Animation Editor and Composition Browser to the left or right.
3. To reset a separator to its previous location, you can move it manually or press **Ctrl** + click.
4. You can also use your mouse wheel to resize the editor.

Animation Editor Feedback

To help simplify large and complex animations, the Animation Editor provides contextual roll-over feedback.

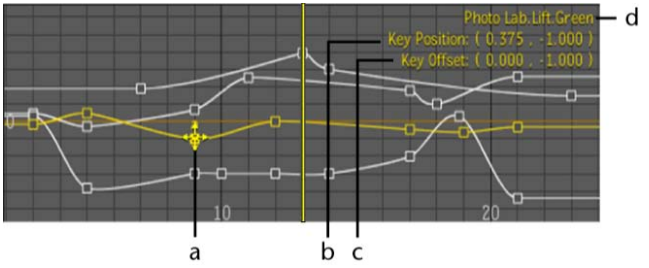
The following list displays the various roll-over feedback and their meanings.

- **Attribute name highlighting** When the cursor is positioned over a curve (selected or not), the name of the animated parameter is displayed and a C appears beside the cursor. This tells you what curve you are about to select or what input a particular curve is associated with.



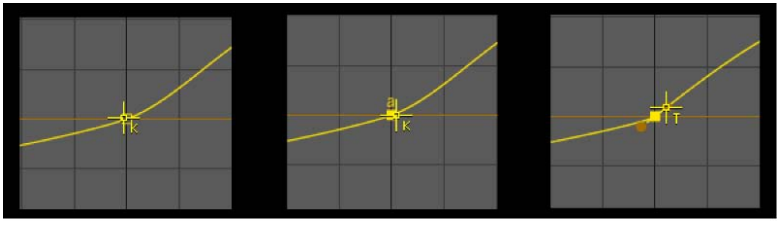
(a) Blue channel of Photolab's Lift attribute (b) Cursor on curve

- **Keyframe values highlighting** When you position the cursor over a keyframe, the keyframe coordinates are displayed beside the cursor, just under the name of the input. When you move a keyframe, the updated value is displayed, as well the translation offset from the beginning of the manipulation.



(a) Moved keyframe (b) Key position (c) Key offset (d) Attribute

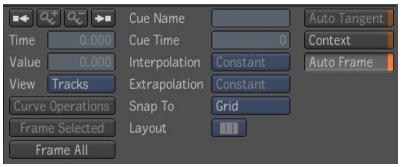
- **Keyframe feedback** Rolling over a keyframe in the Animation Editor will evoke different feedback depending on the state of the keyframe.



(a) Cursor over keyframe. (b) Cursor over keyframe with Auto tangent on, indicated by the lower-case "a". (c) Cursor over keyframe tangent with Auto tangent off. The orange-brown circle indicates the tangent handles.

The Animation Property Area

The Property Area is used to switch the display from curves to tracks, apply curve functions, and edit keyframe parameters.



The Property Area contains the following parameters.

Select:	To:
Auto Tangent	Toggle Auto Tangent on or off for selected keyframes when fine tuning a curve. When a keyframe is selected and Auto Tangent is on, a lowercase "a" appears on top of the keyframe. When Auto Tangent is off, tangents are displayed normally. You can toggle Auto Tangent by selecting the button or by pressing the T key.
Cue Name	Edit the selected cue mark's name.
Cue Time	Edit the selected cue mark's time (in frames).
Frame Selected	Frame and fit the selected keyframe(s) in the Animation Editor.
Frame All	Frame and fit all visible keyframes contained in the Animation Editor.
Curve Operations	Select from a list of math operations, enabling you to modify a keyframe or group of keyframes.
Layout	Select from three different layout options for the Animation Editor
Snap To	Enable Snapping mode for the keyframe when moving or creating it. ■ Grid: Snaps to the grid in Animation Editor. ■ Frame: Snaps the time value of keyframes to the closest frame. ■ Off: turns option off.
View	Switch between curve and track views—see Contextual Menus on page 670.

Select:	To:
Interpolation	Select from a list of extrapolation types to set the interpolation of selected keys.
Extrapolation	Select from a list of extrapolation types to define the shape of a curve after the selected keyframes.
	Move to previous keyframe.
	Move to next keyframe.
	Delete keyframe.
	Add keyframe.
Time	Modify the time (X axis of the graph) of the currently selected keyframe.
Value	Modify the value (Y axis of the graph) of the currently selected keyframe.

Player Controls

The Player Controls are located along the top of the Animation tab and contain several animation controls located on the far left.

Select:	To:
	Select a playback mode.
	Move to previous cue mark.
	Set a cue mark.
	Move to next cue mark.

Select:	To:
	Move to previous keyframe.
	Set keyframe.
	Move to next keyframe.
	Set keyframes automatically—see Setting Keyframes Automatically on page 680.

Working with Cue Marks

Cue marks are time-based points of interest with annotations. They function as high-level keyframe controls that let you associate a keyframe with a mark. Instead of viewing a clip over and over again in order to evaluate and record (usually on paper) important timing points, you can view footage and add cue marks.

Once you've set a cue mark, you can go back to the animation timeline and tweak the positions of the cue marks, give them names, and even add notes to them.

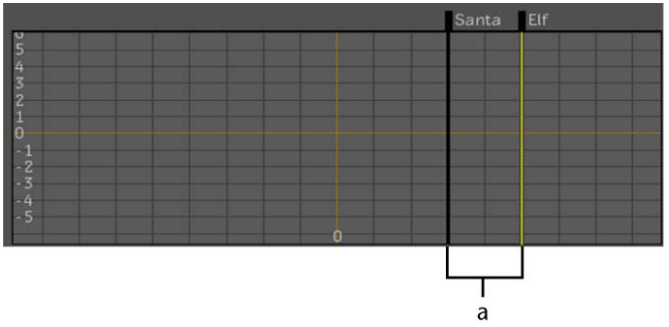
You can also set keyframes based on these marks by navigating from one mark to another using the Next Cue Mark and Previous Cue Mark buttons.

Setting keys links the temporal position of the keyframe to the cue mark at that time; as you move the cue marks, the keyframes follow. If you move a keyframe in time, the link to the cue mark is broken. However, by moving a keyframe to the exact time of a cue mark, you link the keyframe to that cue mark.

To add a cue mark:

➤ Select the Set Cue Mark button in the Player controls.

NOTE You can also set a cue mark by pressing the **U** key while your cursor is in an Animation Editor, or in the Player navigation bar's current frame counter.



(a) Cue marks in Animation Editor

To set a cue mark to a specific frame of a film clip:

1. Do one of the following:
   - Drag the time cursor in the Player navigation bar (or type in the frame number) to the frame on which you want to set a cue mark.
   - Play the composition, so that you can add cue marks during playback.
2. Select the Set Cue Mark button in the Player navigation controls (left side) or set the cue mark while the footage is playing by pressing the **U** key. Play-and-set is a more interactive method of adding cue marks.
3. Repeat steps 1 and 2 to add more cue marks.

NOTE You can also add a cue mark by right-clicking in any Animation Editor and selecting Add at Select Time or Add From Current Time from the Cue Marks sub-menu.

To delete a cue mark:

1. Select the Cue Marks tab in the tool UI.
2. Select a cue mark.
3. Right-click and select Delete.

To delete a cue mark using the contextual menu in any Animation Editor:

1. Select the cue mark you want to delete by selecting the top of it (it will turn from black to yellow when selected).
2. Right-click and select Delete (or Delete All) from the Cue Marks sub-menu.

Adding and Renaming Cue Marks

You can keep track of your cue marks by renaming them. There are several ways you can do this:

To rename to a cue mark using the Cue Marks tab:

1. Select the Cue Marks tab in the tool UI.

   The list of cue marks appears (by default, unnamed cue marks are numbered in succession).

2. Select the name field and press **F2** or right-click and select Edit.
3. Type in a name for the cue mark and press **Enter**.
4. Repeat steps 1 to 3 to add a description to or set a new frame for the cue mark.

To rename a cue mark using the Cue Name field in the Animation tab Properties area:

1. Select the existing name in the Cue Name field.

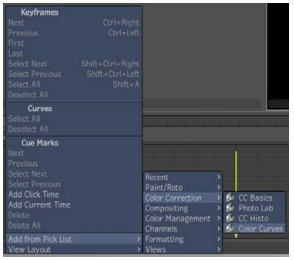
   NOTE The cue mark must be selected before the text edit is allowed.

2. Type in the new name.
3. Press **Enter**.

Contextual Menus

You can access many of the animation controls while in any Animation Editor by right-clicking and accessing a contextual menu.

You can access items stored in your Pick List.



Navigating the Animation Editor

Contextual navigation controls are available for both Keyframes and Cue Marks. They let you perform the following operations:

Select:	To:
Next (Keyframes)	Move the time cursor to the next keyframe among the selected curves.
Previous (Keyframes & Cue Marks)	Move the time cursor to the previous keyframe among the selected curves.
Next (Cue Marks)	Move the time cursor to the next cue mark.
Previous (Cue Marks)	Move the time cursor to the previous cue mark.
First (Keyframes)	Move the time cursor to the first keyframe among the selected curves.
Last (Keyframes)	Move the time cursor to the last keyframe among the selected curves.

Select:	To:
Select Next (Keyframes & Cue Marks)	Select next keyframe or curve.
Select Previous (Keyframes & Cue Marks)	Select previous keyframe or curve.
Select All (Keyframes & Curves)	Select all keyframes or curves.
Deselect All (Keyframes & Curves)	Deselect all keyframes and curves.
Add Click Time (Cue Marks)	Adds cue mark at the position that you click with your mouse.
Add Current Time (Cue Marks)	Adds cue mark at the current time.
Delete (Cue Marks)	Delete a specific cue mark.
Delete all (Cue Marks)	Delete all cue marks.

Local Controls

You can navigate through a set of keyframes using the Previous Keyframe and Next Keyframe buttons in the Tool Options located on the right side of the tool UI, Player controls tool bar, or by using the contextual navigation menu in any Animation Editor.

When using the Next Keyframe and Previous Keyframe buttons in the Tool Options, keyframe navigation is restricted to the keyframes set for that particular tool. However, when using the keyframe navigation controls in either the Player controls or Animation tab Property Area, you can navigate through keyframes of all marked tools attached to the dependency graph.

Selecting Keyframes

For the Contextual selection controls, see the Contextual navigation controls, in Navigating the Animation Editor on page 671.

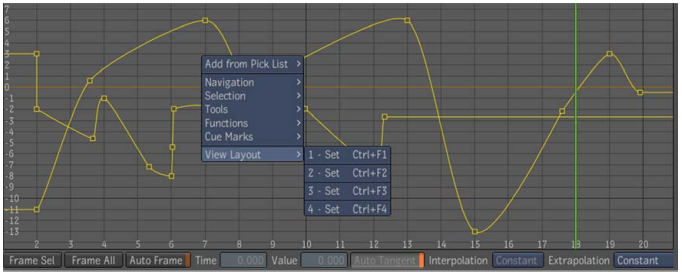
Setting Cue Marks

There are several options related to cue marks. For more information see, Navigating the Animation Editor on page 671.

NOTE The first two options are always available. The other options are only available when a cue mark is selected. The two navigation options are available when next or previous cue marks are available respectively.

Setting Zoom and Pan

Placing the mouse cursor over the View Layout sub-menu displays selections you can use to store up to four zoom and pan settings. This is useful when you want to jump back and forth between different points on an animation curve—see View Layouts on page 153.



To set a zoom and pan view:

1. Zoom and pan to the area of the animation curve you want to set as a view layout.
2. Right click in the Animation Editor view and select one of the four Set options from the View Layout sub-menu or press **Ctrl** + **F(1,2,3,4)**.
3. Repeat steps 1 and 2 to add more layout views (maximum 4).
4. Press **F (1,2,3,4)** to activate the view or select Activate from the View Layout sub-menu.

5 Press **Shift** + **Ctrl** + **F (1,2,3,4)** to delete the layout view or select Delete from the View Layout sub-menu.

Keyframing Workflows

The following is the workflow for setting animation keyframes:

- Mark the attributes for keyframing—see Marking Attributes for Keyframing on page 674.
- Set a keyframe by selecting the Set Keyframe button—see Setting Keys Manually on page 676.
- Setting keyframes automatically - see Setting Keyframes Automatically on page 680.
- See also Working with Cue Marks on page 668.

Marking Attributes for Keyframing

The workflow for keyframing using marking consists of the following:

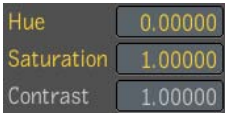
1. Selecting attributes to be marked.
2. Changing values.
3. Setting keyframes for all marked values.

So the first step in setting keyframes for animation is marking which attributes you wish to animate. Almost all attributes can be marked for keyframing. You can mark attributes for keyframing in either the tool UI, Composition Browser, or Animation tab UI.

To mark an attribute for keyframing in the Tool UI:

➤ Select the attribute label to turn marking on or off.

The label and value editor change to yellow, as in the following example where the Hue and Saturation of the CC Basics tool have been marked.



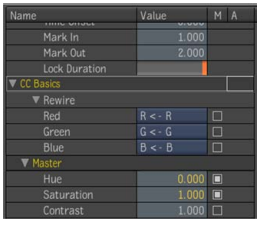
You can also mark attributes by right-clicking a value editor and choosing one of the following options:

Select:	To:
Mark Tool	Mark all animatable attribute labels of the current tool. Note that this is only available if no attribute is presently marked in the tool node.
Mark	Mark current attribute label.
Set Key	Set keyframe for current attribute value field—see Setting Keyframes on page 660.
Set Expression	Set expression for current attribute value field—see Validating and Applying the Expression String on page 729.
Reset	Reset the values to default state.

To mark an attribute for keyframing in the Composition Browser:

1. Expand the tool channel to the attribute level (in this example, the Hue and Saturation attribute of the Master range of the CC Basics tool).
2. Select the radio button of each attribute that you wish to mark in the M (mark) column.

   The Hue and Saturation attributes are now marked for keyframing.



You can also mark attributes in the Composition Browser by right-clicking an attribute field and choosing the same Mark and Mark Tool options provided in the tool UI.

To unmark an attribute in the Animation tab:

➤ Do one of the following:

- Click on the attribute label.

- Right-click the attribute field and select an unmarking option.

Select:	To:
Unmark All	Unmark all animatable attribute labels of all current tools in the dependency graph.
Unmark Tool	Unmark all animatable attribute labels of current tool.
Unmark	Unmark current animatable attribute label.

You can also unmark attributes in the Composition Browser by right-clicking over an attribute field where you will be given the same Unmark All and Unmark Tool and Unmark options provided in the tool UI.

Setting Keys Manually

You can set keyframes for any animatable parameter at any time, and in any order. When you add a keyframe, the interpolation between the previous and the next keyframe is computed automatically. Setting a keyframe over another keyframe replaces the old keyframe.You can set keyframes the following ways:

- The Keyframe button in the Player controls, which only sets keyframes on marked attributes.
- The Add Keyframe button in the Tool Options.
- The Add Keyframe button in the Animation Tab.
- The Set Keyframe option list when you right-click on an attribute's value field in the tool UI or in the Composition Browser.
- The Set Keyframe hotkey (**K**).

NOTE When you use the **K** hotkey in the Animation Editor, keyframes are added as follows:

- If none of the inputs of the selected node are selected, a keyframe is added at the current viewer's time for every input in the selected node (as if you selected every input in the tool).
- If at least one input is selected, keyframes are set on the selected inputs only.
- Every time a keyframe is set, there is no check for previous animation on inputs. This means that pressing the **K** hotkey will set keyframes on inputs even if the inputs were not already animated.

- The Set Keyframe for marked attributes hotkey combination (**Shift** + **K**).

NOTE The **Shift** + **K** hotkey has the same behavior as the Set Keyframe buttons mentioned above. It will set keyframes for all the marked attributes of the currently selected tool. The **K** hotkey, on the other hand, can only be used in an Animation Editor context.

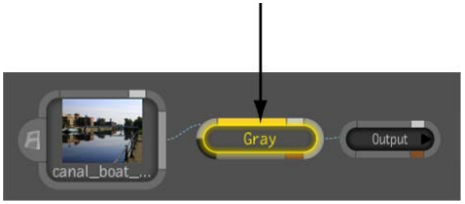
Example: Animating Using the Tool UI

In this example, you'll be creating a simple animation that would change the amount of gray in an image over the span of 96 frames. You'll be manually setting keyframes using the tool UI.

NOTE It is assumed that the composition is loaded and that the Gray tool has been added to the dependency graph.

To animate the amount of gray in an image using the Gray tool UI:

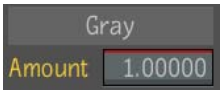
1. Click on the Gray tool and select the Amount attribute label, marking it for keyframing. A yellow rectangle appears on the tool node indicating that it has been marked for keyframing.



2 At the first frame leave the value at 1.00000.

3 Select the Set Keyframe button in the Tool Options (see Node-Specific Tabs on page 26) or the Player controls.

A green line appears at the top of the attribute label, indicating that the attribute is now animated and has a keyframe at the current frame.



4 Move the time cursor in the Player navigation bar to frame 30.

The green line changes to red to indicate the attribute is animated but has no keyframe at the current frame.

5 Change the value in the Amount field to .50000.

6 Select the Set Keyframe button.

The green line changes back to red. Again, this indicates that a new keyframe has been added to the timeline.

7 Repeat steps 3 to 6 as required.

8 Select the Gray attribute in the Composition browser.

9 Select Frame or Frame All to view the curves in a Animation Editor (or press **Home**).

NOTE You can also set keyframes in the tool UI by right-clicking an attribute field and choosing any of the options to mark attributes and set keyframes.

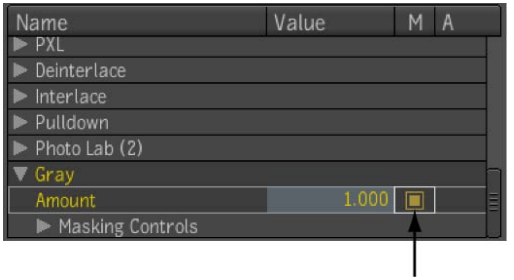
Example: Animating Using the Composition Browser

You can also use the Composition Browser to manually set keyframes. The following procedure uses the same example of adjusting the amount of gray in an image.

NOTE It is assumed that the film footage is loaded and that the Gray tool has been added to the dependency graph.

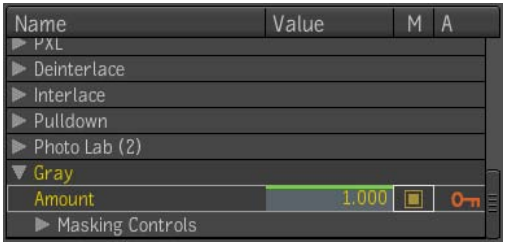
To animate the amount of gray in an image using the Gray tool in the Composition Browser:

1. Expand the Gray tool in the Composition Browser.
2. Mark the Amount attribute for keyframing by selecting its radio button in the M column.



3. At the first frame, leave the initial value as is.
4. Select the Set Keyframe button to set a keyframe on the Amount attribute.

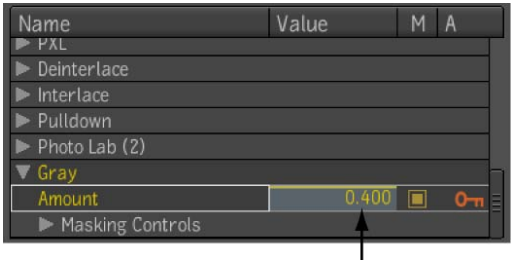
   A green line appears over the attribute's value, indicating a keyframe has been added for this attribute at that frame.



5. Move the time cursor in the Player navigation bar to frame 30.

NOTE When you move the time cursor to the next frame, you will notice that the green line turns red, indicating that the attribute is animated, but has no keyframe at the current frame.

6 Click in the value editor in the Composition Browser and either type the new amount of gray, then press Enter, or drag inside the value editor.



7 Repeat steps 4 to 6 as required.

8 Select Frame or Frame All to see the curves in the Animation Editor.

Setting Keyframes Automatically

You can set animation keyframes automatically each time you modify the attribute values. This process is called AutoKey.

AutoKey lets you to skip the steps of marking attributes and pressing a button or hotkey every time you want to set a keyframe. When AutoKey is on and you modify an attribute value in any way (using the tool UI, direct manipulation, or scripting) a keyframe for the modified attribute value is set at the current time, whether or not the attribute is marked for animation. The Autokey setting therefore overrides all markings for animation settings. This is also valid while playing and is usually done this way to define the initial shape of the animation.

To turn AutoKey on:

1 Select the AutoKey button.

You are now in AutoKey mode.

2 Select AutoKey again to turn off AutoKey mode.

AutoKey

Editing Keyframes in the Tool UI

Aside from modifying attribute values, you can also perform a variety of common operations by right-clicking in a value field, then selecting an operation from the menu. The following table lists these operations and describes their function:

Select:	To:
Copy frame	Copy the keyframe.
Cut frame	Cut the keyframe but maintain its information in the memory.
Paste frame	Paste the keyframe to another location on the timeline where there isn't already a key. This operation is available if a keyframe has been copied or cut.
Delete Keyrame	Remove the keyframe from the timeline.
First frame	Navigate to the first keyframe in the timeline.
Previous frame	Navigate to the previous keyframe.
Next frame	Navigate to the next keyframe.
Last frame	Navigate to the last keyframe in the timeline.
Copy Animation	Copy the animation of the selected channel.
Cut Animation	Remove the animation from the selected channel but retain its values in the clipboard.
Paste Animation	Paste the animation values that were placed in the clipbaord using copy or cut.
Remove Animation	Remove the animation from the attribute without retaining its information in the pasting buffer.

Editing Keyframes in the Animation Editor

The Animation Editor gives you precise control over keyframes and animation between keyframes. You can also move a keyframe in time in the Animation Editor.

To edit a keyframe in the Animation Editor in the Animation tab:

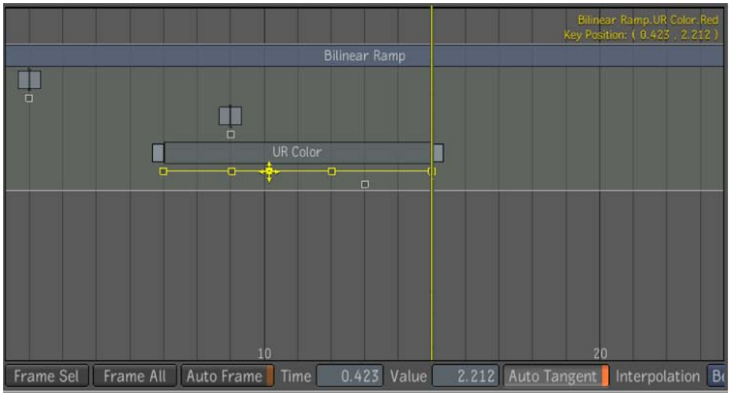
1. In the Property area, select View to Curves.
2. In the Composition Browser, select the channel whose keyframe you want to edit. When you select a channel, all its keyframes are displayed.

   NOTE To select several channels for editing, draw a selection box around the channels you want to select or **Ctrl** + click each one.

   The curve appears in the editor.
3. Select the keyframe(s) that you want to edit.
4. Select and drag a keyframe up or down to change its value. Drag left or right to move it in time.
5. Drag a tangent handle to change the shape of the curve at the keyframe when the Bezier interpolation is chosen.

To move a keyframe in time in the Animation Editor Tracks view:

➤ Select and drag the selected keyframe to the right or left.

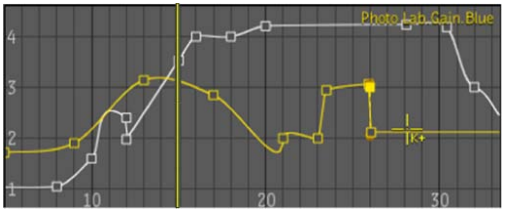


Adding and Deleting Keyframes

You can add or delete keyframes in a variety of ways when focused in a Animation Editor.

To add a single keyframe to an animation curve:

1. Press and hold the Alt key.
2. Place the cursor on the curve at the point where you want to add a keyframe.

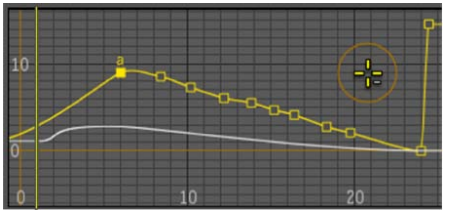


3. Click once.

   A single keyframe is added.

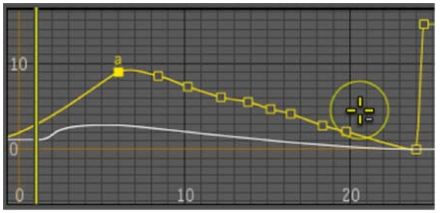
To delete a single keyframe from an animation curve:

1. Press and hold the Alt key.
2. Move the cursor over an existing keyframe. “K+” will be shown next to the cursor.
3. If needed, enlarge radius of eraser by pressing Ctrl + click and moving the cursor to the right (to increase) or to the left (to decrease).



4. Place the cursor on the curve at the point at which you want to remove the keyframe.

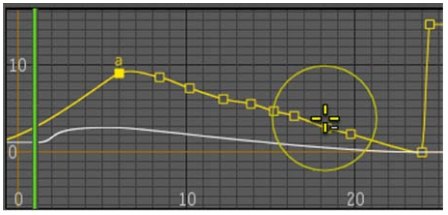
   Note that the cursor becomes active (turns from orange to yellow) when it comes in contact with a keyframe.



5 Click once.

The keyframe is deleted.

To remove more than one keyframe with a single mouse click, enlarge the cursor so that it encircles several keyframes.



6 Click once. All keyframes within the area of the cursor are deleted.

To quickly delete a series of keyframes along an animation curve:

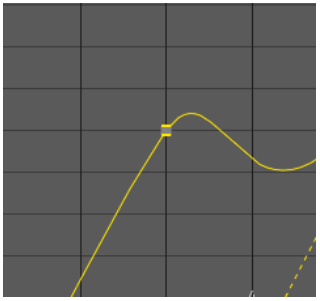
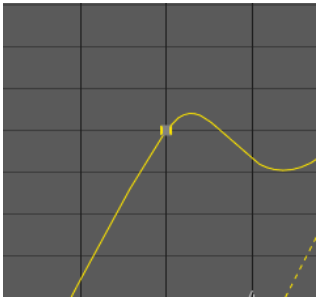
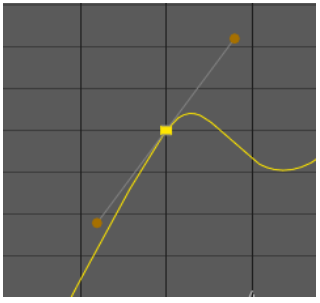
1 Press and hold Alt + Shift.

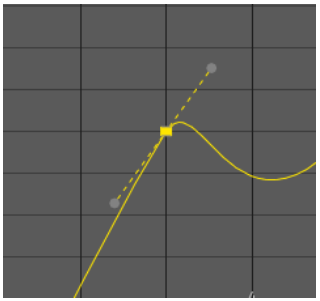
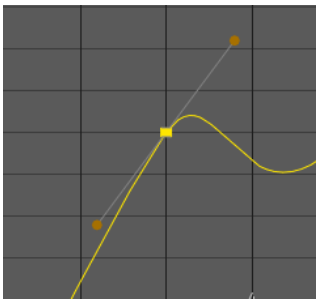
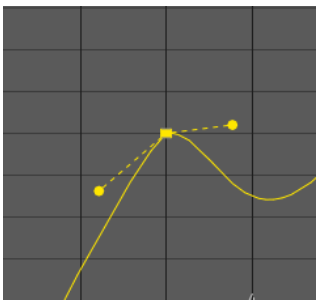
2 **D**rag the mouse along the curve.

The keyframes are deleted.

Precision Keyframe Editing

There are several precision keyframe editing controls available to you in the form of hotkeys. The following section lists the hotkeys, their function, and how the keyframe appears in the editor—see Animation Editor on page 662.

Hotkey and Function	Keyframe in Animation Editor
Shift + T Locks keyframe time	
Shift + V Locks keyframe value	
Shift + L Locks tangent length	

Hotkey and Function	Keyframe in Animation Editor
Shift + **O** Locks tangent orientation	
Ctrl + **Shift** + **L** (Windows and Linux) or **Cmd** + **Shift** + **L** (Mac OS) Unifies tangent length	
Ctrl + **Shift** + **O** (Windows and Linux) or **Cmd** + **Shift** + **O)** (Mac OS) Unifies tangent orientation	

NOTE To return the keyframe to its previous state, press the hotkey again.

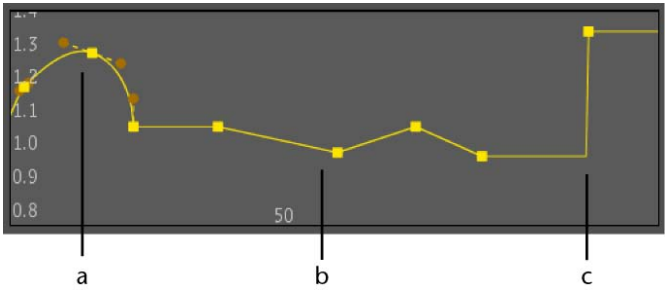
Other hotkeys are available for keyframing—see Hotkeys on page 817.

Modifying Interpolation

Once you create keyframes for a channel, you can use interpolation to determine the channel values for the in-between frames. The appearance of the animation in the Animation Editor depends on the interpolation method used. In the Animation Editor, the interpolation method is represented by the shape of the curve that joins the keyframes. You can set the interpolation for an entire curve or for individual keyframes.

The following interpolation methods are supported:

- Constant (key values do not change in time between keys, tangents are absent)
- Linear (straight interpolation between keys, tangents are absent)
- Bezier (spline interpolation, you can edit the tangent length and orientation)

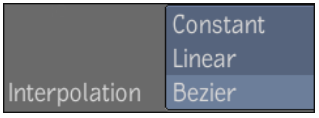


(a) Bezier (b) Linear (c) Constant

NOTE The default interpolation setting is Bezier.

To set the interpolation method for an individual keyframe or an entire curve using the Animation tab Property area or the Animation Editor view controls:

1. Select the keyframe(s) for which you want to change the interpolation.
2. Select Interpolation and select the interpolation method.



Modifying Extrapolation

Extrapolation is used to determine the behavior of a channel before the first, or after the last, keyframe. Extrapolation is useful when you want to create a cyclical effect, such as shutter flicker (as you would find in an old-time movie). You create one period in the flicker cycle, then use extrapolation to repeat the period over and over again. You will see the effects of extrapolation only if the film image has frames before the first keyframe or after the last keyframe.

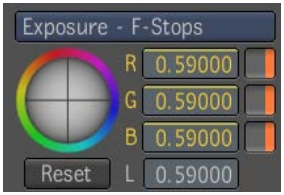
The following extrapolation types are supported:

- **Constant** Applies the value at the first or last keyframe to all the frames that come before or after the last or first keyframe.
- **Linear** Continues the curve in a linear fashion before the first keyframe or after the last keyframe using the tangent.
- **Loop** Cycles the animation as it appears between the first and last keyframe.
- **Ping Pong** Cycles the animation as it appears between the first and last keyframes forward then backward.
- **Relative** Cycles the animation as it appears between the first and last keyframe. This is similar to "Loop", but the relative offset is based on the channel value derived from either the first or last keyframe.
- **Key Linear** Continues the curve in a linear fashion before the first keyframe or after the last keyframe, disregarding all tangents.

Temporarily Modifying Attribute Values

Even if an attribute is animated, you can temporarily modify its value to see what the result would look like, without actually modifying the animation data. You can do this by using the tool UI or by direct manipulation when AutoKey is off. When you do so, the green or red bar across the top of the value field in the tool UI turns yellow.

NOTE To acknowledge the change, you must set the keyframe either using the keyframing controls with marked attributes or the context menu actions.



Modifying the current editing time (the time at which the editors are currently parked) in any way, clears the temporary modifications if the attribute is animated. You can change the editing time using the Player controls.

This is how existing applications usually deal with temporary modifications, but you can display the same film image at different times in two different players. If you are viewing the same image in two Players stationed at different times, then changing the focus from one Player to the other changes the current editing time, resulting in the loss of the temporary modifications, unless you have used the hotkey as an override.

Another difference with applications is the presence of time offsets which can result in multiple instances of the same object being processed. In this context, modifying attribute values can affect the attribute by modifying it at more than one time. This is dealt with by maintaining an attribute modifying curve, and not just a modifying value. These curves are not visible and are not displayed in the curve viewer.

Customizing the Layout when Working with Animation Curves

Once you are ready to work with animation curves, you may want to modify the layout.

A generic Animate UI Preset displaying a Animation Editor view, Composition Browser and Player view is available on the taskbar. However, you can modify this preset and save it, or create an entirely different preset consisting of whatever views you need —see Working with a Task-Based UI on page 140.

Global Time vs. Local Time

Global time is the time a Player, Schematic, or Animation Editor view uses. This time may be either master or standalone time. Remember, you can use master and standalone time to synchronize or separate playback between two or more views—see Playing Back in Multiple Views on page 192.

Local time becomes relevant when you add a Time Offset tool to a node of the composition. A Time Offset tool creates a local time in all nodes that precede it along the branch, i.e. the nodes at the left of the Schematic. When you select a node affected by a Time Offset tool, you can disable the Global button in the Tool Options area of the Tool UI to see the local time of the tool—see Converting the Frame Rate on page 695 and Offsetting Time on page 696.

The Time Offset tool lets you adjust the timing relationship between two branches of the dependency graph. For example, if you need to synchronize action in two separate shots, you can create a composition with a branch for each shot and use a Time Offset tool along one branch to adjust the timing relationship between the two shots.

You can examine the local time and global time of a node affected by a Time Offset node in the Tool Options area of the tool UI.

Time Offsets, Keyframing, and Instancing

Only a single animation channel and a single set of animation keyframes per attribute are stored. Selecting the Set Key button sets a keyframe per instance and per local time.

Time Tools

28

Topics in this chapter:

Retiming Footage with the Retimer Tool

The Retimer tool lets you generate output footage at a frame rate that is different from that of the input by retiming the image. Two interpolation modes are available to perform the retiming: the Warp interpolation mode, which uses motion estimation to generate intermediate frames, and the Blend interpolation mode, which generates a weighted average of adjacent frames. The Warp mode provides significantly better quality.

Additionally, motion blur (temporal anti-aliasing) can be added so that moving objects appear blurred, as they would when filmed with a real camera, which has a finite shutter speed. You can then modify the timing and/or speed curves through the Animation editor in the UI to create a constant or non-constant time warp.

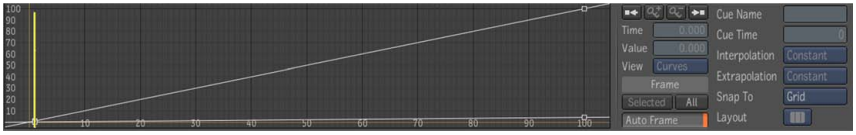
About Timing Curves

Time warps are created by modifying the timing of the source footage or by changing its playback speed. When you change a clip's timing, you are essentially changing the time that it takes for a clip to reach a given frame. For example, in a 100-frame clip, you can adjust the timing so that only the 50th source frame is reached by the end of the clip's 100 frames. This halves the clip's playback rate. A negative playback rate can be achieved by reversing a clip's timing curve (by changing the value at the last keyframe to the value of the first keyframe and the other way around). In the Timing Animation editor, a steeper timing curve results in a faster playback rate.

Adjusting the clip's speed has different results. Setting the speed to 2.0 doubles the playback rate. A speed value of .5 plays the clip at half speed (or in slow motion). A negative speed value corresponds to playing the clip in reverse (where a -1.0 speed value creates a clip that plays back at the normal rate, but backward).

NOTE To avoid indexing black frames, set clip link Repeat parameter to Loop.

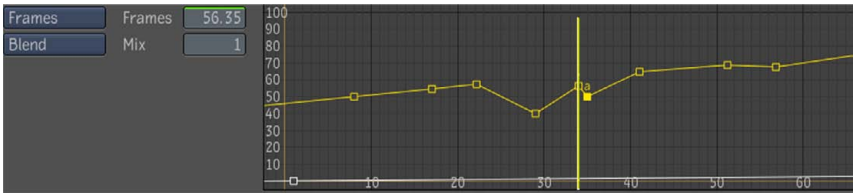
Speed and time curve editors are identical to animation curve editors and have most of the same controls.



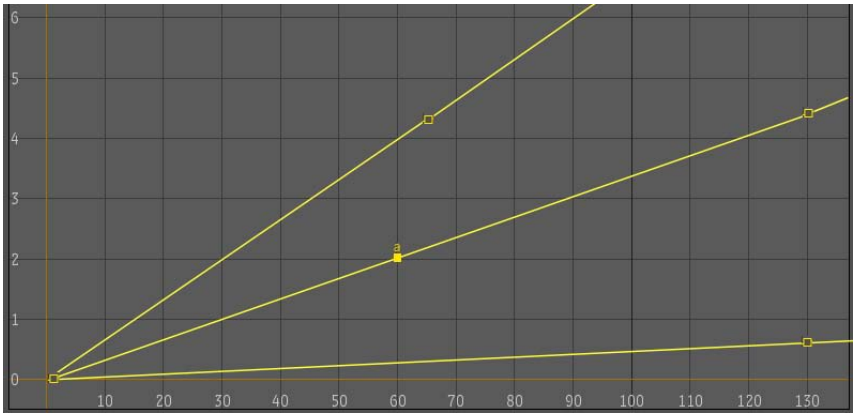
When in Warp interpolation mode, the Retimer has the following parameters:

Use:	To:
Frames	Implement “a frame is a frame” timing. You can control the mapping of output frames to input frames using the Animation

Use:	To:
	Editor. For example, you can add keyframes and have the output frames jump ahead, repeat, run backwards, slow down, or speed up. The Current Frame field will display the input frames and the Frames value editor will display the mapped output frame.



Use:	To:
Seconds	Control the mapping of output time to input time. When output = input, the clip has normal speed. When output > input, the clip is accelerated. When output < input, yet is increasing, the clip is slowed. When output stays constant, a constant frame is shown. When output decreases, time reverses.



Use:	To:
Speed	To control the mapping of input time to output time. When speed = 1, the clip has normal speed. When speed > 1, the clip is accelerated. When 0 < speed < 1, the clip is slowed down. When speed = 0, a constant frame is shown (time remains constant). When speed < 0, time is reversed. The speed curve begins with two key frames. Once created, the key frame times are not automatically

Use:	To:
	updated with respect to changes in start and end times. Remember to set clip link Repeat parameter to Loop to avoid indexing black frames.

Use:	To:
Quality	Control the quality of the internally-generated motion vectors in warp mode by applying the motion analysis only to lower-resolution versions of the input image, up to the resolution specified by the quality parameter. This parameter is disabled as soon as one vector input is connected. Limiting the quality of motion vectors to half or a quarter for a high-resoultion (ex: 1080p or film resolution) input clip can greatly accelerate warp processing, with little loss in resulting retimed output clip quality.
Shutter (camera analogy)	Control how long the camera shutter stays open, and therefore the time range over which the samples will be distributed. A value of 1 means a full frame time. Values above 1, though not physically possible, can be specified for artistic effect.
Reference	Control when a reference motion vector offset is used. If reference is set, the motion vector at the Reference Position is subtracted from all motion vectors in the image. The result is that the pixel at the reference position will appear to be motionless, and so will pixels with similar motion vectors. Conversely, pixels which were motionless will appear to move with the subtracted Reference Position motion vector. This reference motion vector offset changes over time using the velocity of "neutralized" vectors, so you will seldom have to manually set offset keyframes using Reference Position.
Reference Weight	Set the amount of blur to apply at the reference position. A value of zero means that no blur should be applied. This is the default. A value of one means to blur by the same amount as the forward vectors at the Reference Position. This is equivalent to disabling the use of a reference vector offset. A value of one half means to blur half as much as the forward vectors at the Reference Position.
Reference X	Describes the x coordinate of the normalized reference position.
Reference Y	Describes the y coordinate of the normalized reference position.

When in Blend interpolation mode, the Retimer has the same frames, seconds and speed controls as in the Warp interpolation mode. In addition, it also has the Mix control:

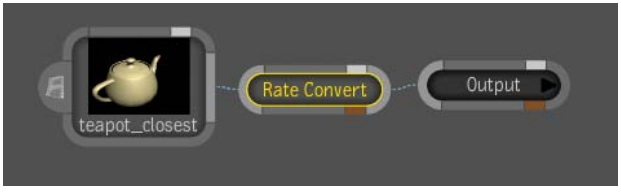
Use:	To:
Mix	Control the number of next and previous frames to blend.

Converting the Frame Rate

You can convert the frame rate of an input source with the Rate Convert tool.

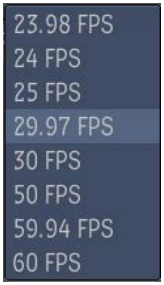
To convert the frame rate of input source footage:

1 In the Tools tab, drag the Rate Convert tool to the Schematic view and connect it to the dependency graph.



The Rate Convert tool UI appears.

2 Select a frame rate to convert to.

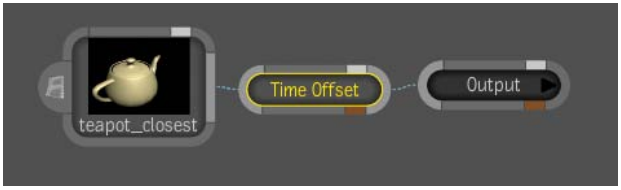


Offsetting Time

You can define a time offset (in units of frames) for an input source with the Time Offset tool. Time offsets are useful when you need to animate local time (tool time) to a master cue.

To define a time offset for an input source:

1 In the Tools tab, drag the Time Offset tool to the Schematic view and connect it to the dependency graph.



The Time Offset tool UI appears.

2 Adjust the offset by the value in the Frames field.

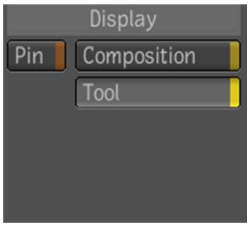
If you have an Animation Editor open, you can view both the global time and tool time as they appear on the same animation curve. Setting the Animation editor display parameters to Tool will set global time frame markers.

To view global time frame markers in an Animation Editor:

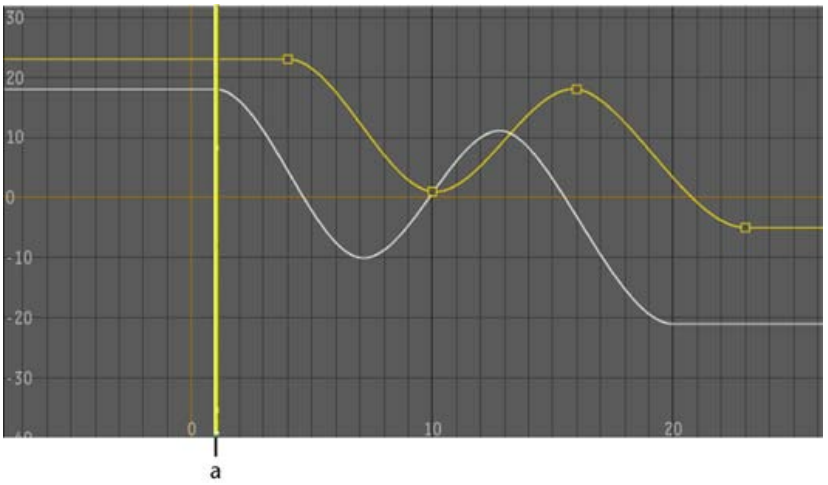
1 While focused in the Animation Editor, open the Gate UI and swipe south to the Animation Editor Options.

The Animation Editor's Display UI appears.

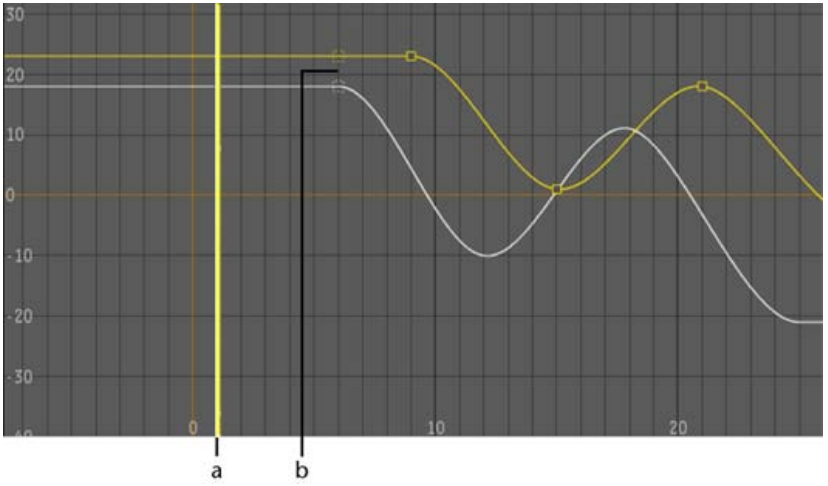
2 Under Display, select the Tool option.



Note that the animation curve changes to reflect the time offset that has been applied and global time frame markers appear.



(a) Global time cursor as shown when Animation editor display is set to Composition



(a) Tool time cursor (b) Global time frame markers

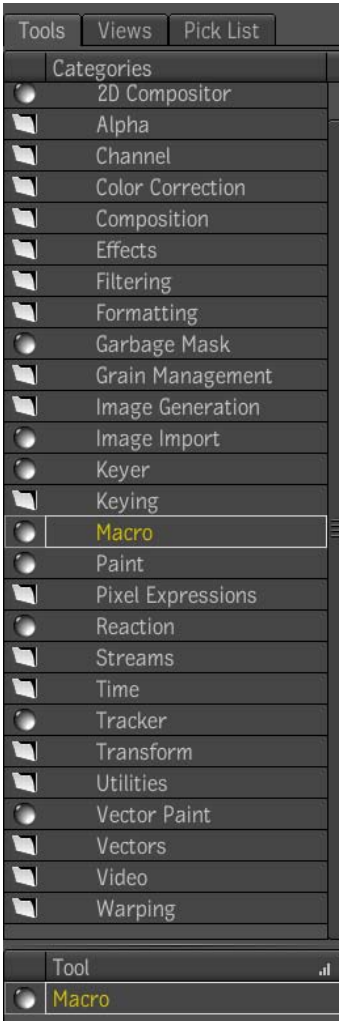
Customization Tools

29

Topics in this chapter:

The Macro Super Tool

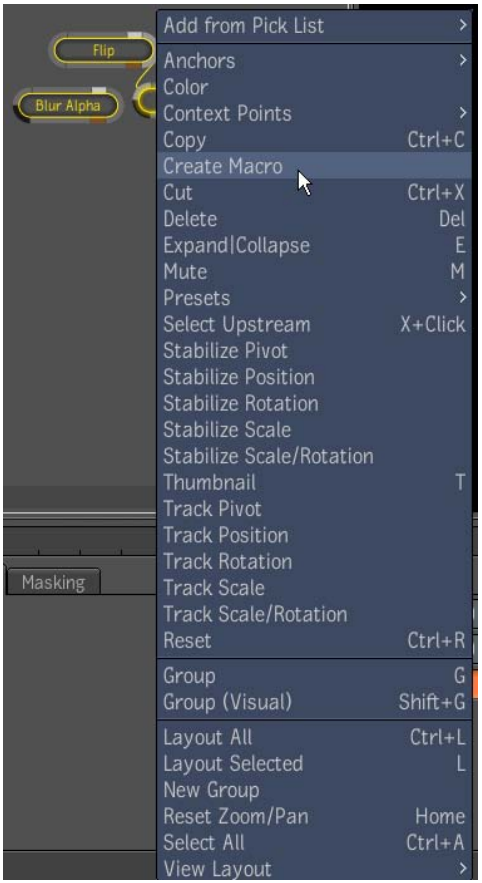
The Macro super tool lets you design your own tool. You can drag tools from the Tools tab directly into the Macro super tool in order to customize it for your composition.



To use the Macro super tool:

1. Do one of the following:
   - Middle-click your mouse or press the tilde key (~) to open the Gate UI. Swipe to the east to open the Tools tab. From the Tools tab, select the Macro super tool or select a Macro super tool preset, if you have previously created one. Drag the Macro super tool into the Schematic view.

- Or, from the Schematic view, select the nodes that you want to include in the Macro super tool, right-click and then select Create Macro.



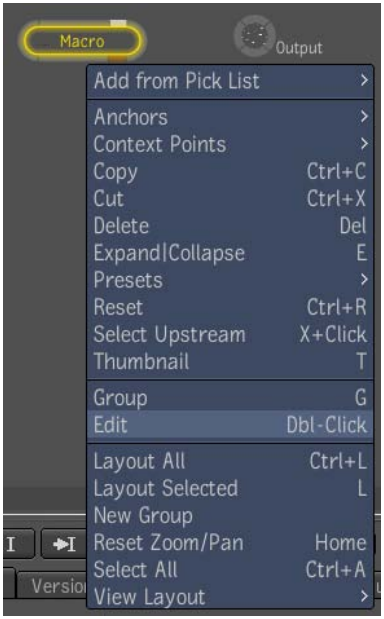
2 Click the Macro super tool to display the Macro tab in the Tool UI. Notice that most of the tab is empty. You'll customize the tab with the tool inputs that you add to the Macro super tool.

On the left side of the tab, the Customize UI button allows you to customize the Tool UI. The Customize UI button is enabled once you are in edit mode.

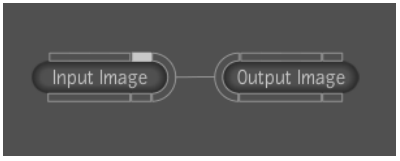
Just above the Customize UI button, there are six more buttons. Once you have added tools to the Macro super tool, you can assign each of these buttons to a tool. Select a tool in the Schematic and then click a button while holding down the Ctrl key to assign it to a tool.

NOTE When the Customize UI button is pressed, these six buttons are disabled.

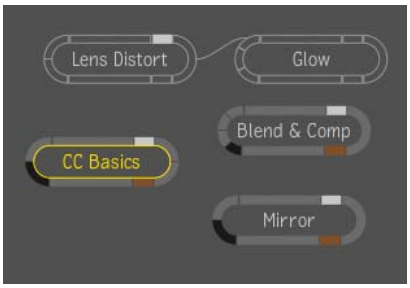
In order to customize the Macro super tool, you must be in edit mode. Right-click on the Macro super tool and select Edit or double-click on it. This allows you to enter the Macro super tool.



Once inside the Macro super tool, what you see will depend on whether you connected the tool to a dependency graph or not. If the Macro super tool is not connected to a dependency graph, you will see an input and an output image, which are simply placeholders that represent the Macro's potential connections outside of edit mode. If the Macro super tool is connected to a dependency graph, then the two nodes that it is connected to will be displayed instead of the input and output images.



Next, you can decide what tools you want to add to the Macro super tool. Open the Gate UI and swipe to the east to open the Tools tab. Now you can drag tools from the Tools tab directly into the Macro super tool.

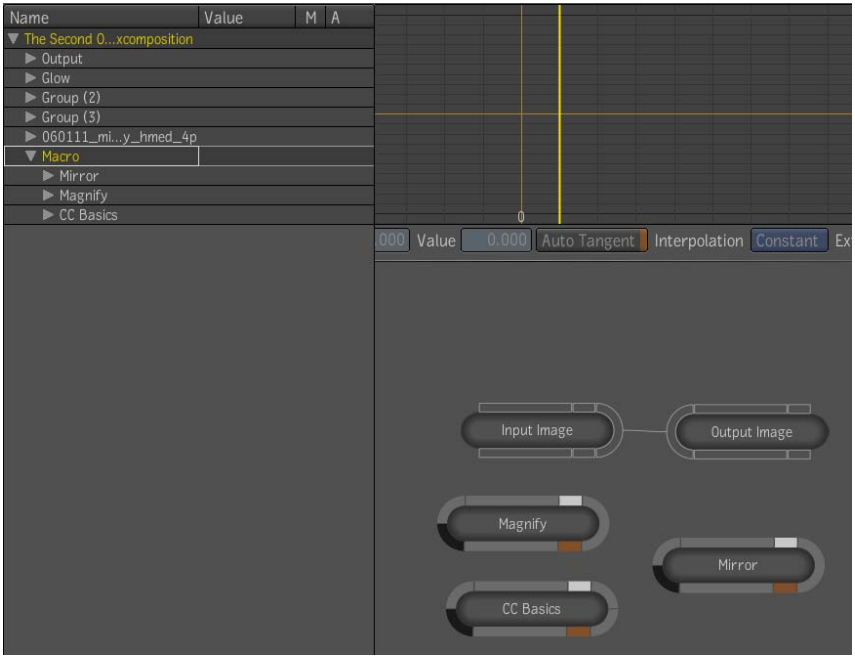


You'll notice that when you click on any of the tools you've added to the Macro super tool, the tab corresponding to the tool in the Tool UI will specify that it is part of the Macro super tool.

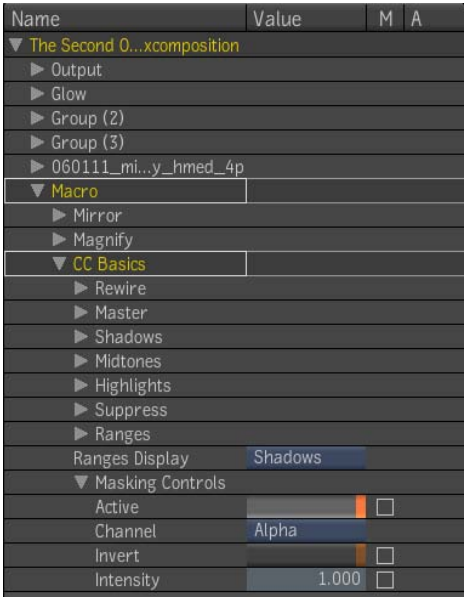
To customize the Macro super tool:

1. Once you have added tools to the Macro super tool, click the Customize UI button.
2. Open a Composition view. Middle-click your mouse or press the tilde (~) key to open the Gate UI. Then swipe east.
3. Click the View tab. From the View tab, drag a Composition view into a viewport. Once the Composition view is open, you will see a Macro sub-menu.

4 Click the Macro sub-menu to reveal the tools that you added to the Macro super tool.

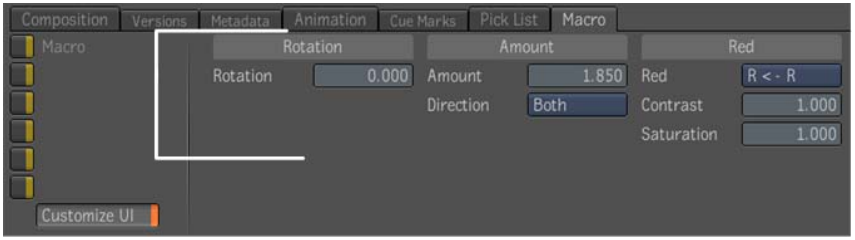


5 Now you can open each tool individually and load the tool inputs that you want into the Tool UI of the Macro super tool.



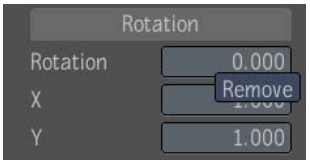
You can do that by dragging and dropping tool inputs from the Macro sub-menu of the Composition view into the Tool UI of the Macro super tool. As you drag tool inputs into the Tool UI, either a green arrow or a red x will appear, indicating whether it is possible to drag the input into the Tool UI or not.

NOTE Animation data and other nodes cannot be dragged into the Tool UI.



You can add as many inputs as you like, however if there are too many inputs, some of them will be hidden at the bottom of the pane.

You can remove an input at any time by right-clicking it and selecting Remove.

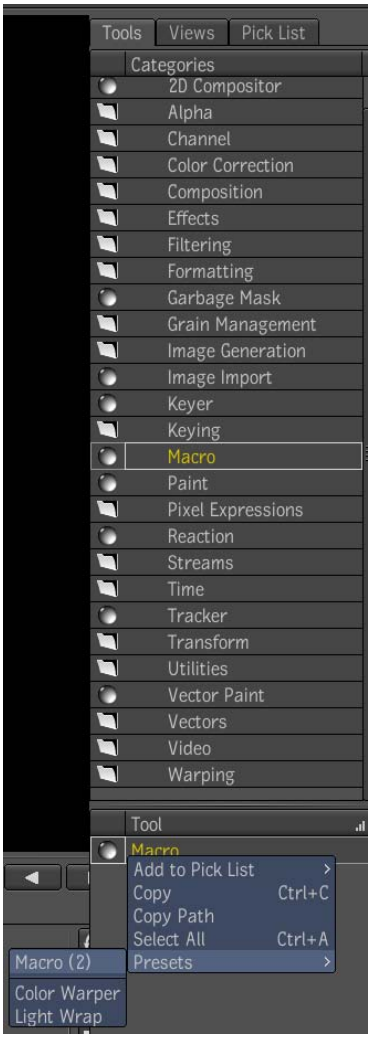


You can rename an item by double-clicking its name and entering a new name in the field.

6 Once you have determined which inputs you want to include in the Macro super tool, you can save it as a preset by clicking the Save button in the Tool options area. You will be required to give the preset a name.

You can access this preset through the Tools menu. Middle-click your mouse or press the tilde key (~) to open the Gate UI, then swipe to the east to open

the Tools tab. Select the Macro super tool. In the lower menu, right-click the Macro super tool and select Presets.



To use a preset in a composition, click it and drag it into the Schematic view.

NOTE Adding a preset to a composition does not prevent you from modifying it. You can continue to add or remove tool inputs from the Macro preset and modify its UI. However, these changes will not be reflected in the composition that the preset has been added to since there is no direct link between the preset and the composition that it is added to.

Vectors

30

Topics in this chapter:

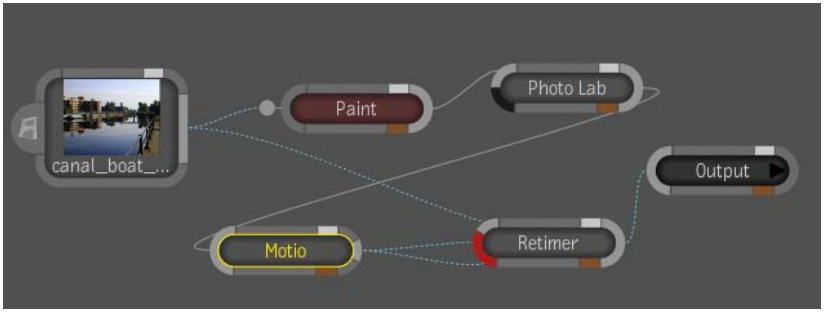
About Motion Vectors

Motion Vectors are 2D vectors that represent the displacement in normalized pixel units of a pixel in the current frame to the next frame (forward motion vector), or its displacement from the previous frame (backward motion vector).

Motion Vectors can be rendered by a 3D application when dealing with synthetic images, or produced through image analysis when images come from live action footage.

The Motio tool lets you generate forward and backward motion vectors with an image source. Motion vectors can be used as inputs for a wide variety of image transformation, analysis, or restoration tools. In Composite, Motion Vectors can be used as inputs to the Retimer tool (see Retiming Footage with the Retimer Tool on page 691), the Blur tool (see About Blurs on page 385), the Remove Dust tool (see Remove Dust Tool on page 380), and the Remove Grain tool (see Removing Grain from a Composition on page 504).

NOTE To avoid nonsense backward vectors on the first frame and nonsense forward vectors on the last frame, the repeat mode of the clip should be such that the image before the first frame and the image after the last frame are similar or identical to the first and last frames, respectively.



Importing Motion Vectors From Maya

There are two types of render passes that you can use in Maya to render your motion vectors: the 2D Motion Vector and the Normalized 2D Motion Vector.

NOTE You can import 2D motion vectors that were computed in another application (e.g. 3D render)—see Importing Media on page 118

The 2D Motion Vector appears as “mv2dtoxik” when it is created. If you render your motion vectors in Maya using the 2D Motion Vector render pass, the motion vectors data is kept intact. Furthermore, you don’t need to set any motion vector options on import in Composite.

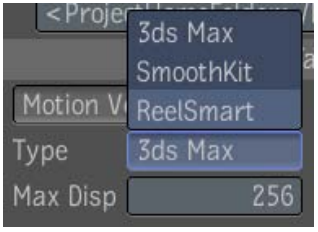
The Normalized 2D Motion Vector appears as “mv2dnormremap” when it is created. If you render your motion vectors in Maya using the Normalized 2D Motion Vector render pass, the motion vectors are normalized between 0 and 1. When importing the render pass in Composite, you have to set motion vector import options to convert the motion vector data to the true values.

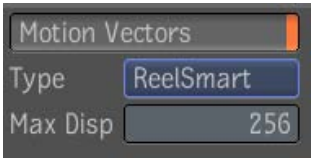
NOTE For both the 2D Motion Vector and the Normalized 2D Motion Vector, you must make sure to render your motion vectors in the EXR format.

In Composite, you can set the motion vector options on import or after importing the render pass.

To set the motion vector options on import:

1. Open an Import Browser.
2. Browse to the render pass and select it.
3. Next, from the tool UI, enable the "Motion Vectors" button and set the Type menu to "ReelSmart."
4. Finally, set the Max Displacement to the same value that was set in Maya.





In Maya, the Max Displacement value appears under the Normalized 2D Motion Vector render pass options. You can access the render pass options by double-clicking the render pass in the Render Settings window. By default, the Max Displacement value is set to 256.

To set the Motion Vector options after importing the render pass:

1. Select the import node in your composition.
2. From the tool UI, enable the Motion Vectors button.
3. Set the Type menu to "ReelSmart."
4. Finally, set the Max Displacement to the value that it was set in Maya.

Importing 2D Motion Vectors from 3ds Max

If your motion vectors are coming from 3ds Max, you have to set the Motion Vectors type to "3ds Max" and set Max Displacement values as they were set in 3ds Max.

Importing 2D Motion Vectors from SmoothKit

If your motion vectors are coming from SmoothKit, then you have to set the Motion Vectors type to “SmoothKit”.

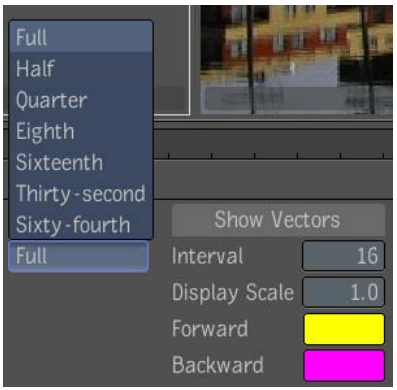
Once you have imported your motion vectors, refer to Vector Blurs in theAbout Blurs on page 385 section in order to know how to set motion vectors.

Computing Motion Vectors

The Motio tool uses the assumption of brightness consistency of an image to generate motion vectors from one frame to the next. That is, the luminance values remain constant over time, but their 2D position in the image may change. Flashing lights, shadows, and other image changes that violate brightness consistency may interfere with the generation of motion vectors and cause problems with your retiming operation. You may want to first fix these problems using a Paint tool or CC Basics tool for example, then calculate the adjusted forward and backward motion vectors with a Motio node. Once this is done, you can connect the adjusted vectors to a Retimer tool and use your original image as the input.

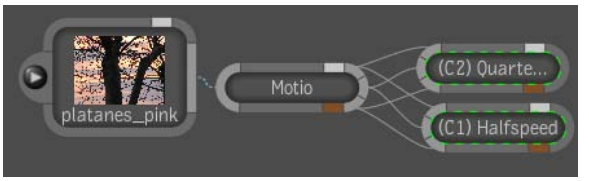
If no motion vectors have been imported, you can compute them inside a composition by adding a Motio node. Certain tool nodes, such as the Retimer, will detect the absence of motion vector inputs, and will trigger the use of its internal Motio engine to automatically compute motion vectors. Computing motion vectors explicitly allow you to use the vectors for more than one vector consumer tool. For example, you may want to retime some footage with different speeds and then quickly compare the results. Instead of computing the vectors twice (in each Retimer), you can use the Motio tool's output twice.

The Motio tool has a non-animated scalar parameter, called Quality, as a well as a Show Vectors parameter—see Show Vectors on page 716. This parameter controls the quality of the motion vectors by applying the motion analysis only to lower-resolution versions of the input image, up to the resolution specified by the quality parameter.



To compute motion vectors inside a composition with the Motio tool and compare retimed footage:

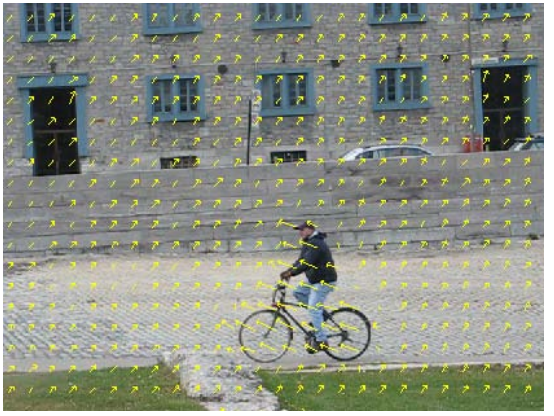
1. Drop a Motio tool into the Schematic view of a composition you want to retime.
2. Connect the output of the footage to the input of the Motio node.
3. Add a Retimer tool to your dependency tree, and rename it HalfSpeed.
4. Make the Speed value 0.5.
5. Set the Retimer's context point to C1—see Setting Context Points on page 191.
6. Connect the three Motio outputs to the corresponding inputs of the HalfSpeed Retimer.
7. Set the target of a Player to context point C1—see Setting the Target on page 191.
8. Add a second Retimer to Schematic and change its name to QuarterSpeed and enter 0.25 into its Speed value editor.
9. Set the Retimer's context point to C2—see Setting Context Points on page 191.
10. Connect the three Motio outputs to the corresponding inputs of the QuarterSpeed Retimer.
11. Set the target of a second Player to context point C2—see Setting the Target on page 191.
12. Play and compare the two retimed clips.



Show Vectors

The Show Vectors tool is a three input, single output image pass-through tool that allows in-player viewing of 2D vectors as overlays on an RGBA image. Images representing a 2D vector per pixel can be used as image inputs to indicate pixel displacement from one frame to another. These 2D vector images can either be generated for artistic effect, the result of an analysis, or imported from a render generated from a 3D package.

The 2D vector (x, y) components are directly stored as the red and green channels in the vector image; the blue and alpha channels are unused. The Show Vectors tool makes viewing 2D vector images far more intuitive. Instead of showing a red, green, yellow (mix of red and green) or black (negative components) image, actual arrows are shown, indicating the origin, direction, and amplitude of motion, for either the forward vector input image, the backward vector input image, or both. Because the vectors are shown as overlays on top of a regular RGBA image, the positional relationship between the vectors and the RGBA image can clearly be seen. This is often crucial to understand how the 2D vectors will affect that RGBA image.

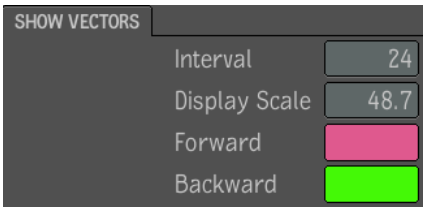


Forward vectors

Backward vectors

Forward and backward vectors

The Show Vectors tool has the following parameters:



Select:	To:
Interval	Set the interval between shown vectors. By default, this value is 16 (shows a vector every 16 pixels). This controls the density of the displayed vectors, and is applied to both the forward and backward vectors.
Display Scale	Set the display Scale of the shown vectors. By default, this value is 1.0 (shows a vector with its original length). This controls the displayed length of the vectors, and is applied to both the forward and backward vectors.These values only control the player display. They have absolutely no influence on processing, since the Show Vectors tool is an image pass-through of its primary input.
Forward/Backward vectors color pots	Set the color of displayed vectors. Click the color pot to display the color picker—see Color Picker on page 154.

To display motion vector overlays on an image with the Show Vectors tool:

1. Drop a Show Vectors tool node onto a composition.
2. Connect the Show Vectors primary image input to the footage node.
3. Connect the Show Vectors Forward Vectors input, and/or the Backward Vectors input to vector image output(s).
4. Select the Show Vectors tool node. The Player shows forward vectors (if connected) as 2D vectors on top of the input image.
5. Press the m key (lowercase) to display backward vectors (if connected) as 2D vectors are on top of the input image. Press it again to display both forward and backward vectors on top of the input image. Pressing it a third time will only show the input image. Pressing it a fourth time will start the cycle over.

Expressions

31

Topics in this chapter:

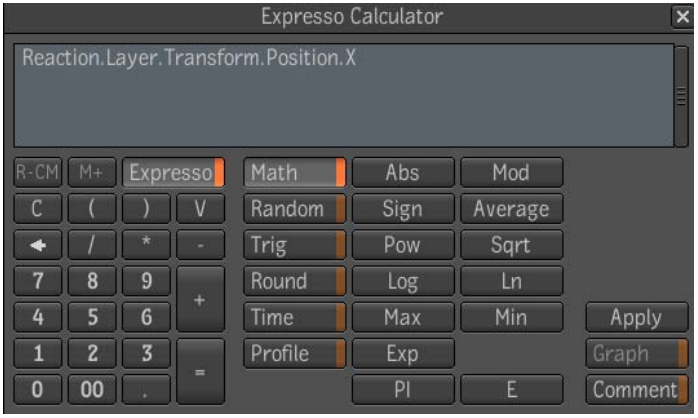
About Expressions

Expressions are mathematical formulas that let you control any parameter that can be animated, such as translation, rotation, scaling, material, or texture. You can create almost any association between parameters, from simple A=B relationships to very complex ones using predefined variables, standard math functions, random number generators, and more.

An expression is a string of characters that may include object and parameter names, math operators, and tokens representing functions or constants. For example, to constrain object A's Y rotation to object B's X translation, you might set an expression on **`A.Transform.Rotation.Y`** consisting of the string **`B.Transform.Position.X`**

Expressions are created by using the Expresso Calculator, which is

an extension of the regular Calculators on page 161. To set an expression you need to access the calculator by right-clicking in the value field of a parameter.



The Expresso button is a toggle. Clicking it again collapses the Expresso Calculator.

NOTE If you display the regular calculator by using Alt + C, then the Expresso button is inactive. The Expresso button only toggles if you first prompt the Expresso Calculator.

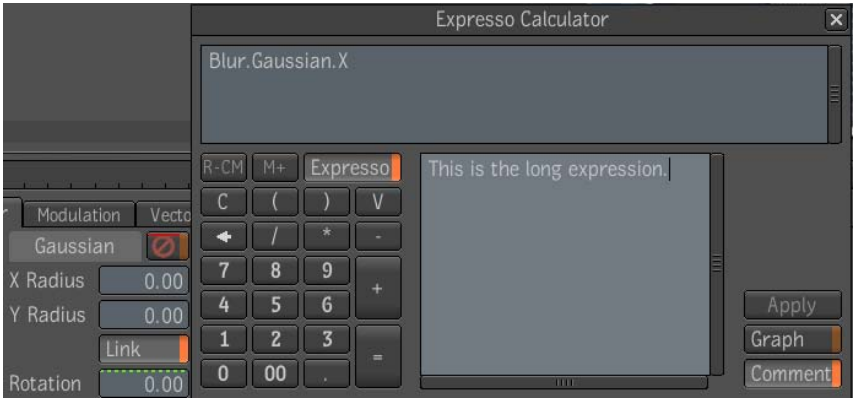
What attributes can expressions refer to?

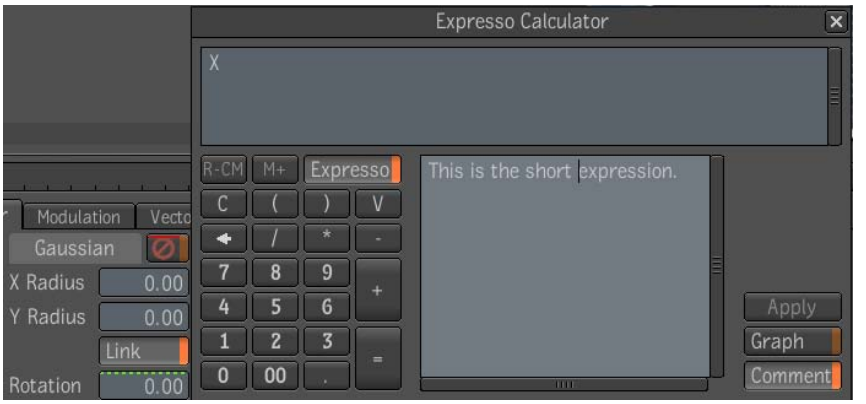
Expression strings refer to sockets by name. The sockets can be either input or output sockets. Expressions perform operations based on numerical values, therefore you do not access pixels in the context of expressions. However, you can still access numerical values of image attributes such as height and width.

Short Expressions

In cases where you may not want to type the full name of an attribute that is at the same level as the attribute being expressed, you can enter expressions that refer to an input by its short name.

For example, if you want to add an expression to the Gaussian Rotation based on the value of the Gaussian X Radius, instead of writing out the long expression, you only need to refer to its short expression.





This applies to all grouping levels, be it inputs on a node, or inputs in a structure, or inputs on a node that is in a group.

Expression Input Paths

To avoid having to manually enter a long expression based on the input of an attribute, you can right-click on that attribute's value field either in the tool UI or Composition Browser and select Copy Path, then use **Ctrl + V** (Windows and Linux) or Cmd + V (Mac OS) to paste the path into the Expresso Calculator.

Visual Linking

Visual linking based on expressions is used to quickly make a connection between nodes or inputs without having to manually set expressions through the Expresso Calculator. You can link simple or structured inputs. Linking structured inputs links individual inputs in a single operation. Its purpose is to quickly define a link between two attributes. It does so by defining a B=A expression, B being the link target and A the link source.

Visual Linking Methods

You can visually link attributes within the Tool UI, in the Composition Browser or between the Tool UI and the Composition Browser. However, visual linking of structured inputs can only be done through the Composition Browser.

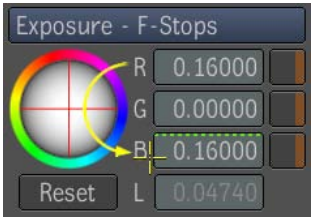
Visual Linking in the Tool UI

The Tool UI displays editors (value editors, toggle buttons or menus) and labels. Visual linking relies on the attribute labels as zones to trigger a link gesture. Labels are already used for marking.

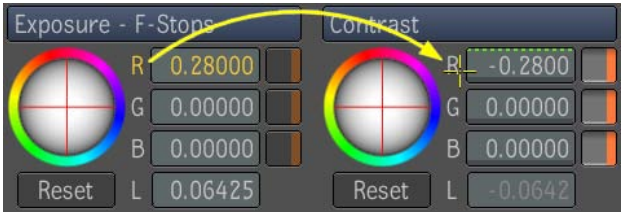
To link the Red and Blue components of a color input in the same group:

➤ Click the Red channel label, drag and drop on the Blue channel label.

This sets an expression on the Blue channel attribute referring to the Red channel attribute without ever displaying the Expresso Calculator.



In the same way, you can link inputs belonging to separate groups, setting an expression in the target attribute referring to the source input.

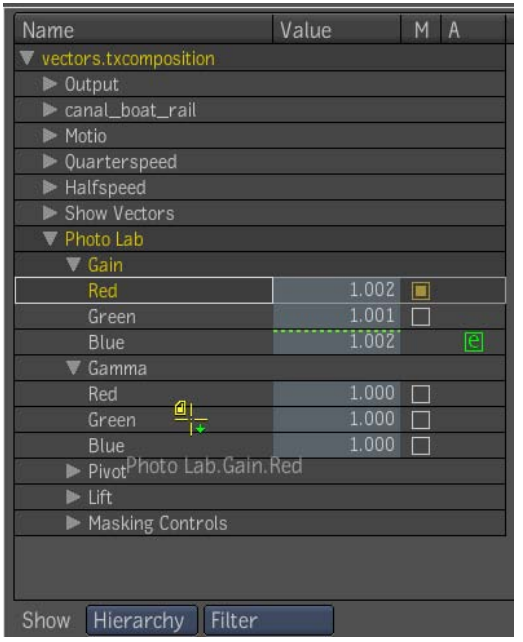


Visual Linking in the Composition Browser

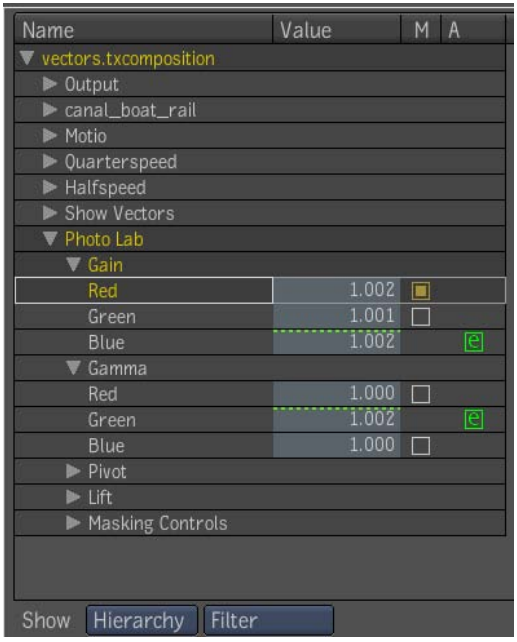
You can perform visual linking operations within the Composition Browser.

Linking Simple Inputs

Linking simple inputs in the Composition Browser works like in the tool UI; you can click and drag attribute names (labels) to define a link expression. Simple inputs are defined as those inputs that have only one socket name.

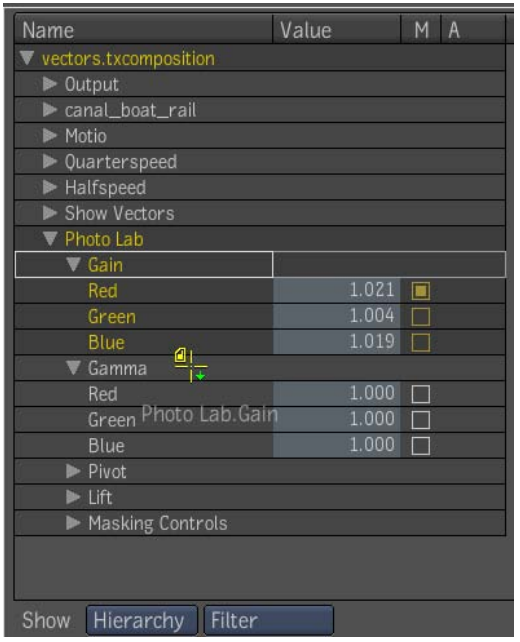


Once the visual link has been established, the target attribute's value assumes that of the reference attribute. A green E then appears in the A column of the Composition Browser.

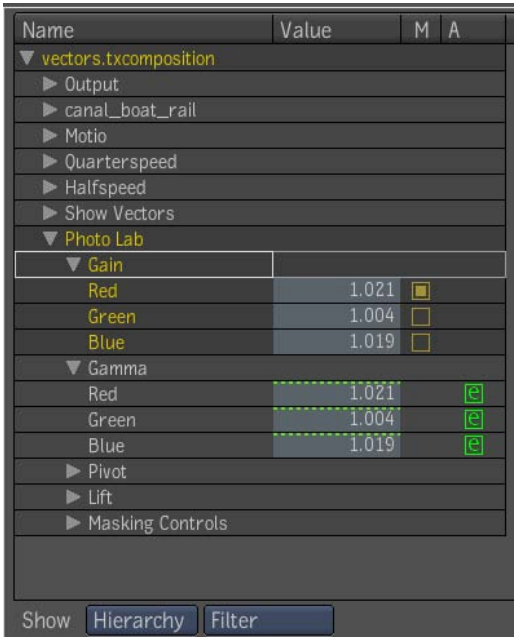


Linking Structured Inputs

Linking structured inputs is done by dragging and dropping a structured input onto another. If the target structured input's topology doesn't match the source one, the drop is not accepted (i.e. the drop feedback is not given to you while dragging the pointer). The graphic below illustrates how an attribute with structured inputs (Master Gamma) can be visually linked with another attribute with structured attributes (Master Gain).

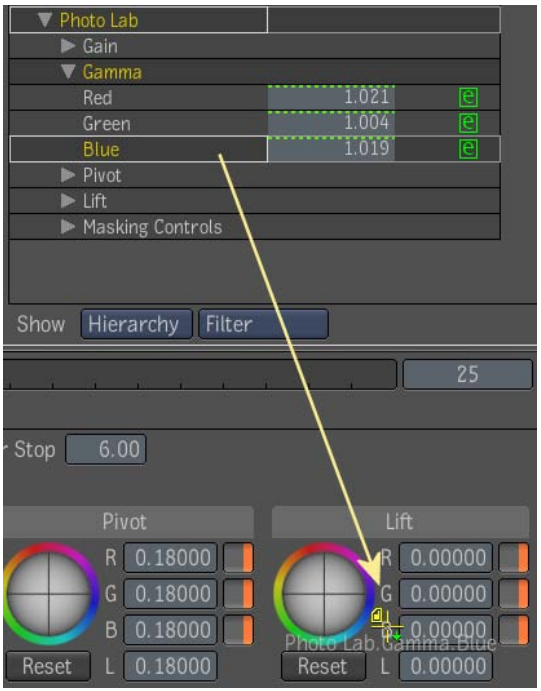


When the source and target structured inputs match (that is, when they have the same number of inputs) an expression per input is set, linking inputs individually.



Visual Linking between the Composition Browser and the Tool UI

You can initiate a drag from the Composition Browser and drop it on a label on the tool UI, linking the source and target attributes with an expression.



Navigating the Expression String

You can type an expression into the text field at the top of the calculator or press the buttons to generate an expression string.

All the text field hotkeys are enabled when editing the expression string. You can therefore navigate the string normally. When using the function buttons to insert function tokens into the expression text field, the cursor position is updated to the following local position in the string. This lets you simply tap with a pen and insert expression tokens without having to move the cursor position manually.

Expression functions and operators are grouped by categories. The Math category of functions is available by default. Although you can type into the text field directly, showing the buttons helps you to memorize the available functions.

Clicking the C button clears the expression string.

Once an expression is used to drive an attribute value, you can no longer keyframe the value. You must delete the expression before setting a keyframe.

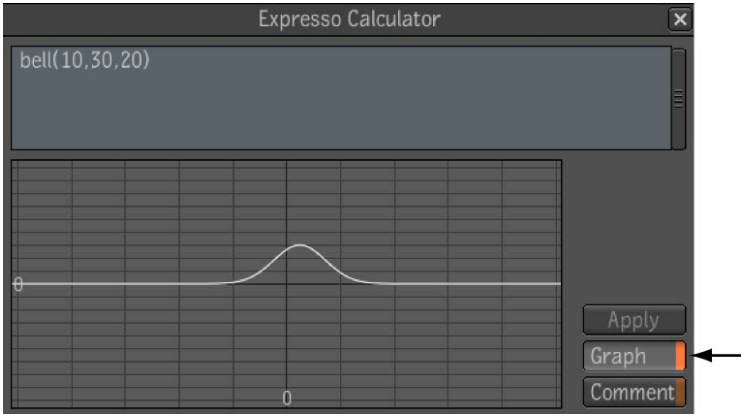
Validating and Applying the Expression String

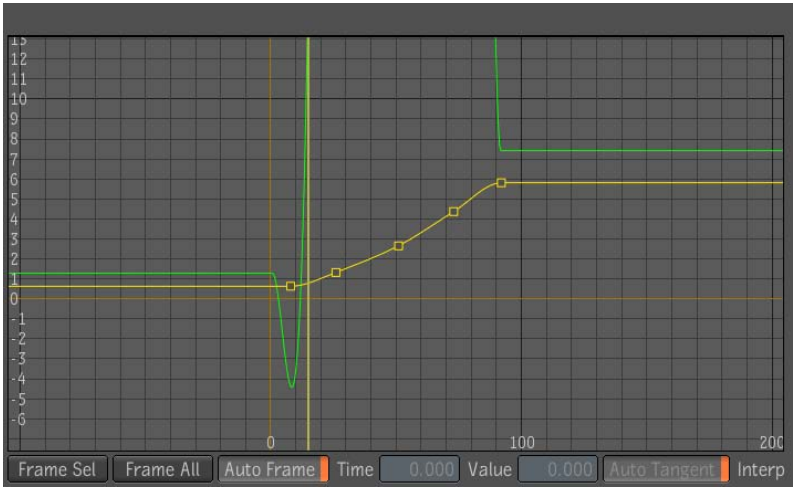
Once you have entered the expression, you need to validate it. This is done by pressing the **Enter** key on your keyboard. If no error is found, the expression is parsed and validated. An invalid expression string returns an error message in the taskbar at the bottom of the screen. The Apply button is grayed out unless the expression has been previously validated. Pressing it applies the expression to the associated attribute by attaching an expression node to it.

Viewing the Expression

You can view a valid expression in a graph view by clicking the Graph button. The graph view displays the shape of the expression curve plotted over time. The initial time extents are the same as those of the composition.

You can also display the expression curve in the Animation Editor Viewer with other animation curves. The expression curve is visually differentiated by being displayed in a green line and will always appear underneath animated curves. The expression curves cannot be selected nor modified. To Display the curve, select the attribute containing the expression from the Composition Browser.



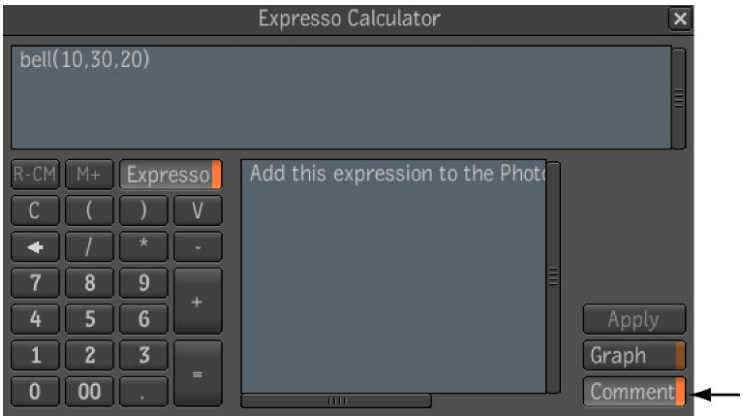


NOTE You can zoom the view by placing the cursor inside the view and using the Up and Down arrow keys, or by using **Ctrl** + **Space**bar + click (Windows and Linux) or Cmd + Spacebar + click (Mac OS). You can also pan the view by simultaneously holding down the **Spacebar** on the keyboard and the left mouse button.

NOTE To return to the main Expresso Calculator view, select the Graph button again.

Associating Comments with an Expression

You can associate comments with an expression by clicking the Comment button.



NOTE To return to the main Expresso Calculator view, select the Comment button again.

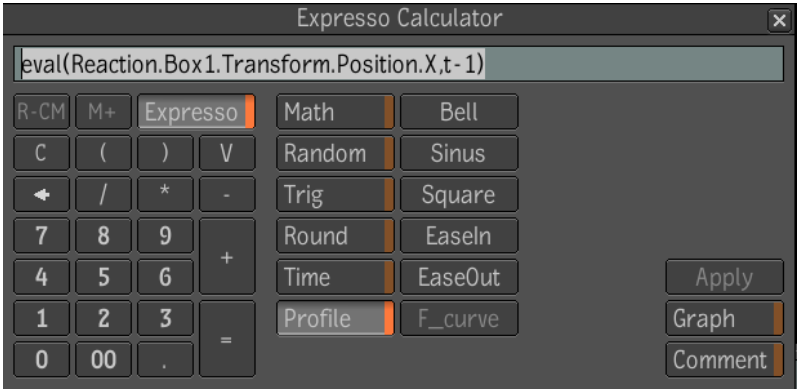
Editing an Expression

You can easily edit an expression.

To edit an expression:

1. Right-click the parameter field that contains the expression you want to edit.
2. From the menu select Edit Expression.

The Expresso Calculator appears with the expression highlighted in the text field.



3 Edit and reapply the expression.

To save time, you can cycle through the various expression tokens that make up the string by using **Alt + slash** (/) on the keyboard. This is particularly useful when you are not sure of the exact sequence or writing convention of the expression string.

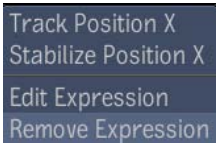
Removing an Expression

You can remove expressions from any attribute.

To remove an expression node from an attribute:

1 Right-click the value field of the parameter that contains the expression you want to remove.

2 From the menu select Remove Expression.



The expression node is removed.

Setting Expressions Examples

The following examples will show you some of the ways expressions can be used to save time in building your animations. You can create an expression for one channel and use the same expression for other channels so that the other channels automatically behave in relation to the first one.

Example: Using Linked Attributes to Drive Animation

This example consists of a simple animation where the Photo Lab red channel Gain attribute drives the Radial Ramp tool's Y Center attribute—see Setting Keys Manually on page 676, Radial Ramp Tool on page 426, and Processing Images with Photo Lab on page 646.

NOTE It is assumed that you currently have the Player, Schematic, and Composition Browsers open. With the focus on Schematic, use **Ctrl** + **N** (Windows and Linux) or Cmd + N (Mac OS) to create a new composition.

To create an animation where the Photo Lab's red channel Gain attribute drives the Radial Ramp's Y Center position attribute:

1. From the Tools tab, drag the Radial Ramp tool from the Image Generation folder to the dependency graph in the Schematic view.
2. Drag the Photo Lab tool from the Color Correction folder to Schematic and place it after the Radial Ramp tool.

   The dependency graph should look like this.

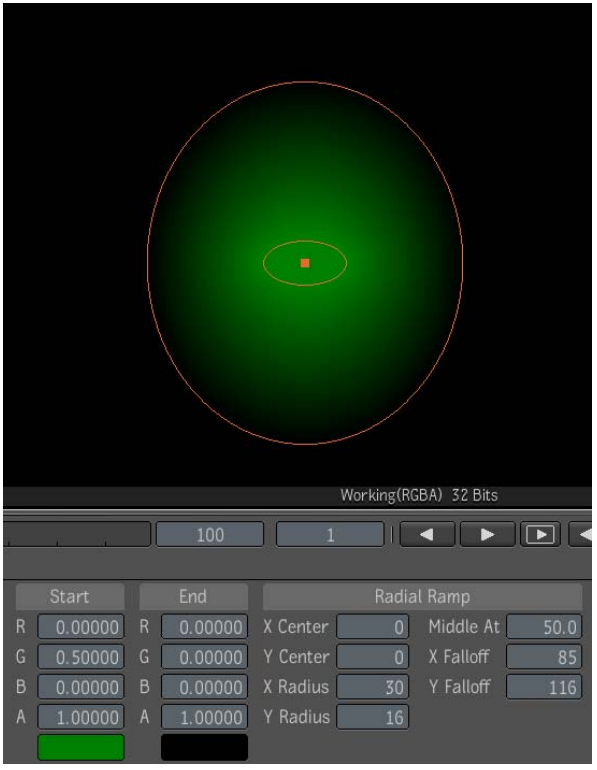
3. 

4. In the Player controls, change the composition's total number of frames to 100.

5 In the Radial Ramp tool UI, modify the following attribute values with the following settings:

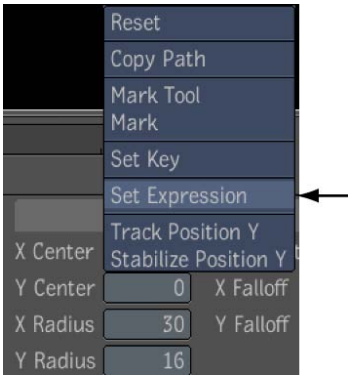
Attribute	New Setting
Start Color Green (G) channel	0.500
X Radius	30
Y Radius	16
X Falloff	85
Y Falloff	116

The Radial Ramp should look like this.

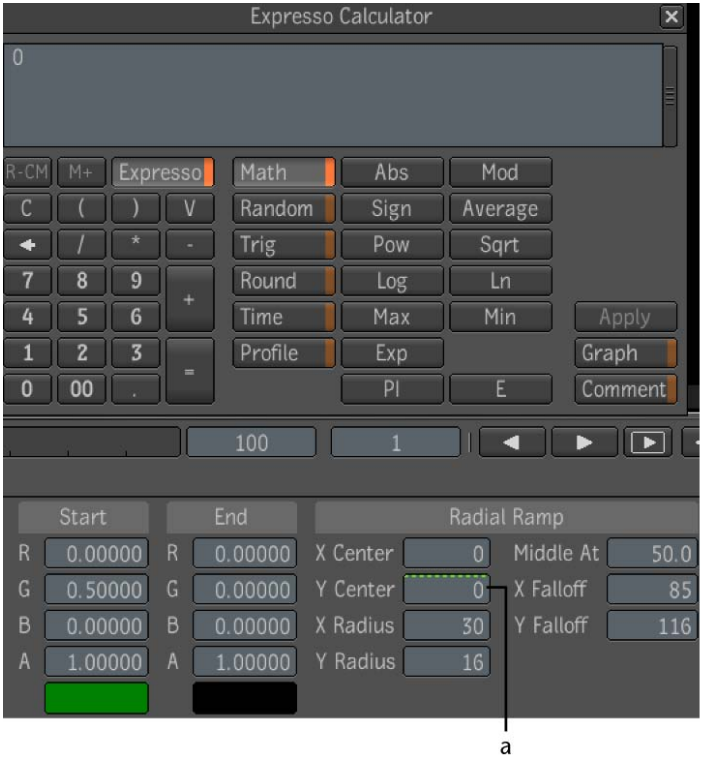


You can now set an expression for the Y Center attribute that enables it to be driven by the Photo Lab's Gain, Red channel attribute.

6 Right-click inside the Y Center value field. And select Set Expression.



The Expresso Calculator appears along with a dashed green line at the top of the Y Center value field. This indicates that an expression has been applied.



(a) Bar above value changes to a dashed green line

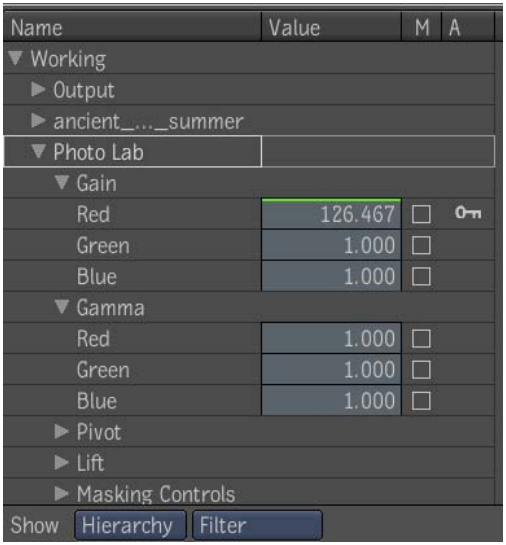
7 In the text field at the top of the calculator, type in the following expression string.

```
Photo Lab.Gain.Red
```

This string is typical of what an expression looks like. The Y Center of the Radial Ramp points to the red channel of the gain controls in the Photo Lab tool, and links them. If you are familiar with expressions, you can simply type the expression into the text field (using the correct syntax and order), but if you are not sure of the exact sequence or writing convention, you can cycle through the various expression tokens that make up the string by using **Alt** + **slash** (/) on the keyboard.

NOTE Both **Alt** keys work, but only the **slash** (/) that is on the question mark (?) key can be used.

By opening a Composition Browser view, you can see the socket names of each tool attribute once the menus are expanded. You can use these names as a reference when creating expression strings, as this is how they should look in the expression.



NOTE When cycling through and selecting a tool's attribute levels, you must separate each level with a period.

8 Open the Photo Lab tool UI.

9 Select Autokey.

10 Change the exposure units to Gain.

11 Mark the red (R) channel of the Gain for keyframing by clicking the R label—see Marking Attributes for Keyframing on page 674.

12 In intervals of 20 keyframes, increase the gain on the red channel, so that it is at its maximum by frame 100.

13 Play the animation.

Example: Using Functions to Drive Animation

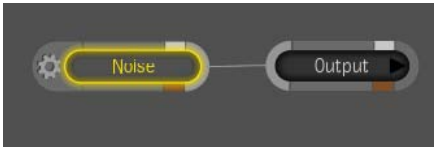
This example shows you how pre-established functions can be used as expressions to drive animation—see Expression Reference Tables on page 745.

The goal of this example is to create random noise using the Noise tool—see Noise Tool on page 427. You could create the appearance of random noise by modifying the Noise Seed attribute using periodic keyframes. However, a more simple approach is to use a random number function. In this case, the TrueRand function is used.

NOTE It is assumed that you currently have the Player, Schematic, and Composition Browsers open. With the focus on Schematic, use **Ctrl** + **N** (Windows and Linux) or Cmd + N (Mac OS) to create a new composition.

To create random noise using a random number function:

1 From the Tools tab, drag the Noise tool from the Image Generation folder to the dependency graph in the Schematic view and connect it to the Output node.

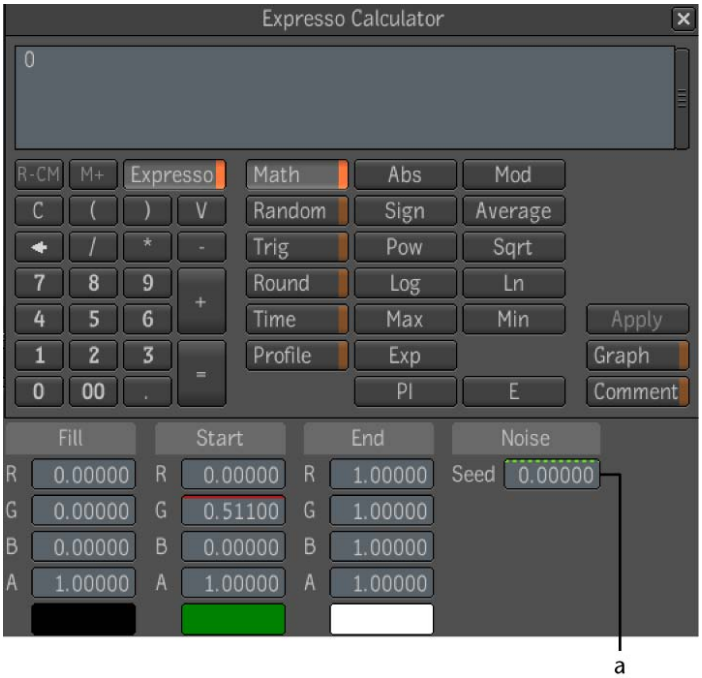


2 In the Player controls, change the composition's total number of frames to 100.

3 In the tool UI, set the Fill, Start, and End color preferences.

4 Right-click the Noise Seed value field and select Set Expression from the menu.

The Expresso Calculator appears along with a dashed green line at the top of the Noise Seed value field. This indicates that an expression is about to be applied, or has already been applied.



(a) Dashed green line appears

5 Select the Random button if it is not already selected.

6 Select the TrueRand button.

An empty expression appears in the text field.

NOTE If you are familiar with the syntax of expressions, you can type them directly into the text field.

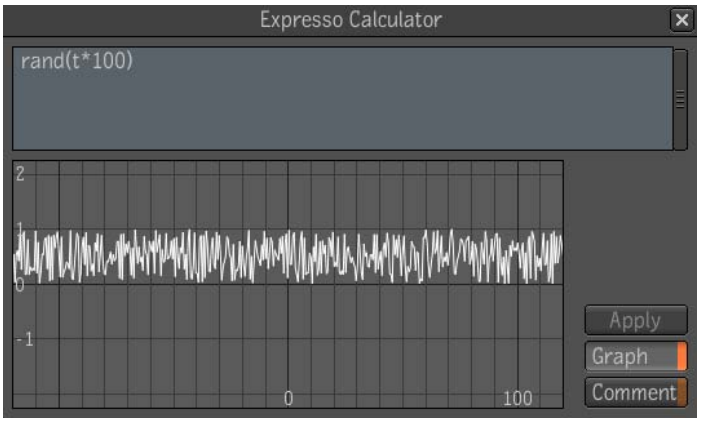
As the TrueRand function description indicates (see Random Number Functions on page 749) the function returns a truly random value between two given numbers. Since the Noise Seed value can range from 0 to 1, it is suggested that these values be used as the range.

7 Enter the value range of 0 to 1 between the brackets in the text field, making sure you separate the values with a comma.

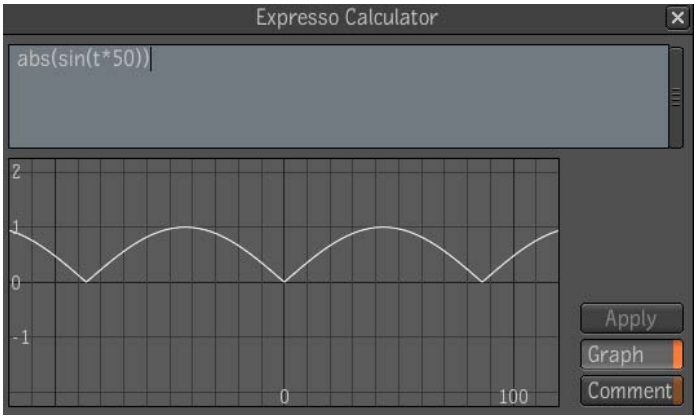
8 Play the composition.

NOTE You can view the expression in a graph view at any time by right-clicking the attribute's value field containing the expression, and then selecting Edit Expression from the menu. The Expresso Calculator appears. Select the Graph button to view the expression in a graph.

There are other expressions, and combinations of expressions that you can use to achieve the same results. For example, you can create random noise (with the Noise tool) by using the Rand function where the seed is calculated as time (t) multiplied (*) by the number of frames in the composition. Therefore, if the composition is 100 frames long, the expression would look like this:



Another expression that can be used to create random noise with the Noise tool is the Abs math function.



This expression indicates the Noise Seed is based on an absolute value (anything between 0 and 1) of the angle (in radians) of the sine wave created by the current time (t). It is then multiplied (*) by a frequency of 50. The expression would still be valid if the Abs math function were not part of the string.

Example: Animating Layers in Reaction

This example is somewhat more involved in that it uses the Eval time function within a Reaction node—see Time Functions on page 751 and Basic Compositing in Reaction on page 243.

This example has two parts. First you'll set up the display and then you'll create the animation.

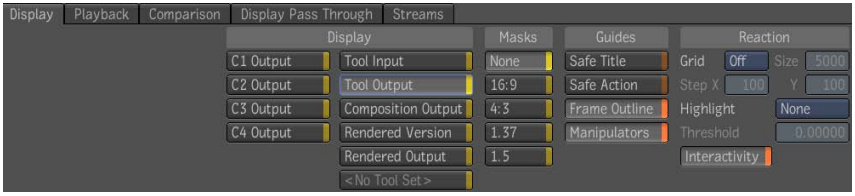
NOTE It is assumed that you currently have a Player, Schematic, and Composition Browser open. With the focus on Schematic, use **Ctrl + N** (Windows and Linux) or Cmd + N (Mac OS) to create a new composition.

Before you begin, make sure that you set the Player display settings as follows.

To set Player display settings for Reaction:

1. Open the Gate UI and swipe south to display the Player options.

   The Player control panel is displayed across the bottom of the screen. By default the Display tab is selected and you can immediately access the display settings and options.



2. Select Tool Output to view the results of the Reaction composition in the Player.
3. Under Guides, select Frame Outline.
4. Under Reaction, select Interactivity.

To create an animation in Reaction using expressions:

1. From the Tools tab, drag the Reaction tool to the dependency graph in the Schematic view, and connect it to the Output node.

   NOTE Usually, before you composite or add effects such as an animation in Reaction, you typically set the background. The background is the rendering plane for the composition and sets the format. If you connect a Reaction tool to an image or tool output in an existing dependency graph, the background is set automatically. You can change the background at any time.

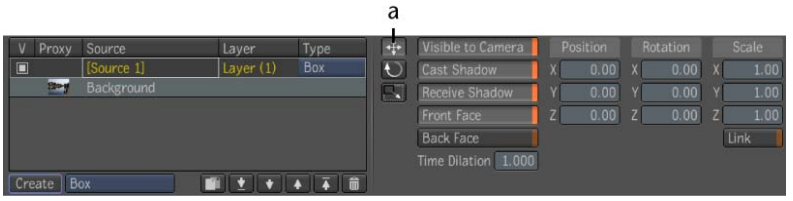
2. Select the Reaction tab.
3. In the Player controls, change the composition's total number of frames to 100.

4 At the bottom of the Layer Editor, select Box and click Create.

A new layer (a cube) is added to Reaction. Rename the layer by clicking in its text field and pressing **F2**. Type in the new name, Box1, and press **Enter**.

Depending on which preset you have chosen to view, the box you have created most likely takes up the entire Player view.

5 Select the Reaction tab.



(a) Translate tool

NOTE To help you create an appropriate view of the animation, you can modify the Player view by adjusting the zoom—see Zooming and Panning on page 39.

6 Use the Translate tool to move the box to the bottom-left corner of the view by grabbing the X or Y axis with the cursor. If you do not see the

axes icon in the Player, go to the Player display options and select Tool Output under Display, and Icons under Guides.

7 Mark the X and Y Position attributes for keyframing.

8 Select Autokey.

9 Using the Translate tool, create a motion path with the cube in Autokey to the right-hand side of the Player view. Try to span all 100 frames.

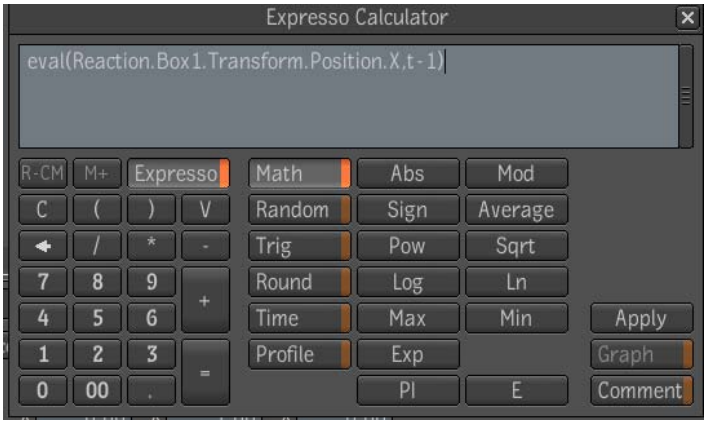
10 Play the animation.

11 Select the Reaction tab and create a second box, Box2, layer by following steps 5 through 8.

12 Right-click in Box2's X Position attribute value field and select Set Expression.

The Expresso Calculator appears.

13 Insert the following expression.



14 Repeat steps 12 and 13 for Box2's Y Position attribute, but make sure the Position token is replaced with a Y.

15 Play the animation.

This expression indicates that the X and Y Position attributes of Box2 are being driven by the X and Y Position attributes of Box1. There is also a time offset of one second (or number of frames per second) added to the expression.

NOTE Unfortunately, it is beyond the scope of this guide to provide examples of all the expressions and possible combinations of functions that you can use to drive animations. The examples provided are just a small sampling of what is available. It is strongly suggested that you explore the use of expressions as they can prove to be an extremely powerful tool in creating complex effects and saving time.

Expression Reference Tables

The following expression reference tables serve as a set of basic expression syntax formats in order to get you started using expressions. They are classified by function, and include operator reference tables and function reference tables.

There are 12 expression reference tables:

Expression Table	Description
Arithmetic Operators	Perform basic mathematical functions—see Arithmetic Operators on page 746.
Comparison Operators	Compare two values with each other—see Comparison Operators on page 746.
Operator Precedence	Operations are performed in a specific order—see Operator Precedence on page 747.
Math Functions	Advanced math functions—see Math Functions on page 747.
Vector Functions	Operators that are based on vector behavior—see Vector Functions on page 748.
Random Number Functions	Operators that return random values based on given vectors and seed values—see Random Number Functions on page 749.
Rounding Functions	Returns rounded values—see Rounding Functions on page 750.
Trigonometric Functions	Returns or converts values based on trigonometric functions—see Trigonometric Functions on page 750.
Constants	Returns values based on quantities that never vary—see Constants on page 751.
Time Functions	Returns values based on time—see Time Functions on page 751.

Expression Table	Description
Profile Functions	Returns values based on curve profiles—see Profile Functions on page 751.
Conditional Functions	Returns values based on explicit conditions—see Conditional Functions on page 752

Arithmetic Operators

Use the following arithmetic operators to perform basic mathematical operations.

Arithmetic Operator	Description
+	Addition
-	Subtraction
*	Multiplication
/	Division
%	Integer Modulo
^	Exponentiation

Comparison Operators

Use the following operators to compare two values with each other. When two values are compared using these operators, the result is 1 if the comparison is true and 0 if the comparison is false.

Comparison Operator	Description
==	Equal to
!=	Not equal to
<	Less than
<=	Less than or equal to
>	Greater than

Comparison Operator	Description
>=	Greater than or equal to
&&	Boolean AND
||	Boolean OR
!	Boolean NOT

Operator Precedence

When you combine several operators in a single expression, the operations are performed in the following order.

Order	Operator	Description
1	!	Boolean NOT
2	-	Negation (as in -1)
3	*, /, ^, and %	Multiplication, division, exponentiation, and modulus
4	+ and -	Addition and subtraction
5	<, <=, >, and >=	Comparison
6	== and !=	Equivalence
7	&&	Boolean AND
8	||	Boolean OR

Math Functions

Function	Description
Abs (number)	Returns the absolute value of a given number.
Sign (number)	Returns the sign of the given number.
Pow (number, power)	Returns a number to the power of an exponent.

Function	Description
Sqrt (number)	Returns the square root of a given number.
Mod (number, divisor)	Returns the remainder from dividing one number by another.
Average (number1, number2)	Returns the average of the given values.
Log (number)	Returns the logarithm to base 10 of the given value (for base -2 log, use Ln).
Ln (number)	Returns the natural logarithm of the given value.
Exp (number)	Returns the exponential of the given value.
Min (number1,number2)	Returns the smaller of the given numbers.
Max (number1, number2)	Returns the larger of the given numbers.

Vector Functions

Function	Description
(expr1, expr2, expr3)	Returns a vector composed of the evaluation of the individual component expression.
Dot (vector1, vector2)	Returns the dot product of two vectors.
Cross (vector1, vector2)	Returns a vector, the cross product of two vectors.
Length (vector)	Returns the length of a given vector.
LookAt (targetPos,ObserverPos, AlignVector)	Returns a rotation vector based on a target position, an observer position and an align direction vector.
Align (targetPos, alignVector, bankingAngle)	Returns a rotation vector such that a designated axis of an object is aligned with the direction of the object's movement. The banking angle controls the rotation around the axis, in radians.

Function	Description
Distance (pos1, pos2)	Returns the distance between the two points represented by the vectors pos1 and pos2.
relangle	relangle (point1, point2, delta1, delta2): returns the angle between two vectors (point1,point2) and (point1+delta1,point2+delta2). All parameters are 2D vectors.This is used for implementing rotation tracking.
relscale	relscale (point1, point2, delta1, delta2): returns the ratio between the length of vector (point1+delta, point2+delta2) and (point1, point2).All parameters are 2D vectors.This is used for implementing scale tracking.

Random Number Functions

Function	Description
Rand (seed)	Returns a random value between 0.0 and 1.0, based on the given seed.
TrueRand (low, high)	Returns a truly random value between two given numbers. The sequence of returned values constantly changes, never producing past results.
Noise (seed)	Returns a random value between -1 and 1, based on a given seed.
Turbulence (seed, level)	Returns a random value between -1 and 1, based on a given vector and with the ability to control the level of smoothness of the resulting curve.
Noise3 (seed)	Returns a random vector for all elements in a vector between -1 and 1.
Turbulence3 (3d_seed,level)	Returns a random vector with all the elements in the vector between -1 and 1, with the ability to control the level of smoothness of the resulting curve.

Rounding Functions

Function	Description
Round (number)	Returns a number rounded to the nearest integer.
Ceil (number)	Rounds a number up to the next integer value regardless of its value.
Floor (number)	Rounds a number down to the next integer value regardless of its value.
Trunc (number)	Returns the integer value of a number by truncating its fractional part.

Trigonometric Functions

Function	Description
Degrees (angle)	Converts angle units from radians into degrees.
Radians (angle)	Converts angle units from degrees into radians.
Cos (angle)	Returns the cosine of a given angle in degrees.
Sin (angle)	Returns the sine of a given angle in degrees.
Tan (angle)	Returns the tangent of a given angle in degrees.
Acos (number)	Returns the arc-cosine (in degrees, from 0 to 180) of a given number.
Asin (number)	Returns the arcsine (in degrees, from -90 to 90) of a given number.
Atan (number)	Returns the arctangent (in degrees, from -90 to 90) of a given number.
Atan2 (X, Y)	Returns the arctangent of y/x, using the signs of both arguments to determine the quadrant of the return value. The returned angle is given in degrees within the range 180 to -180.

Constants

Constant	Description
PI	Archimedes' Constant (PI = 3.14159265358979323846264338327950)
E	Natural logarithmic base (E = 2.71828182845904523536028747135266)

Time Functions

Function	Description
t	Returns the current time in seconds.
Eval	Returns the value of a given expression at another point in time.

Profile Functions

Function	Description
Bell (center, height, width)	Returns the Gaussian (normal distribution) function, also known as a bell-shaped or bell curve, based on the center time, height and half-height-width of the curve.
EaseIn (start time, end time)	Returns an increasing Hermite-like transition S-curve between two given times.
EaseOut (start time, end time)	Returns a decreasing Hermite-like transition S-curve between two given times.
Sinus (period, amplitude,offset)	Returns a sinus profile curve based on the given period, amplitude and offset values.
Square (period, amplitude,offset)	Returns a square profile curve based on the given period, amplitude and offset values.

Conditional Functions

Function	Description
If(expr1, expr2, expr3)	Returns the value of the second or third expression, depending on the evaluation of the first one: if expr1 evaluates to true, returns to expr2; returns expr3 otherwise. Expr1 is a boolean expression. Expr2 and expr3 can be two values, two vectors, or two boolean expressions. This enables users to piece together snippets along the time domain.

Tracking and Stabilizing

32

Topics in this chapter:

About Tracking and Stabilizing

Understanding how to use the Tracker tool and apply the tracking data to elements in a composition, and determining to which parameters you want to apply the tracking data are key factors for producing basic to complex effects. The more experience you gain from using the Tracker tool and applying tracker data to elements, the more you see all the effects that trackers can produce. If your composition is jittery, use the stabilizing feature within the Tracker tool or the Panner tool to remove, for example, unwanted camera jitter.

Tracking and stabilizing are often processes of trial and error. You should track or stabilize with default settings, and if the tracker box strays, fine-tune the analysis.

Much of the tracking and stabilizing work you do will be done using the Reaction supertool—see Reaction Compositing and Effects on page 240, and the 2D Transform Tool—see 2D Transform Tool on page 432.

Tracking Concepts

The Tracker is a tool for following and capturing the motion of an on-screen feature, and then applying that stored motion to a second element, so that the two appear to be locked together.

You can track any number of features in a length of footage:

1-point tracking Tracks a single feature for position.

2-point tracking Tracks two features, using the relationship between the two to track scale/rotation.

4-point tracking Tracks four features, also referred to as corner pinning.

Multiple-point tracking Tracks as many features as you want.

You can apply the stored motion to various elements:

Center point of an object For example, you could track a halo over the head of a person in a clip. Assume that in the clip a woman tilts her head and advances toward the camera. You would want the halo to reflect the position, scaling, and rotational changes of the tracked features on the woman's head.

NOTE When you track scaling, you are not performing 3D tracking in true Z-space. You are simply measuring the amount of change over time in the size of the tracked object.

Control points of an object The shape of the object changes to reflect the motion of the reference feature its control points are locked to. For example, if you want to replace the label on an actor's clothing, each corner on the new label would track a corner on the old label. The shape of the new label would change to correspond to those movements.

Stabilizing Concepts

Stabilizing is a powerful process that represents one of the most common uses of the Tracker.

In most tracking the Tracker applies the reference motion to the desired element so that the two are locked. In stabilizing, the Stabilize filter uses the Tracker data to invert the analyzed motion to correct unwanted camera or subject motion.

You can stabilize any number of features in a composition:

1-point stabilizing The positional (location in X and Y) motion you want to eliminate is vertical, horizontal, or both vertical and horizontal.

2-point stabilizing The motion you want to remove involves scaling/rotation. It may also involve vertical/horizontal shifts.

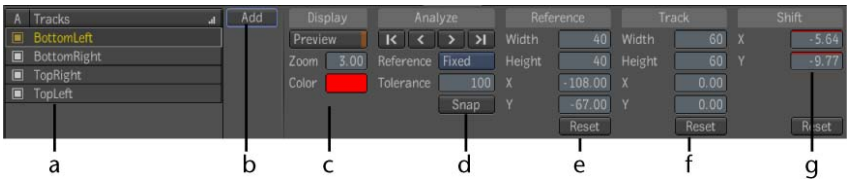
With stabilizing, the shift values are inverted, for example, (3, -2) would become (-3, 2) before they are applied. This inversion results in an elimination of the motion.

The Stabilize filter uses the Tracker data to move the image so that the reference feature occupies the same position it did in the preceding frame. In other words, the reference feature stays still because the frame is being moved.

4-point stabilizing By adding two extra Tracker Analyzers you can perform 4-point stabilization and eliminate translation, rotation, scale, and pivot motion.

Tracker UI

The Tracker UI contains the following parameters:



(a) Tracker browser (b) Add Analyzer (c) Display controls (d) Analyze controls (e) Reference box controls (f) Tracker box controls (g) X and Y shift controls

Use:	To:
Tracker Browser	View a list of all tracker analyzers you add to the tracker. Right-clicking any tracker analyzer to display a list of options to delete, rename or select all tracker analyzers.
Add	Add tracker analyzers to the tracker.
Display Preview	View the most recent match found for the tracker. Clicking it again disables the control.
Display Zoom	Determine how much the reference area appears magnified in the reference box by setting the Zoom factor.
Display Color	Change the color of the tracker and reference boxes by clicking the color pot, then selecting a new color from the color picker.
Analyze Forward	Run the analysis forward. Click again to turn tracking on and off.
Analyze Backward	Run the analysis backward. Click again to turn tracking on and off.
Analyze Forward one frame	Advance the analysis forward by one frame.
Analyze Backward one frame	Advance the analysis backward by one frame.
Fixed Reference	Leave the contents of the reference box unchanged during tracking. This is faster than Roaming. It is also less flexible since it assumes little or no change to the reference feature.
Roaming reference	Update the contents of the reference box at each frame. Use this option when the reference feature undergoes significant changes in luma or shape cues during the footage. For example,

Use:	To:
	if the reference feature undergoes lighting changes, choosing Roaming increases the chances of finding a match at each frame.
Tolerance	Set how much tolerance for error there is in locating a match for the reference box. A keyframe is only set for the frame if a match is found. If Tolerance is set to 100%, a match is always found, regardless of how wrong it may be. If Tolerance is set to 0%, a keyframe is only set for the frame if an exact match for the reference box is found.
Snap	Update the contents of the reference box at the current frame.
Reference	Change the size and position of the reference box.
Tracker B	Change the size and position of the tracker box.
Shift	Change or reset the shift values.

Tracking Workflow

Tracking workflow usually consists of the following steps:

- Choosing a reference point.
- Positioning the reference box.
- Resizing the reference and tracker boxes.
- Resetting the reference and tracker boxes (optional).
- Analyzing the footage.
- Correcting errors.

Adding a Tracker Tool to the Dependency Graph

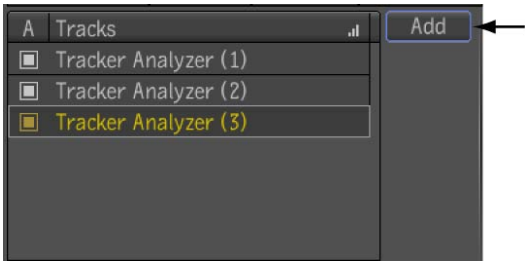
Once your project folders, project preferences, layout (including the Player Options) are set up, and the media is imported and opened, you are ready to begin your tracking task.

To add a Tracker tool to the dependency graph:

➤ Do one of the following:

- From the Tools tab, drag the Tracker tool to Schematic.
- From the Tools tab, open the Tracker folder and drag the Tracker Analyzer tool to Schematic. You do not need to attach it directly to the dependency graph but rather to the image sequence you want to analyze as it is not going to be modifying the image but rather you will be using data associated with that tracker.

Unlike the Tracker Analyzer, the Tracker lets you add multiple Tracker Analyzer tools. If you added the Tracker tool and you want to add more than one Tracker Analyzer, click Add in the File Browser.



You can also add Trackers and Tracker Analyzers to a dependency graph using the Create Trackers feature found in any tools with position, rotation, scale, and pivot parameter controls. These tools include the Panner, 2D Transform, Reaction, Warp 2D, Garbage Mask, and Remove Dust. Furthermore, shape tools (Warp 2D, Garbage Mask, and Remove Dust) also let you create Trackers on a per point basis.

The Create Trackers toggle button is found in the Tracker Selection Dialog. It only applies if you are going to press the Link button. Ordinarily the Tracker Selector Dialog allows you to select a set of 1 or more tracker analyzers to link or copy to. However, if Create Trackers is selected, then when you press Link, it will actually create the required number of tracker analyzers. Where it creates them depends on whether you have a tracker tool selected or not. If one is selected, it creates the new analyzers in that tool, otherwise, it creates a new tracker tool, with the required number of analyzers. In the following example procedure, the 2D Transform tool is used to create the Tracker and required number of tracker analyzers specified by the Transform Type selected.

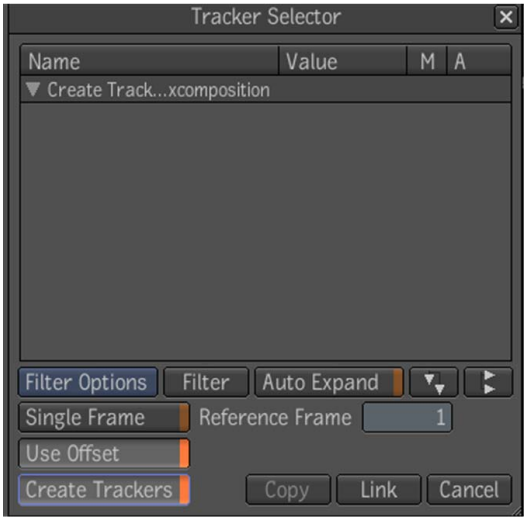
To create Trackers and Tracker Analyzers for 4-point tracking using the Create Trackers feature:

1. Add a 2D Transform tool to the dependency graph.
2. Select 4 Point from the Transform Type menu.

3. Right-click the Source label and select Set Trackers.

The Tracker Selector window is displayed.

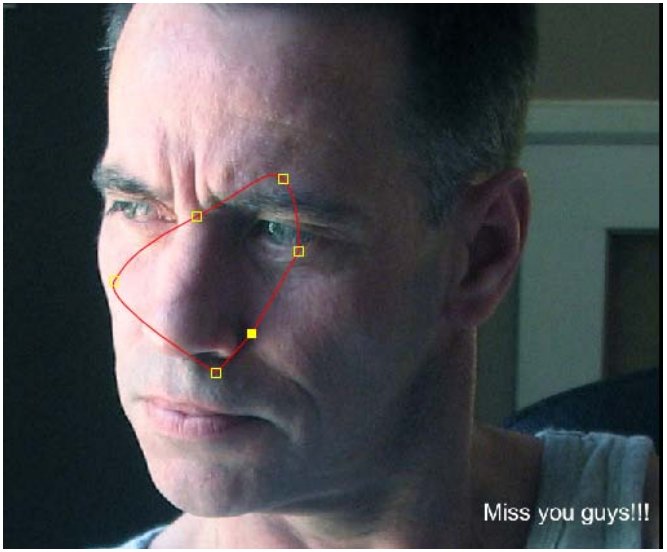
4. Toggle the Create Trackers button and click Link.



A Tracker with four Tracker Analyzers is created in the dependency graph.

To add a Tracker and Tracker Analyzer for a single point in a shape (Warp 2D, Remove Dust, Garbage Mask):

1 Select the point you want tracked in the Player.



NOTE If you want to track all the points in the shape, use **Ctrl** + **A** to select all the shapes while your cursor is in the player.

2 Right-click the point and select Set Trackers.

The Tracker Selector window is displayed.

NOTE You can also right-click the Position label under the Point # label to display the Tracker Selector window.

3 Toggle the Create Trackers button and click Link.

A Tracker and Tracker Analyzer is created in the dependency graph.

NOTE If you selected all the points in the shape, a Tracker Analyzer will be created for each point.

Choosing a Reference Point

A good reference point is a high-contrast pattern that has good definition both vertically and horizontally, which allows for perfect registration in both directions. Preferably, the reference point should not change much over time.

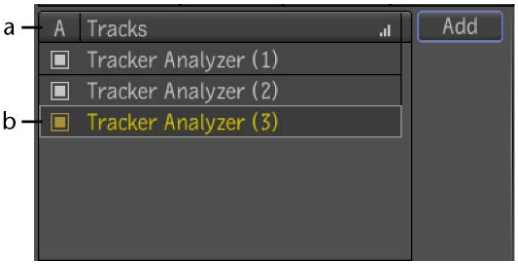
To choose a good reference point, play the footage several times to become familiar with the material. Ideally, you should try to find a pattern that is present in every frame. However, this is not always possible, and you may have to track two different features, such as object that disappears behind another one, or an object that moves out of the frame. The reference frame (the frame on which you place the reference box) should be the frame where the reference point is most representative in terms of shape, size, and rotation. Generally, you position the reference box over the reference point in the first frame of the sequence, but you could also place it on the last frame and analyze backwards. Another alternative is to place it on a frame in the middle of the footage and analyze from that frame backward to frame 1, and then analyze from that frame forward to the end of the footage.

Positioning the Reference Box

When you position the reference box, the tracker box automatically moves to the same location so that when you start analyzing, both boxes are positioned over the reference point.

To position the reference box:

1. Go to the frame where you want to position the reference box.
2. If the tracker you want to position does not appear on the image, click the appropriate Tracker in the File Browser, and enable Active.



(a) Active column (b) Selected Tracker Analyzer set to Active

NOTE Each tracker is automatically assigned a different color. However, you can customize the color at any time—see Changing the Color of a Tracker on page 768.

3. Position the cursor to the center of the reference box until it changes from a cross to a multi-arrow shape.

4 Click and hold the reference box and drag it over the pattern you want to track (in the example below, the rider's head). The reference box is magnified to make it easier to select a position. By default, the tracker box stays with the reference box.

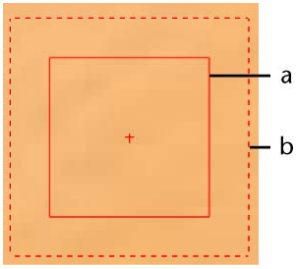
Resizing the Reference and Tracker Boxes

You can resize the reference box to make it surround the feature you chose to track more closely. By only including the recognizable feature in the box

with no other details that may change throughout the footage, you minimize the possibility of the tracker losing the reference point.

NOTE Sometimes a small reference point does not give good results. Try enlarging the box to include more of the selected feature. Note that the larger the box, the slower the analyzing speed.

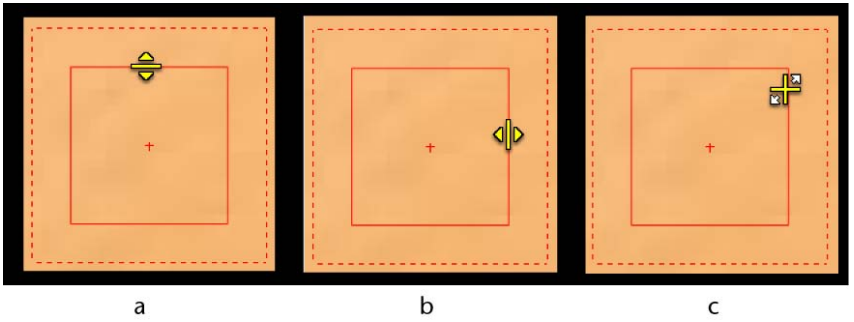
Resize the tracker box, so that it is large enough to accommodate the most frame-to-frame movement of the reference point. For example, if the movement of the reference point is mostly horizontal, you can increase the width and reduce the height of the tracker box. If there is a large amount of movement both horizontally and vertically, increase both the width and the height of the tracker box.



(a) Reference box (b) Tracker box

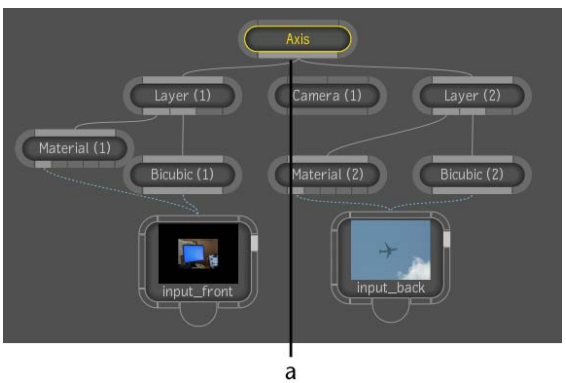
To resize the reference and tracker boxes:

➤ Do one of the following:



(a) Resize height cursor (b) Resize width cursor (c) Resize proportionally cursor

- To resize a box interactively on the frame, position the cursor on the edge of the box you want to resize. The cursor changes shape. Drag the box to the desired dimension.
- To use the Tracker or Tracker Analyzer menu to move or resize the reference or tracker box for the current tracker, enter values in the appropriate fields.



(a) Reference box position and size fields (b) Tracker box position and size fields

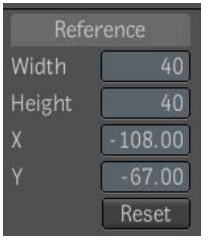
Resetting the Reference Box

If you decide to choose a different reference point, you can reset the reference box.

To reset the reference box:

➤ Do one of the following:

- Click Reset at the bottom of the Reference label portion of the UI.



The reference box size is reset to its default values and it is positioned in the center of the Player window.

- Type values into the respective value fields and click **Enter**.

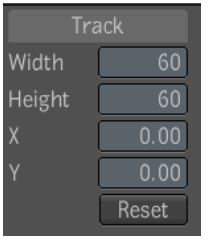
Resetting the Tracker Box

By default, the tracker box stays with the reference box as you position it. Generally, you do not need to move the tracker box separately from the reference box. If you separated the tracker box from the reference box and want to reposition it over the reference box, you can reset the tracker box.

To reset the tracker box:

➤ Do one of the following:

- Under Track, click Reset.



The tracker box size is reset to its default values and it is positioned in the center of the player window.

- Type values into the respective value fields and press **Enter**.

To reset the shift values:

➤ Do one of the following:

- Under Shift, click Reset.

  The shift values are reset to their default values.

- Type values into the respective value fields and press **Enter**.

To reset a tracker:

1. Select the tracker from the File Browser.
2. Click Reset in the Tool Options.

The size and position of the reference box, tracker boxes, and the X and Y Shift values are reset.

Changing the Color of a Tracker

Each Tracker Analyzer is automatically assigned a different color. However, the color may be obscured by existing colors in the footage you are tracking. You can change the color of the tracker to make tracking easier.

To change the color of a tracker:

1. In the File Browser, click the Tracker that corresponds to the tracker you want to change.
2. Under Display, click the color pot next to Color.

   The color picker appears.
3. Select a new color that can be easily seen in the footage and click Set.

   The tracker changes color.

How the Tracker Works

The Tracker first tracks motion of the reference feature(s), then applies or inverts that motion, depending on if you are tracking or stabilizing.

Once you have set up your trackers, the analysis can begin. This is how the Tracker analyzes and remembers the motion of the reference feature:

- The Tracker stores the position of the reference box on frame 1.
- The Tracker advances to the next frame and positions the tracker box in the same place as in the previous frame.
- It searches the tracker box for an area that matches the contents of the reference box. The Tracker uses luma cues (white versus black) and shape cues to determine what is a match.
- Once it finds a match, the Tracker uses the location of the match to move the center of the tracker box over the center of the match. This change in position is stored as an X and Y shift value.
- On the next frame, and on all future frames, the Tracker gets a little smarter about where it positions the tracker box. It extrapolates the likely position of the reference feature based on the amount and direction of the motion that occurred between the last two frames. The Tracker makes an educated guess of where the feature may be, based on past matches.

  NOTE This means the Tracker works best when the motion of the reference feature is regular and does not suddenly or erratically change direction or speed.

- Once the Tracker has analyzed the motion of the reference feature, and stored it as Shift values for each frame, it copies the motion of the reference

feature to the element that you are tracking, so that the two appear locked together.

In all tracking except for stabilizing, the Tracker applies the shift values without modification.

With stabilizing, the Stabilizer operator inverts the shift values, resulting in a negation (and therefore elimination) of the motion—see Stabilizing on page 783.

Tracking Position

In this example, you want the object to track only the changes in position of the reference feature. The object follows the X and Y motion without undergoing changes in scale or rotation.

For example, you have footage of a bird flying and you want to attach a small blue arrow to the to top of the bird's wing. You can add an image of an arrow from the File browser to the footage. You need to track the arrow to one point on the wing. Note in this example there are no scale or rotation changes to the wing.

NOTE Although this example uses the 2D transform tool, there are many other tools that have position and translation parameters that can be tracked, such as Reaction, Panner, the Garbage Mask, Remove Dust tool, and the Warp 2D tool.

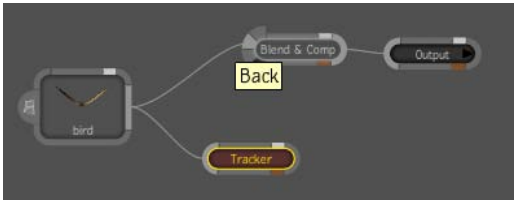
To track position:

1. Play the footage a few times to evaluate what reference feature you want to use and if there are any foreseeable problems for the tracking.
2. Determine a good reference frame. In this example, it is the first frame of the footage.
3. From the Tools tab, drag a Tracker Analyzer tool to Schematic. You do not need to attach it directly to the dependency graph as it is not going to be modifying the image but rather, you will be using data associated with that tracker. Once you have finished tracking, you can remove the tracker tool from the dependency graph.
4. Position the Reference and Analyzer boxes on the position you want tracked. In the case below, the tip of the bird's wing has been selected.

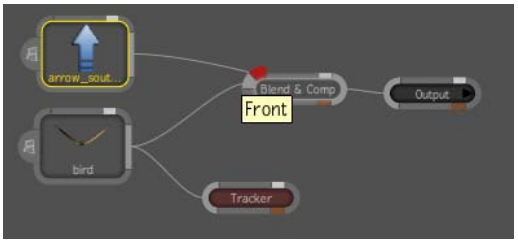
5 Click Analyze Forward to track reference position in the footage.

6 From the Tools tab, open the Composition folder and drag a Blend and Comp node to the dependency graph between the footage node and the Output node.

7 Connect the bird footage to the Back input of the Blend and Comp node.



8 The image of the arrow is then dragged into Schematic and linked to the composition and attached to the Front input of the Blend and Comp node.



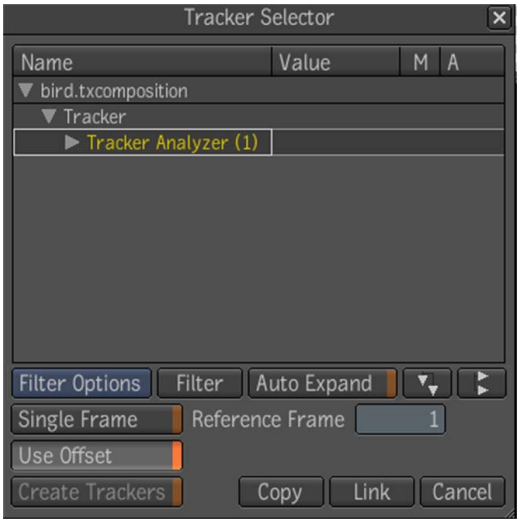
9 A 2D Transform tool is then added between the arrow image and the Blend and Comp node. Select the 2D Transform node.

10 From the 2D Transform tool UI, select 1 Point from the Transform Type menu. The Source and Destination coordinate value editors appear.

11 The Source X and Y coordinates are set to 0, 0 fixing the center of the arrow image exactly at the center of the tracker.

12 To set the Destination transform, right click on the Destination label and select Set Tracker. The Tracker Selector appears. Expand the Tracker and select the Tracker Analyzer.

13 Enable Use Offset and click the Link button.



The arrow is tracked to the tip of the bird's wing. Play the footage.

Tracking the Scale of an Object

In some cases, you may want the object to remain stationary and track only changes in scale. For example, assume that a camera is looking down a straight road at a car approaching. You need to track the license plate as the car approaches.

The Tracker requires two tracker boxes to measure scaling. The changing distance between the centers of the two reference boxes determines the scale.

In our example, you want to lock a new license plate to the old one. You would position one reference box over one corner of the license plate and the other reference box over the diagonally opposite corner.

The license plate is a layer. It assumes you have a Reaction node with two layers—the background layer and the source layer. In this example, the background is the car footage and the source is the license plate footage or object.

NOTE You can also use the 2D Transform, and the scale parameter controls of the Garbage Mask, Remove Dust and Warp2D tools.

To track changes in scale only:

1 Determine a good reference frame and position the background footage at that frame.

2 Select the Tracker Analyzer tool from the Tools tab and drag it to the Schematic. You do not need to attach it directly to the dependency graph as it is not going to be modifying the image but rather, you will be using data associated with that tracker.

3 Attach the composition to the input of the tracker.

4 Add a second Tracker Analyzer.

5 Resize and reposition each Tracker, so that they track the scaling change. (In this example, the Trackers are placed over the upper-right and lower-left corners of the license plate.)

6 Adjust the settings for each Tracker.

7 Select the first analyzer from the File browser and click Analyze to start the analysis.

8 Return to the first frame.

9 Select the second analyzer from the File browser and click Analyze to start the analysis.

10 Drag a Reaction tool from the Tools tab and place it in the dependency graph between the footage node and the Output node.

11 With the cursor over the Reaction node, right-click and select Add Source.

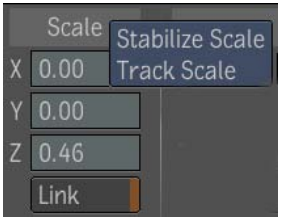
12 At the bottom of the Layer Editor, select Bilinear and click Create.
A new layer is created.

13 Attach the second composition (or object) as source 1.

14 Go back to the first frame.

15 Click the Reaction tab.

16 Right-click Scale and select Track Scale.

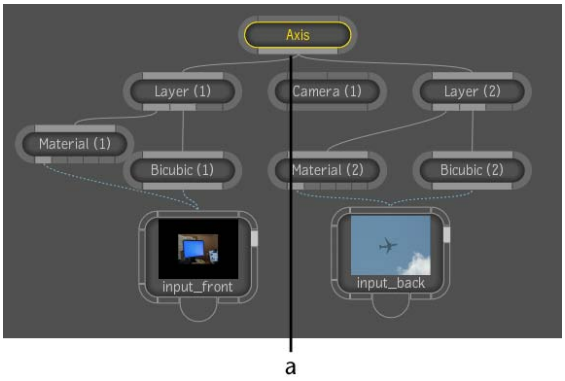


17 Select both Tracker Analyzers using **Ctrl** + click, and click Link.

An expression is automatically created linking the tracking data to the scaling of the new license plate.

18 Play the footage to view the result.

NOTE When tracking data needs to be applied to more than one layer in a composition, add an extra axis node to the dependency graph and connect it to the appropriate layers. Linking the axis to several layers facilitates panning multiple objects inside a reaction. In this case you would then click the axis node instead of the Layers node at step 15, and right-click the Scale label and continue the procedure.



(a) Axis node connected to two layers inside Reaction.

Tracking the Rotation of an Object

In some cases, you may want the object to track only a rotational change. The Tracker requires two tracker boxes to track rotation. The first Tracker goes over the pivot point of the rotation you want to track. You place the second over an area that describes the axis of the rotation you want to apply to the object.

NOTE You can also use the 2D Transform, and the rotation parameter controls of the Garbage Mask, Remove Dust and Warp2D tools.

To track changes in rotation only:

1 Determine a good reference frame and position the footage at that frame.

2 Select the Tracker Analyzer tool from the Tools tab and drag it to Schematic. You do not need to attach it directly to the dependency graph

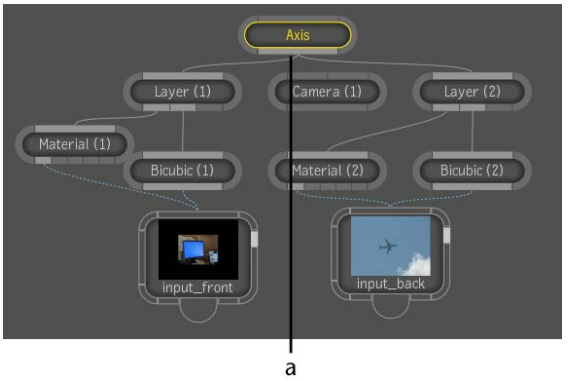
as it is not going to be modifying the image but rather, you will be using data associated with that tracker.

3. Attach the composition to the input of the tracker.
4. Add a second Tracker Analyzer
5. Place the first Tracker over the pivot point of the rotation you want to track.
6. Click Analyze Forward to analyze the footage.
7. Go back to the first frame.
8. Place the second tracker over an area that describes the axis of the rotation you want to apply.
9. Click Analyze Forward to analyze the footage.
10. Drag a Reaction tool from the Tools tab and place it on the dependency graph between the footage node and the Output node.
11. With the cursor over the Reaction node, right-click and select Add Source.
12. To associate a layer with the source you just created, select Bilinear at the bottom of the Layer Editor and click Create.
13. In the Layer Editor, right-click inside the Z field under Rotation and select Track Rotation.

    The Tracker Selector browser appears.
14. Using **Ctrl** + click, select both the trackers then click Link.

    The rotation has been tracked.

NOTE When tracking data needs to be applied to more than one layer in a composition, add an extra axis node to the dependency graph and connect it to the appropriate layers. Linking the axis to several layers facilitates panning multiple objects inside a reaction. In this case, you would then click the axis node at step 13 instead of the Layers node, and continue the procedure.



(a) Axis node connected to two layers inside Reaction.

4-Point Tracking

In order to correctly project images onto televisions, bulletin boards, and other quadrilaterals in an image sequence use the 4-Point tracking workflow.

With four-point tracking (also known as corner pinning), you use four trackers to generate tracking data for anchoring the four corners of a bilinear surface to background footage.

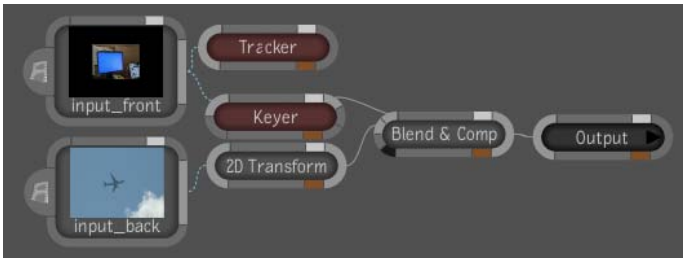
The reference points you use must be well-defined; it is recommended that you plan them when shooting the sequence (for example, add markers to the scene). It is not always possible to do four-point tracking when the reference points are not well-defined.

In the following example, the goal is to pin an image of a jet to the screen of a monitor as the camera moves.

NOTE You can also use the Reaction tool, and the position and translation parameter controls of the Warp2D tool, and the position and translation vertices parameter controls of the Garbage Mask, and Remove Dust tools.

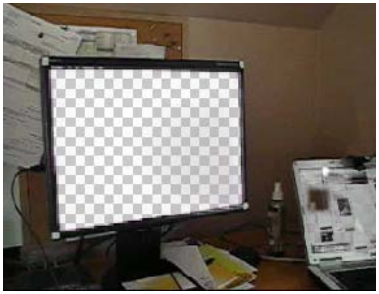
To corner pin and track a bilinear surface to background footage:

1 In the Schematic view, the basic dependency graph has been set up with the following nodes:

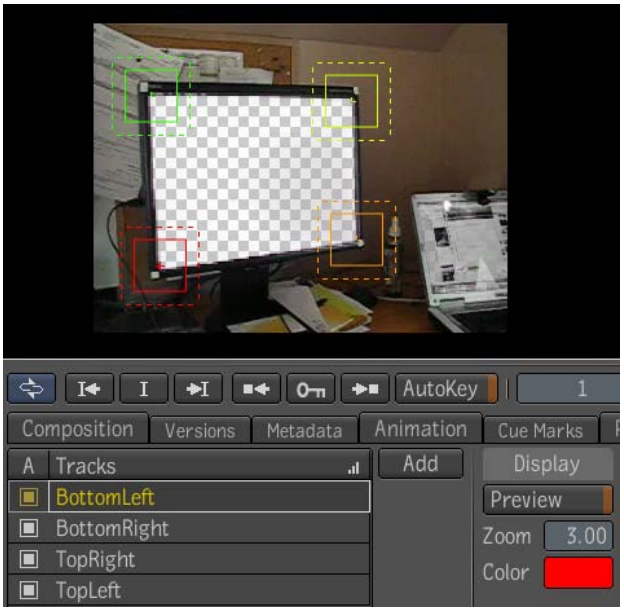


Node	Purpose
Input image (front)	This is the footage that will tracked.
Input image (back)	This image will be corner-pinned onto the front footage.
Keyer	The Keyer is used to remove the blue portion of the monitor.
Tracker super tool	The Tracker will track and analyze the movement of the four corners of the monitor. The Tracker super tool is used because multiple Tracker Analyzers are required.
Blend & Comp	The Blend & Comp node will combine the two processed images.
2D Transform	The 2D transform tool will apply the necessary transformations to the image to be corner-pinned.
Output	Outputs the final composition.

2 Remove the blue screen from the monitor with the Keyer.



3 Create four Tracker Analyzers and name them as per their corner positions, then position them in the locations that will make up the four corners to pin.



4 Adjust the Display, Analyze, Reference box and Tracking box settings.

5 Analyze each track (do not forget to reset the footage back to the start frame after each analysis). You can also select the first Tracker Analyzer and **Shift** + click the last one to select all trackers to analyze at the same time.

6 Connect the RGBA output from the Keyer to the Front input of the Blend & Comp node, and the output of the second image into the Back input of the Blend & Comp node.

7 Add a 2D Transform tool to the graph between the second image and the Blend & Comp node.

8 With the 2D Transform tool highlighted, select 4 Point from the Transform Type menu.

9 Click Fit To Source

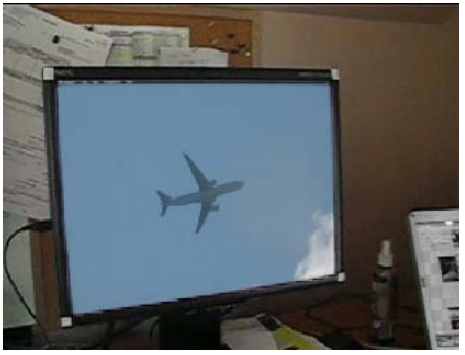
10 Right-click on the Destination label and select Set Trackers.

11 Select the trackers in the same sequence as you created them from the Tracker Selector window and activate the Use Offset toggle.

12 Click Link.

The bilinear image is pinned to the background image.

13 If the corner-pinned image is too small or too large for the screen it's replacing, create another transform and adjust its scale while in the SRT transform type.



Tracking Difficult Shots

This section provides different strategies that you can use to track difficult shots.

Positioning the Tracker Box Manually

When the reference point is temporarily covered by another object, position the tracker manually.

To manually position the tracker box:

1. Click the Analyze Forward button to stop the analysis.
2. Advance the clip to the frame where the reference point becomes visible again, and reposition the tracker box over the reference point.
3. Click Analyze to restart the analysis at this frame. Since the X and Y shifts are recorded as keyframes in channels, the translation values for the frames in which the reference point was covered are calculated. The final result will be a smooth motion.

Tweaking the Tolerance Value

The Tolerance value determines how exact a match the Stabilizer requires when identifying the reference point. With a tolerance of 100%, the Stabilizer accepts anything as a match. With a tolerance of 0%, the feature being tracked must be exactly the same as the reference point. If a match is not found, a keyframe is not set, and the position of the reference point is interpolated between the previous keyframe and the next keyframe.

If the reference point is temporarily hidden by another object, you can use the Tolerance value to make the tracker ignore the reference point in parts of the clip where it is hidden, then continue tracking it normally when it reappears. This can prevent or reduce the need to manually reposition the tracker box during an analysis.

To use tolerance to track a temporarily hidden object:

1. Analyze the footage with full (100%) tolerance.
2. The tracker loses the reference point when it is hidden. Press Backspace to delete keyframes and go back to the last bad keyframe. Lower the Tolerance value until the cross hair disappears from the tracker box. The keyframe at that frame is deleted. The cross hair indicates that a frame has a keyframe.
3. Click Analyze to continue the analysis. Further analysis automatically discards the undesirable keyframes, and the position of the reference point is interpolated until the reference point reappears.

Correcting Errors

If the tracker box strays from the reference point that it is supposed to be tracking, incorrect keyframes result. If such an error occurs, you can stop the analysis, correct it, and restart it at any frame.

There are a number of different methods you can try to correct tracking errors. Try the strategies given below, then redo the analysis to generate new keyframes based on the updated information you provide.

Updating the Reference Point

If the tracker loses the reference point, you can delete the incorrect keyframes, then, at the last good keyframe, update the reference point to the current contents of the tracker box.

To update the reference point to the tracker box contents:

1. Select the appropriate tracker by clicking the appropriate Tracker button.
2. Press **Backspace**.

   The last keyframe is deleted and the positioner moves back one frame. The tracker box is now on the previous keyframe.
3. If you want to remove this keyframe, press **Backspace** again.
4. Repeat step 2 for all incorrect keyframes.
5. When you reach the last good keyframe, click Snap. This updates the reference point from the one originally specified in the reference frame to the image currently inside the tracker box.
6. Click Analyze to continue the tracking process.

Resetting the Tracker

If the tracker loses the reference point early in the analysis, you may want to reset the entire tracker and find a new reference point—see Resetting the Tracker Box on page 767.

Adjusting the Tracker Box

If the Stabilizer cannot find the reference point within the boundaries of the tracker box during analysis, the tracker box strays from the reference point and produces incorrect keyframes. Although you can manually correct these keyframes, it is easier to make the tracker box large enough to accommodate the movement of the reference point. Note that processing time increases as the size of the tracker box increases.

To adjust the size of the tracker box:

1. Stop the analysis by clicking the Analyze Forward button.
2. Press **Backspace** until you reach the last good frame before the tracker box strayed from the reference point. Adjust the size of the tracker box, so that it is large enough to accommodate the largest frame-to-frame movement of the reference point.

   NOTE Backspacing deletes the keyframes as you go back. If you do not delete the keyframes, they will be overwritten when you reanalyze. Backspacing in this instance makes it easier to see the tracking path clearly.

3. Click Analyze Forward to continue the analysis.

Stabilizing

Stabilizing is a powerful process that represents one of the most common uses of the Tracker. Stabilizing is most often used to remove unwanted camera jitter.

The stabilizing workflow is almost identical to the tracking workflow except the data analyzed by the tracker is inversed to negate the unwanted movement in the footage.

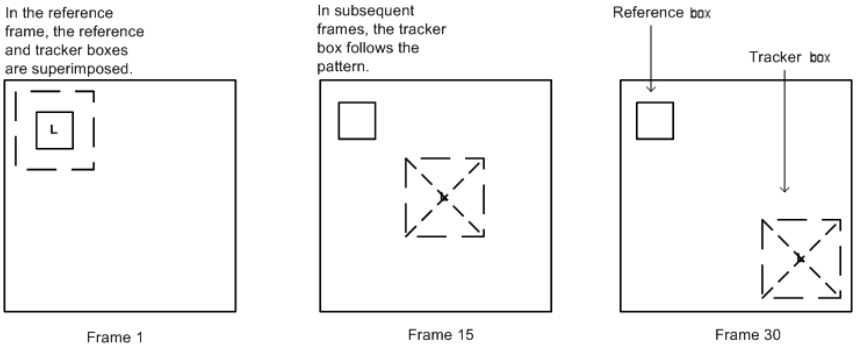
In stabilizing, link the Tracker data to the Panner tool to invert the analyzed motion and correct unwanted camera or subject motion in the X and Y directions.

How the Stabilizer Works

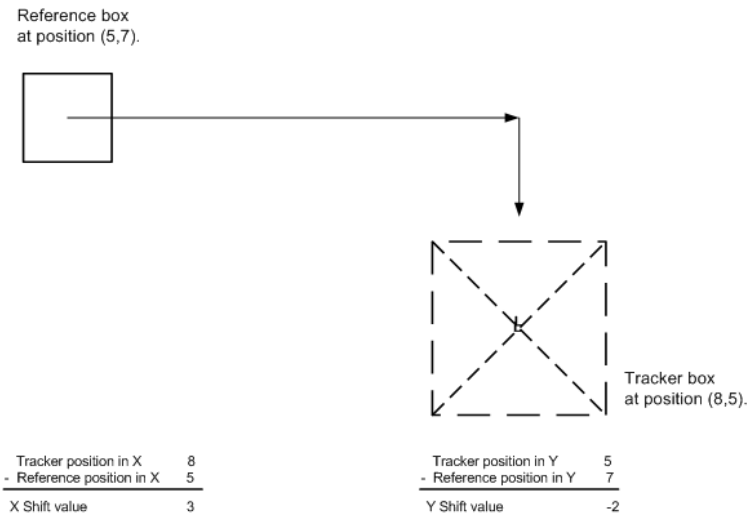
To start stabilizing, select one or more reference points on your clip. Locate the first frame containing the movement to be tracked (the reference frame).

In general, the reference frame is the first frame of the sequence. The choice of reference point depends on whether you are tracking or stabilizing. When tracking, the reference point is a feature you want to track; when stabilizing, the reference point represents the point around which the image is stabilized—see Choosing a Reference Point on page 762. Place the reference box(es) around the selected feature(s).

Once you have set the tracker positions, start the tracking process, also referred to as analyzing the footage. During the analysis, the tracker box associated with each tracker moves as the Stabilizer looks for a pattern that matches the reference in each frame of the clip.

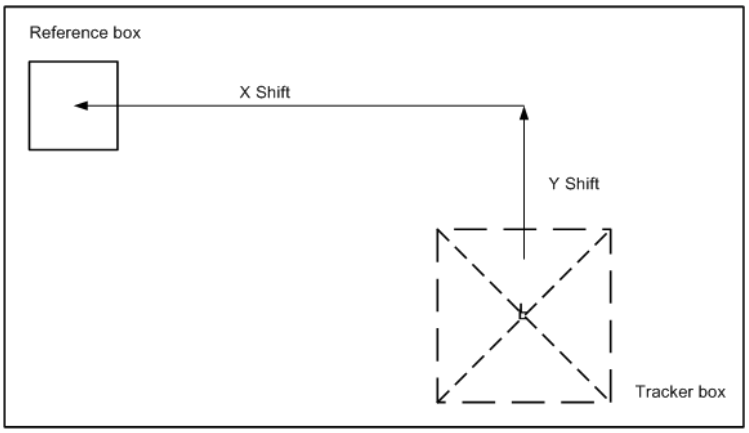


The Stabilizer calculates the difference between the position of the tracker box and the position of the reference box to produce X and Y Shift values. Shift values are a measurement (in pixels and subpixels) of how much the reference point has moved.

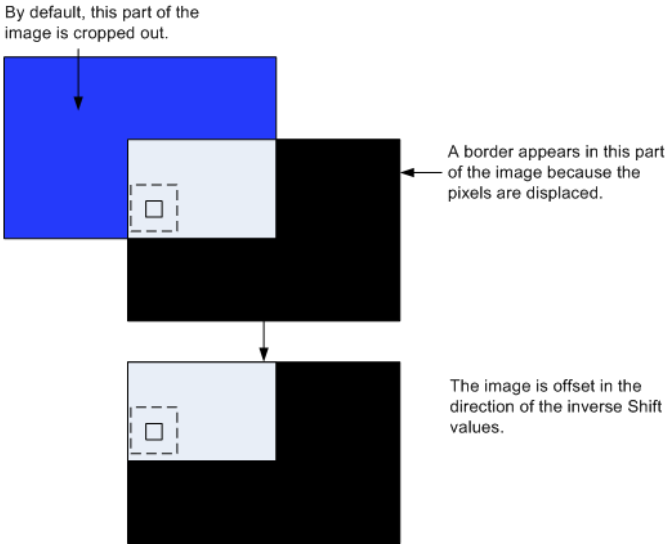


When the analysis is complete, you can fine-tune it if a tracker box has strayed from the reference it was supposed to follow. Once you are satisfied with the results, you can apply the data to the footage.

To track, the Stabilizer applies the Shift values as is. To stabilize, the Stabilizer inverts the X and Y Shift values in each frame of the sequence, and moves the image according to these values. This gives the impression that the reference point stays in the same position throughout the sequence. Because the image is moved during stabilization, a border appears on one or more edges, which means that some pixels are lost. The following illustrations show the process.



The image is moved so that the contents of the tracker box are brought back to the position of the reference box.



1-point Stabilizing

1-point stabilizing is useful when the motion you want to remove is horizontal/vertical (side-to-side/up and down).

Hand-held camera and helicopter shots nearly always have undesired horizontal or vertical motion jitter.

In the following example, the camera was bumped several times during filming which caused unwanted camera motion along the Y axis.

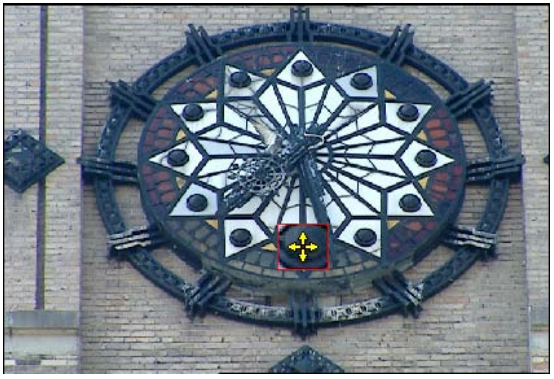
To perform 1-point stabilizing:

NOTE Although this example uses the Reaction tool, any tool that has position and translation parameter controls can be used to perform 1-point stabilization, such as the Panner tool, 2D Transform, Warp 2D and the vertices in the Garbage Mask and the Remove Dust tools.

---

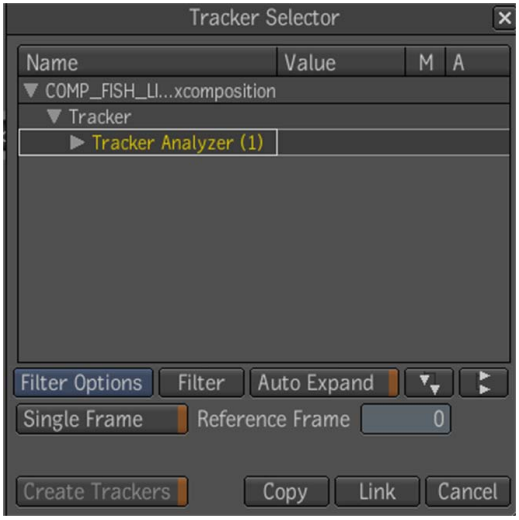
1. Play the footage to verify that you have a good reference feature.

   A decorative point at the six o'clock position was chosen as the reference feature.



2. Click Analyze Forward to track the selected reference position in the footage.
3. Drag a Reaction tool from the Tools tab and place it on the dependency graph between the footage node and the Output node.
4. With the cursor over the Reaction node, right-click and select Add Source.
5. To associate a layer with the source you just created, select the source layer from the Layer Editor. At the bottom of the Layer Editor, select Bilinear and click Create.
6. In the Reaction tab, under Position, right-click inside the Y field and select Stabilize Position Y.

The Tracker Selector appears.



7 Select the Tracker Analyzer from the list and click Link.

An expression is automatically created linking the tracking data to the stabilizing filter for position Y.

8 Play the footage to view the result.

2-point Stabilizing

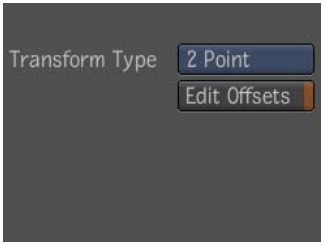
2-point stabilizing is useful when the motion you want to remove involves changes in scale/rotation (zoom/roll). For example, your camera may be zooming out from someone in a rocking chair.

As in tracking, when you choose to stabilize combinations of position, scale, and rotation simultaneously, the roles of each of the points are additive:

- If you track both scale and rotation, the first point indicates the pivot point of the rotation; the second indicates the arc of the rotation; and the distance between the two indicates the scaling changes.
- If you track position at the same time, the first point also indicates the vertical/horizontal motion you want to eliminate.
- If you track scale, choose two reference features that are at the same Z-depth throughout the shot. For example, if you have two rows of trees lining a driveway, choose two trees at about the same Z-depth (distance away from camera).
- If you track rotation, choose two reference features that are as far as possible from each other. This gives the Tracker the maximum amount of roll to analyze and correct.
- If you do 2-point stabilizing, each of the two reference features stays fixed throughout the stabilized clip. That is, if you are stabilizing scaling, the scaling change is eliminated. If you are stabilizing rotation, the rotation is eliminated.

To perform 2-point stabilization:

1. Track the video clip using a Tracker node with 2 trackers.
2. Connect the clip into a Transform2D node.
3. Set Transform Type to 2 Point.



4. Click on Set Source Trackers and choose the 2 trackers.
5. Click on Set Destination Trackers and choose the same 2 trackers, but select the frame number to which you want the clip stabilized. This will set the destination points to a non-animated value, positioning the stabilized clip to match the desired frame. Alternatively, you can just edit

the values of the Destination 1 and 2 boxes to create the desired stabilized position.

6 Click Link.

7 Play the clip. The unwanted motion is eliminated.

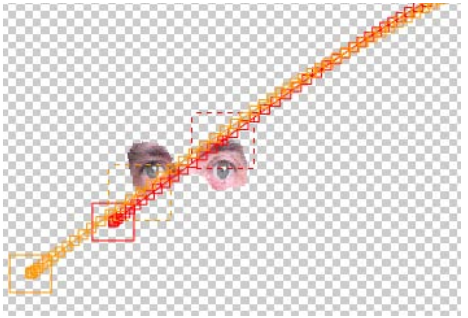
NOTE You can also perform 4-point stabilization by adding 2 extra tracker analyzers.

Simultaneous Stabilizing and Tracking

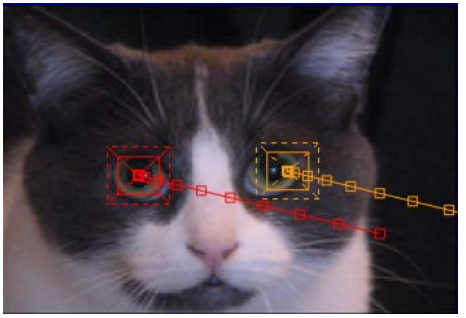
The fact that you can associate trackers with both source and destination points of the transform means that you can effectively do stabilization and tracking simultaneously. This is used to place one moving object in a clip onto a second moving object in a different clip.

To 2-point track one moving element onto another moving element:

1 Track the first video clip using a Tracker node with 2 trackers. In this example, footage of a pair of moving eyes was tracked.

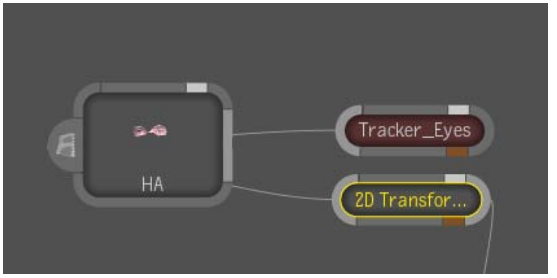


2 Track the second video clip using a Tracker node with 2 trackers. A moving image of a cat was tracked.

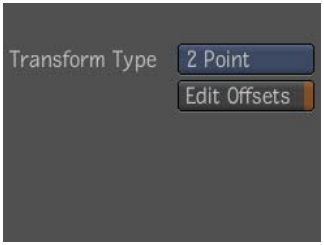


3 Track the second video clip using a Tracker node with 2 trackers.

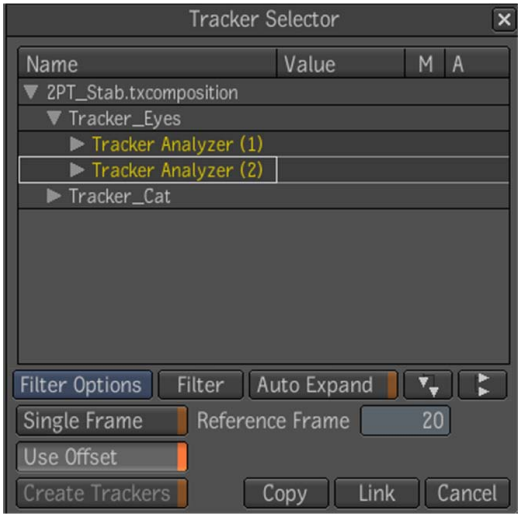
4 Connect the first clip into a Transform2D node



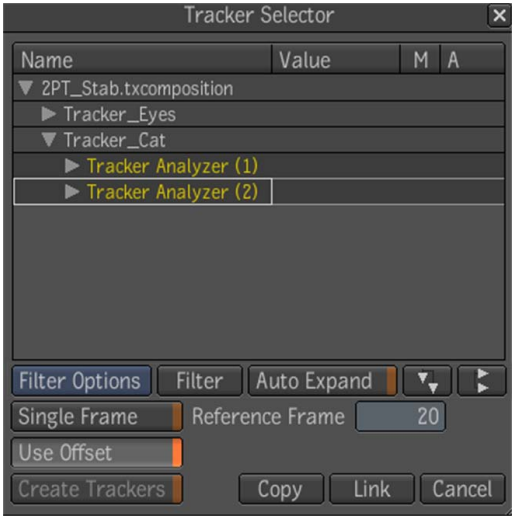
5 Set Transform Type to 2 Point.



6 Click on Set Source Trackers and choose the 2 trackers from the first video clip. Make sure Use Offset is toggled to active, then click Link.



7 Click on Set Destination Trackers and choose the 2 trackers from the second video clip. Make sure Use Offset is toggled to active, then click Link.



8 Add any correction using the Translation, Rotation, Scale parameters.

Video Tools

33

Topics in this chapter:

About Video Tools

Use the Video tools to perform the following operations:

- Inputting interlaced material and converting it to progressive.
- Outputting progressive material to interlaced format.
- Removing or adding 3:2 pulldown to convert the frame rate of footage.

Deinterlace

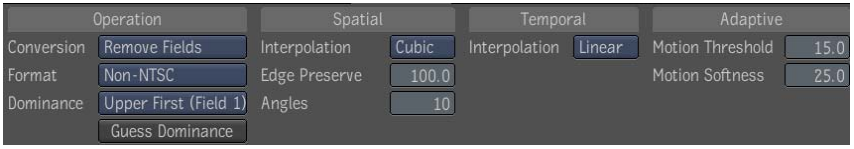
Video media is interlaced, meaning one frame consists of two fields representing different points of time. Since each field is sampled on only the odd or even scanlines, each field is missing half the pixels needed to make a full frame at that point in time. Deinterlacing is a process of interpolating the missing

information on fields to make full frames at particular points in time. A missing pixel can be interpolated by vertical interpolation of the field at the given time or by temporal interpolation using the previous and next field or a blend of the two. Adaptive video de-interlacing calculates a separate blend factor for each pixel based on the amount of local motion, so that vertical scanline interpolation is used in areas of motion, and temporal interpolation is used in static portions of the scene. This preserves detail in static portions of the scene while removing interlacing artifacts in areas of motion.

Use the Deinterlace tool to perform the following steps of the deinterlacing process:

- Determine the field dominance.
- Select fields (frame rate conversion).
- Adjust the adaptive threshold.
- Choose a scanline (spatial) interpolation method.
- Choose antialiasing parameters.
- Choose a temporal interpolation method.

The Deinterlace tool UI has the following parameters:



Field and format settings | Spatial interpolation | Temporal interpolation | Adaptive threshold

Determining Field Dominance

The field dominance is whether the top field comes before or after the bottom field in time. The top field means the field which contains the highest vertical scanline to be displayed.

To set the field dominance:

1 Set the format to either Non-NTSC or NTSC. If the format is NTSC the dominance will be labeled Upper First-Field 2 and Lower-First Field 1. If

the format is Non-NTSC the labels will be Upper First-Field 1 and Lower First-Field 2.

2. Set the value of the Dominance (upper or lower first).
3. If you do not know the field dominance, you can press the Guess Dominance button and it will be determined for you. Keep in mind that this may take several seconds to analyze frames and may take even longer if the input has a long sequence of static frames before the action. To stop the analysis, press the Cancel Guess button.

Field Selection

Basic video de-interlacing discards one field for each frame (say field 2) and uses some form of scanline interpolation to reconstruct the missing lines. Composite provides you with three choices when you convert interlaced media to progressive. You can opt to remove fields which outputs the first field of each frame, merge the fields which outputs a merge of the two fields of each frame, or change fields to frames which outputs all input fields separately as a frame (creates clip of twice the duration).

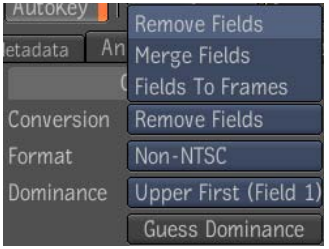
Depending on the parameter settings, the set of output frames will vary as described in the following table. The actual frame rate of the output is unchanged from the input. Each input frame consists of 2 fields, numbered individually as shown in the top part of the table.

	Input Frame 1	**Input Frame 2**	**Input Frame 3**	**Input Frame 4**
	1,2	3,4	5,6	7,8
Conversion	**Output Frame 1**	**Output Frame 2**	**Output Frame 3**	**Output Frame 4**
Remove Fields	1	3	5	7
Merge Fields	merge 1,2	merge 2,3	merge 3,4	merge 5,6
Fields To Frames	1	2	3	4

A Fields to Frames operation converts 30i into 30p having twice the duration of the input. If you desire to have 60p output, you may add a Rate Convert node after the Deinterlace node.

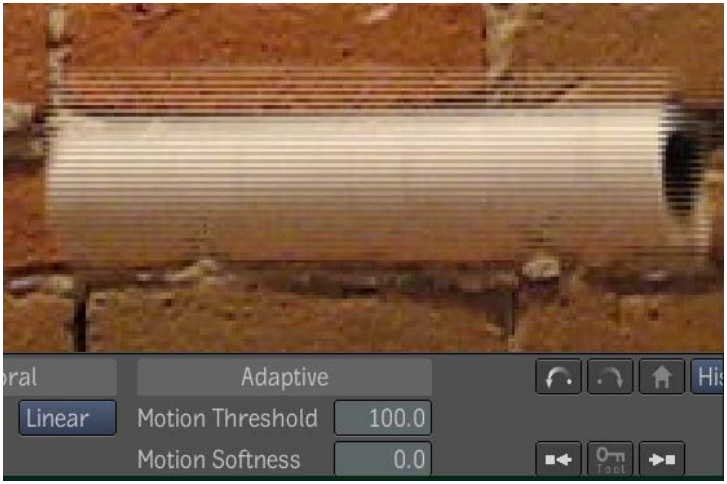
To choose a field selection method:

➤ Click the Conversion button and select a method from the list.



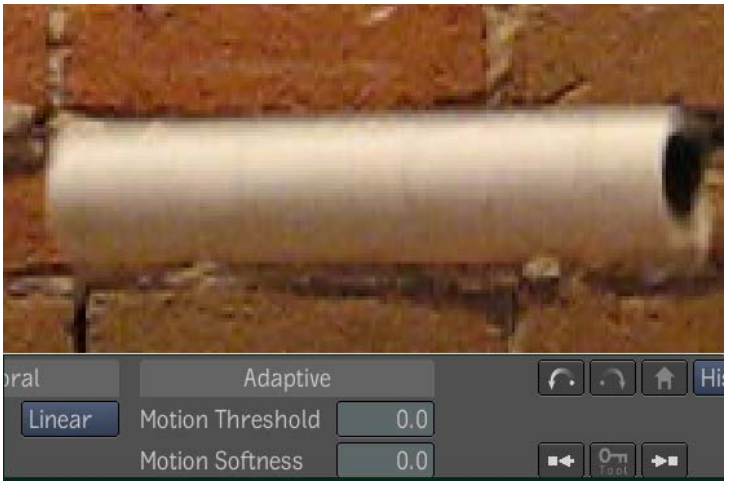
Adjusting the Adaptive Threshold

Adaptive deinterlacing is essentially controlled by the Motion Threshold, which gives a trade-off between interlacing artifacts and preservation of detail. This threshold indicates at what point a pixel is considered to be in motion. All pixels above this threshold are treated as moving and are interpolated spatially, meaning there is a loss of detail because every second line is thrown away. Pixels below this threshold are considered to be static parts of the scene and we preserve both fields of input, i.e., we use temporal interpolation. If you place the motion threshold at 100, then we consider all pixels to be static and output both fields, essentially not deinteracing, as seen in the following image:

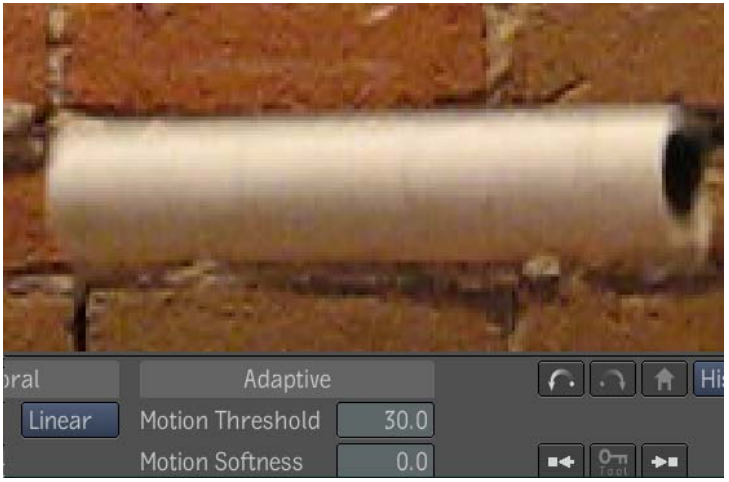


The above image has great detail in the static grey rock region (because it keeps both fields of input), but has objectionable interlacing feathering in the waving baton. If you push the threshold all the other way to 0, as in the following image, the interlacing artifacts disappear, but the static brick region loses

detail, because now all pixels are being vertically interpolated, after throwing away every second line.



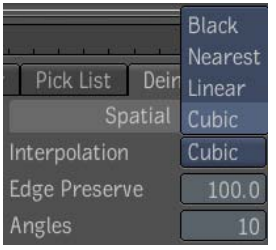
So you must move the slider down from 100 until all interlacing artifacts disappear, as in the following image, where the Motion Threshold is 30. Here we see that the baton has no interlace artifacts, but the details of the static brick areas of the scene are preserved.



The softness threshold can be used to make a smoother ramp between the thresholded regions, but since the internal calculation of motion already has a bit of blurring built in, you may not want to add more.

Setting Spatial Interpolation

The next step is to interpolate the missing vertical scanlines in areas of motion. You can select from four levels of quality with the default setting of Cubic being the best. Cubic is based on information calculated from four scanlines. Linear means to take the average of lines from two fields, whereas Nearest means to just copy the line from above or below. Selecting the Black parameter will display which pixels are being interpolated. This is useful when setting the Motion Threshold.



The Edge Preserve value field lets you set the amount of edge antialiasing you want to apply, while the Angles parameter is a performance/quality trade-off setting which effectively controls the amount of searching for appropriate edge direction. Increasing the number of angles sets the number of steps from 0 to 90 degrees, or the direction of the antialiasing.

Setting Temporal Interpolation

To create an output frame from a given input field, we copy the field, consisting of every second line of input, to the output frame. The remaining lines in the output must be interpolated somehow. Temporal interpolation is what is used in areas of the scene that are considered to be static (non-moving), as controlled by the Motion Threshold. The choices of temporal interpolation are Nearest or Linear. Nearest means to just copy the lines from the other field in the same input frame, whereas linear means to take the average of lines from two fields: the other field in the same input frame and the field either 2 fields before or 2 fields after that.

When should one use Linear and when should one use Nearest? The short answer is to always use Linear unless you need to preserve grain in static areas,

or if you want to do a traditional field merge. The long answer is that if the static region of the image is undergoing lighting changes, then you need to use Linear to avoid alternating lines having different intensities. If the static region is not undergoing lighting changes and is perfectly static, then you get the same result whether you use Linear or Nearest. That is why you should usually use Linear. However, if the static region has grain that changes from frame to frame, then using Linear will blur this a bit (being the average of 2 fields), so you may want to switch to Nearest.

The other time one might want to use Nearest for Temporal interpolation is when performing a traditional field merge. Here the user would set the Motion Threshold and Motion Softness both to 0.0, which disables the Adaptive deinterlacing. By setting the conversion to do Merge Fields and setting the Temporal Interpolation to be Nearest, the final result is exactly equivalent to a field merge, where each output line is a weighted sum of the corresponding input line, and the lines above and below it, with weights of 0.5, 0.25, and 0.25 respectively.

Interlace

Use the Interlace tool to convert a sequence of progressive frames to an output of interlaced video frames. Each pair of input frames result in one output frame.

To convert progressive frames to interlaced frames:

1. Select either NTSC or Non-NTSC from the Format menu.
2. Select the field order. If the upper field is first, then the upper field is taken from the first frame of the input pair and the lower field is taken from the second frame of the input pair; otherwise, upper and lower fields are reversed.

The output is always at the same frame rate as the input, and the action will play twice as fast. You can always use a Rate Convert node before the Interlace if needed. The following table shows how frames are interlaced. The input frames are numbered consecutively from 1 as shown in the top part of the table. Each output frame shows two numbers, which indicates the input frame indices for field 1 and field 2 respectively.

Input Frame 1 1	Input Frame 2 2	Input Frame 3 3	Input Frame 4 4
Output Frame 1 1,2	Output Frame 2 3,4	Output Frame 3 5,6	Output Frame 4 7,8

Pulldown

3:2 pulldown is the process of going from 24 frames per second of film to 60 fields per second of interlaced video (30 video frames per second). This is usually what a Telecine does when transferring film to video. Pulldown can also be applied to convert between 24 frames per second and 50 fields per second of interlaced video (PAL). In this case, it is referred to as PAL pulldown or 2:2:2:2:2:2:2:2:2:2:3.

The 3:2 process works on chunks of 4 film frames to produce chunks of 5 video frames (10 fields); given film frames A, B, C, and D, it produces video frames AA, BB, BC, CD, DD (the first and second letter correspond to what goes in the first and second field respectively).

To recover the original 24 frames per second of film and perform compositing work on progressive frames, you need to go through the reverse process, called 3:2 pullup. In order to do this, you need to know:

- **Cadence** At which video frame the repeating pattern starts.
- **Field Dominance** Which field comes first in time, Field 1 or Field 2?

The Pulldown tool will do both pulldown and pullup for 30 and 25 frames per second. The four possible operations are selected from the Conversion menu.

You can then select the pulldown cadence by choosing one of the following five patterns of whole (W) and split (S) frames from the Cadence menu.

Whole and Split frames are a consequence of using every second input frame to create three output fields and every other input frame to create two output fields, as in the following list of output field pairs.

11	12	23	33	44	55	56
W	S	S	W	W		

PAL Cadences

For PAL pulldown, 12 frames of progressive input become 25 fields of interlaced output. This means that out of every 12 input frames, 11 contribute to exactly 2 output fields, and 1 contributes to 3 output fields.

One example might be the following list of output field pairs where the 6th input frame is used 3 times:

11	22	33	44	55	66	67	78	89...

There are actually 25 cadence possibilities for 25i, as compared to only 5 in the NTSC 30i case. It is too awkward to refer to these cadences in the same way as for NTSC, for example: WSSSSSSSSSSSSWWWWWWWWWWWWW. Therefore, the 25 cadences have been labeled 1 to 25 in the Cadence drop-down menu where the number indicates the index of the split frame that follows a whole frame.

Autodetecting the Cadence

Similar to the Deinterlace tool, you can push the Guess Cadence button to analyze the sequence of interlaced frames to automatically set the proper cadence and dominance. Note that this assumes that the cadence and dominance do not change over the length of the input sequence. This button

would only be active if the operation is set to 30i to 24p, or 25i to 24p. This starts to search for which type of cadence (one of 5 or 25 possibilities) exists.

While analyzing, the Guess Cadence button will be labeled as “Cancel Guessing”, and provides you with a way out of the guessing analysis. As in the Deinterlace tool, a text label appears while guessing to indicate the current frame number being analyzed.

The frame rate of the output is modified to match the ratio of the conversion. For instance, the conversion 30i -> 24p will create output with a frame rate 4/5 of the input. Normally the input would be 30 fps and the output 24 fps, but the input can be any frame rate and the output will just be this frame rate multiplied by the appropriate ratio. Therefore the Pulldown is a true frame rate converter, unlike the Deinterlace and Interlace, and the Pulldown tool will not affect the apparent speed of action of the clip. The 30i to 24p conversion can also be used for 29.97i input, but the frames output will correspond to a frame rate of 23.98. If you wish to transform 30i into 23.98p, then a Retimer tool may be added.

Utilities

34

Topics in this chapter:

About Utilities

Utilities are analytic tools that you can use to view images without changing pixel data; they are display modifiers only. Utilities are available in the Utilities folder of the Tools tab.

The utilities tools available:

Tool	Description
Link Image	Lets you....
Output	Lets you view the output of your composition.
Comparison	Lets you compare images—see Comparison Tool on page 806.
Freeze Frame	Lets you....

Tool	Description
Notes	Lets you add notes to a composition—see Notes Tool on page 812.
Pass Through	An image tool that passes its input to its output unmodified—see Pass Through Tool on page 813.
Show Tiles	Lets you.....
Switcher	An image tool that allows you to switch between nodes—see Switcher Tool on page 814.

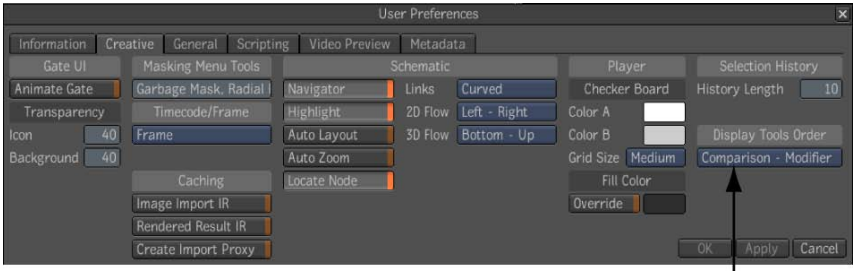
Comparison Tool

The Comparison tool is a viewer tool for comparing two images.

You can place the Comparison tool in either of two locations by dragging and dropping the tool from the Tools tab. Like any other tool, you can place the Comparison tool anywhere in the current composition by dropping the tool in the Schematic, Player, etc.

The Display Tools is the desired location for the Comparison tool. The Display Tools contains two place holders; one for the Comparison tool and one for the Display Modifier. By default these place holders contain the Comparison and the Display Pass Through tools.

The Comparison tool can be placed either before or after the display modifier. This placement is controlled by the Display Tools Order setting in the User Preferences (Creative tab), which has a default value of Comparison before the display modifier.



Access to the Display Tools Comparison tool is through the Player options menu in the south gate. By dropping a Comparison tool on the appropriate tab in the Player options, it will replace the current Comparison tool. This is similar to creating a display modifier.

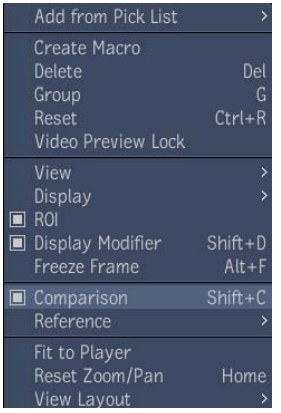
Enabling the Comparison tool

By default the Comparison tool will already exist in the Player (Display Tools) but will be muted.

To enable the Comparison tool:

➤ Do one of the following:

- Right-click in the Player and select Comparison.



- Press **Shift + C** (or **Shift + D** for the Display Modifier).
- Select the Mute button in the tool options of the Comparison Tool tab.

To delete the Comparison tool from the Display Tools:

➤ Select the Delete button in the tool options of the Comparison Tool tab. This causes the Comparison Pass Through tool to be used.

Selecting the Primary and Reference Inputs

The comparison tool's primary input is connected to the output that the viewer would be showing without the comparison tool. The reference input is set to current composition and has all of the display options available in the normal player. They are tool input/output, composition, and context points 1 - 4. These options can be changed by using the context menu Reference sub menu in the player. The display options in this menu refer to those of the reference input.

Hotkeys to switch the display options (1-8) will affect either the primary or reference input depending on which side of the manipulator the hotkey event takes place. The player information will indicate what sources are being shown for the primary and reference inputs—see Cycling Through Inputs and Outputs on page 71.

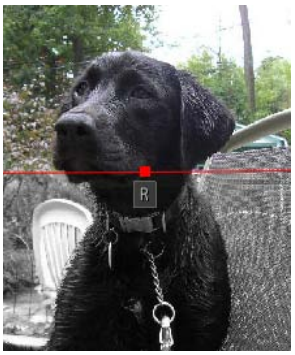
Comparison Tool Controls

The Comparison tool has the following parameters:

The comparison tool contains a list of manipulators; a split bar and a region, which can be selected to change the behavior of what areas are being compared.

The split bar is the default manipulator. Through direct manipulation, the bar can be moved, rotated, and can have its pivot point changed. By default the split bar separates the viewer horizontally in two with the reference input on the bottom side.

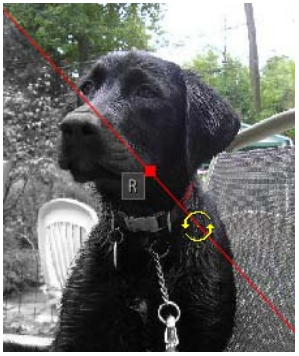
There is a pivot point (indicated by a small box) around which the split bar can be rotated, or from which the split bar can be moved. The letter R next to the pivot box indicates the side of the bar where the reference input is displayed.



Reference processed with Gray tool.

To rotate the split bar around its pivot point:

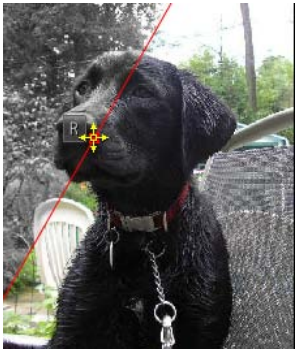
➤ Drag the split bar either above or below the pivot point.



Reference processed with Gray tool.

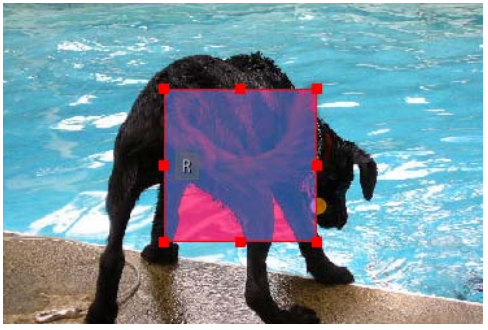
To move the split bar:

➤ Drag the pivot point. You can also **Ctrl** + click the viewer to place the pivot point under the cursor, and then drag over the viewer to move the split bar.



Reference processed with Gray tool.

The region rectangle is created and modified through direct manipulation. By default the region will be centered in the viewers. There are pivot points (indicated by small boxes) on each corner and side to change the width and height of the region. The letter R next to the left pivot point indicates whether the reference input is displayed in the interior or exterior of the region.

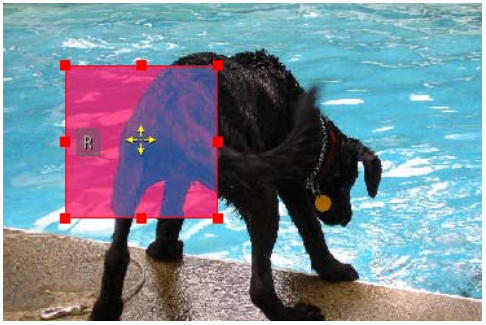


Reference processed with Color Space tool.

To move the region:

➤ Do one of the following:

- Drag the interior of the region.

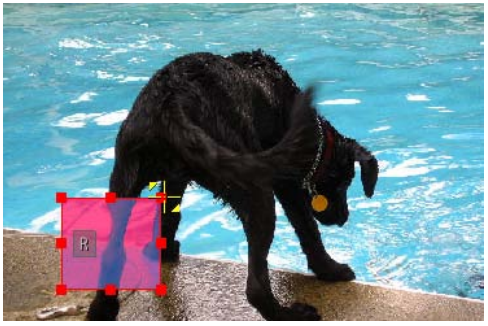


Reference processed with Color Space tool.

- Place the cursor at the location where you want the center of the region to be and press **Ctrl** + click.

To resize the region:

➤ Drag any of the pivot points.



Reference processed with Color Space tool.

To show or hide a particular manipulator:

➤ Press **Shift** + **H**. When the manipulator is hidden, it is still active.

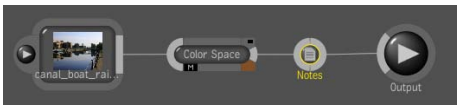
To change which area of the manipulator the reference input occupies:

➤ **Press Shift + R.**

Use:	To:
Offset	Set a time offset—see Linking Compositions on page 234. This parameter applies only to the reference image.
Repeat	Set the repeat mode —see Linking Compositions on page 234. This parameter applies only to the reference image.
Mark In/Mark Out	Set in and out points—see Linking Compositions on page 234. These parameters apply only to the reference image.
Opacity	Vary the opacity of the reference input to reveal the underlying primary image. The transparency will only affect the primary input side of the manipulator. The reference side will always remain 100% opaque. Use **Shift + O** to toggle the opacity.
Zoom/Translate X/Translate Y	Translate and scale the entire viewer using the pan and zoom hotkeys currently implemented. To translate and zoom the reference image, use the controls in the Comparison tool UI. To translate and zoom the primary image, swap the images and use the controls in the Comparison tool UI.

Notes Tool

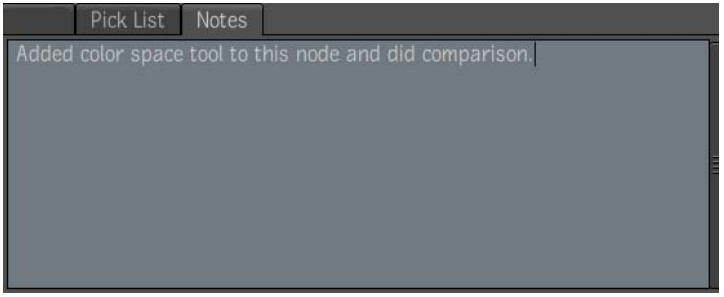
The Notes tool lets you add comments to your composition which can be useful when collaborating on an effect or project with other artists. You can use the Notes tool to add notes to any node in a dependency graph. And because the Notes tool is an image pass through tool, you can add it to an image connection. Once you've added comments, you can use standard cut, copy, and paste commands to edit the text.



Notes added to dependency graph.

To add notes to a composition:

1. From the Tools tab, drag a Notes tool from the Utilities folder to the dependency graph in the Schematic view.
2. In the Notes tool UI, type in your comments.



Pass Through Tool

The Pass Through tool is an image tool that passes its input to its output unmodified. No image copying, format change, or content change is performed. It can also be used as a placeholder in the Display Tools for most color conversion and image modifying tools, and affects only what you see in the Player.

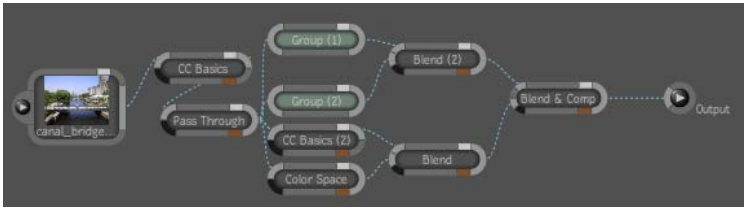
To add a display modifier to the Display Pass Through placeholder:

1. Drag a tool from the Tools tab and pause in the Player, then middle-click or press the tilde (~) key.
2. Go through the south gate to the Display Options and drop the tool onto the Display Pass Through tab.

To delete a display modifier from the Display Pass Through placeholder:

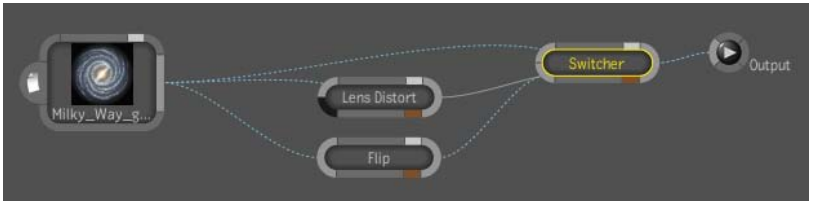
➤ With the Display (modifier) tab active, click the Delete button in the Tool Options area.

The Pass Through tool is very useful as a connection point in the dependency graph. The Pass Through tool node allows you to make multiple output sockets. Use this node as the input to a group to simplify the representation of a complex effect. Once all the necessary Pass Through output connections have been made, any other tool that you want to add before the Pass Through branching point can be done without having to cut and reestablish the connections.



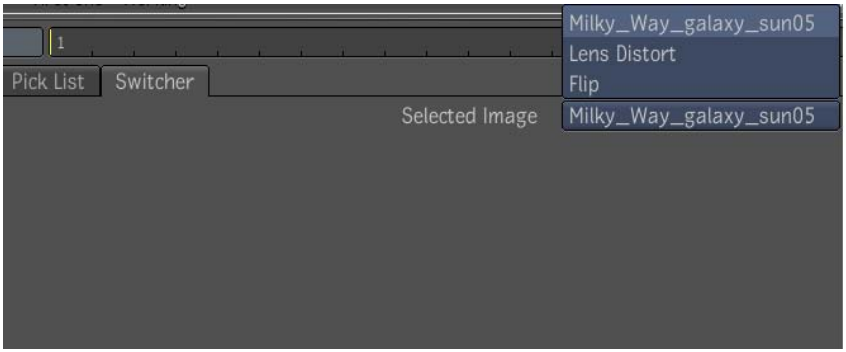
Switcher Tool

The Switcher is a standalone tool that allows you to alternate between different output views from multiple nodes without having to disconnect and reconnect nodes. It has multiple image inputs, but only one output. There is a single primary input, and a secondary input that can receive input from multiple nodes. The Switcher functions as a pass through for one selected image. Specifically, the output acts as a pass through for the primary input.



The Switcher tool has just one option in the UI. The list of node inputs is displayed so you can select a node to connect to the primary input. The name of the node connected to the primary input is always visible. The list of nodes connected to the secondary input can be viewed by clicking the name button in the Switcher tool UI. The user just has to click one of the nodes displayed in the list of nodes connected to the secondary input, and it will automatically become connected to the primary input. The change will immediately be

visible in the Schematic view. You can also use the S hotkey to switch between inputs.



To add a Switcher to a composition:

1. From the Tools & View tab, drag a Switcher tool from the Utilities folder to the Schematic view or directly into a dependency graph.
2. Connect one node to the primary input on the Switcher. Multiple nodes can be connected to the secondary input.

To switch inputs:

1. In the Switcher tool UI, click the name button to display the nodes connected to the secondary input.
2. Select the node from the list of secondary inputs that you want to connect to the primary input.

Hotkeys

A

Topics in this chapter:

Hotkeys

The following chapter is divided into two sections: the first section describes the hotkeys for Windows and Linux on page 818, the second describes the hotkeys forMac OS on page 837.

Windows and Linux

The following are the Composite hotkeys for Windows and Linux.

Login Screen

The following hotkeys work in the login screen.

Press:	To:
Enter	Log in.
Ctrl + Q	Exit Composite.

General

The following hotkeys work throughout the application.

Exit

The following hotkey is to exit the application.

Press:	To:
Ctrl + Q	Exit Composite.

Context Menus

The following hotkeys are available for context, or right-click, menus.

Press:	To:
context menu + click or right-click	Display the context menu. If a keyboard includes the **context menu** key, it is typically located three keys to the right of the **Spacebar**.

Select, Edit, Undo/Redo

The following hotkeys are available to perform basic editing operations.

Press:	To:
Ctrl + **X**	Cut the element.
Ctrl + **C**	Copy the element.
Ctrl + **V**	Paste the element.
Ctrl + click	Add to, or remove from, the selection.
Shift + click	Select the range of elements between this one and the last one selected.
Ctrl + **Z**	Undo the last operation.
Ctrl + **Y**	Redo the last undo.

User Interface Elements

The following hotkeys are available for different interface elements.

Menubar and Taskbar

The following hotkeys are available for the menubar and taskbar.

Press:	To:
Ctrl + **T**	Turn the taskbar on or off.

Press:	To:
Ctrl + M	Turn the menubar on or off.

Layout Presets

The following hotkey is available for the layout presets.

Press:	To:
Ctrl + Backspace	Previous layout preset.

Cursor

The following hotkey is available for the cursor.

Press:	To:
Ctrl + Alt + Shift + C	Reset the cursor

Tabs and Controls

The following hotkeys are available for tabs and controls.

Press:	To:
Shift + Tab	Go to previous control.
Tab	Go to next control
Alt + Left Arrow	Go to previous tab
Alt + Right Arrow	Go to next tab

Text Editor and Text Fields

The following hotkeys are available in the text editor and in any text field.

Press:	To:
Enter	Text editor: Add a line break. Text field: Accept text.

Press:	To:
Double-click	Select current word.
Backspace	Delete previous character.
Delete	Delete next character.
Home	Move to beginning of line.
End	Move to end of line.
Left Arrow	Move cursor left.
Right Arrow	Move cursor right.
Shift + **Home**	Select text from current position to beginning of line.
Shift + **End**	Select text from current position to end of line.
Shift + **Left Arrow**	Select one character to the left of the cursor.
Shift + **Right Arrow**	Select one character to the right of the cursor.
Shift + **Up Arrow**	Text editor only. Select from current position backward to same position on previous line.
Shift + **Down Arrow**	Text editor only. Select from current position forward to same position on next line.
Ctrl + **Left Arrow**	Move cursor left by one word.
Ctrl + **Right Arrow**	Move cursor right by one word.
Ctrl + **Shift** + **Left Arrow**	Select word to the left of the cursor.
Ctrl + **Shift** + **Right Arrow**	Select word to the right of the cursor.
Esc	Cancel text entry.
Alt + /	Auto Complete.

Numeric Fields (Value Editors)

The following hotkeys are available in numeric fields.

Press:	To:
Up Arrow	Increment (with autorepeat).
Down Arrow	Decrement (with autorepeat).
Ctrl + drag	Increment * 10.
Alt + drag	Increment / 10.
Ctrl + **Shift** + drag	Increment * 100.
Alt + **Shift** + drag	Increment / 100.
Enter	Edit text.
Ctrl + click	Reset to default.
Shift + click	Display the standard calculator.
Alt + **Shift** + click	Display the Espresso calculator if you can animate the field, or the standard calculator if you cannot.
context menu + click or right-click	Display the pop-up menu.

Pop-up Dialogs

The following hotkeys are available in pop-up dialogs.

Press:	To:
Enter	Accept
Esc	Cancel

Browsers

The following hotkeys are available in any browser. The Browser view and the File workspace are browsers.

Press:	To:
Delete	Delete
F5	Refresh
Ctrl + **A**	Select all.
Ctrl + drag	Copy
Ctrl + click	Add to, or remove from, the selection.
Shift + click	Select the range between the last element selected and this one.
Shift + click (on Sort area of the column)	Add the column to the sort criteria.
F2	Edit the text.
Esc	Cancel the text edit.
Down Arrow	Move to next row.
Up Arrow	Move to previous row.
Right Arrow	Move to next column.
Left Arrow	Move to previous column.
Page Down	Move to next page.
Page Up	Move to previous page.
Home	Move to first item in list.
End	Move to last item in list.
Double-click	Open (File Browser).
Ctrl + **D**	Bookmark path.

Floating Browsers

The following hotkeys are available for the Floating Browser.

Press:	To:
Ctrl + I	Activate/deactivate Floating File Browser.
Alt + O	Activate/deactivate floating view
Ctrl + O	Activate or deactivate the Floating File Browser.
Esc	Close the Floating Browser.

Composition Hotkeys

The following hotkeys are available when working with a composition.

Press:	To:
Ctrl + S	Save composition
Ctrl + P	Go to Pick List tab.
J	Toggle play backward.
L	Toggle play forward.
Shift + Left Arrow	Move to the first frame.
Shift + Right Arrow	Move to the last frame.
Right Arrow	Move to the next frame.
Left Arrow	Move to the previous frame.
I	Set mark in.
O	Set mark out.
U	Set cue mark.
Ctrl + Right Arrow	Next cue mark
Ctrl + Left Arrow	Previous cue mark.

Press:	To:
Shift + T	Toggle Timecode/Frame.
Alt + Shift + Right Arrow	Select downstream node.
Alt + Shift + Left Arrow	Select upstream node.
Alt + Shift + Down Arrow	Select next input/output.
Alt + Shift + Up Arrow	Select previous input/output.
Ctrl + N	Create a new composition.
Ctrl + Shift + Tab	Previous view.
Ctrl + Tab	Next view.
F5	Refresh. This includes a refresh of the current composition if it is out of date (its composition icon is red).
Ctrl + R	Reset node.
Ctrl + W	Close composition.
Shift + Tab	Go to previous tool.
Tab	Go to next tool.
Shift + Home	Go to home tool.
Alt + Right Arrow	Go to next tab.
Alt + Left Arrow	Go to previous tab.
Shift + A	Toggle Marking State of Current Selection.
Alt + Left Arrow	Selection History back.
Alt + Right Arrow	Selection History forward.

Gate UI

The following hotkey is available for the Gate UI.

Press:	To:
Middle-click	Display the Gate UI.
(backquote key)	Display the Gate UI. The **backquote** key ( ) shares the same key with the tilde key (~) and is located below the **Esc** key on the keyboard.
Esc	Close the Gate UI.
Alt + swipe (through a gate)	Continue displaying the UI element of the gate you swiped through.

Calculator

The following hotkeys are available for the Calculator.

Press:	To:
Alt + **C**	Activate or deactivate the Calculator.
Esc	Close the Calculator.

Color Picker

The following hotkeys are available in the Color Picker.

Press:	To:
Ctrl + drag in the color spectrum	Perform an average (rectangle).
P	Use the eyedropper to pick a color without displaying the Color Picker.
Ctrl + click color pot	Pick a color for a color pot without displaying the Color Picker.
Alt + **P**	Activate or deactivate the pop-up Color Picker.
Ctrl + **Enter**	Accept.

Press:	To:
Esc	Cancel.

Views

The following hotkeys are available in Schematic, Animation Editor, and Player views.

Press:	To:
Spacebar + drag	Pan
Home	Reset zoom and pan.
Up Arrow	Zoom in.
Down Arrow	Zoom out.
Ctrl + **Up Arrow**	Integer zoom in.
Ctrl + **Down Arrow**	Integer zoom out.
Ctrl + **Spacebar** + drag	Zoom
Shift + **Spacebar** + drag	Zoom region.
Ctrl + **Home**	Zoom selected items.
Ctrl + **Alt** + **Home**	Zoom all scene.
[F1 - F4]	Activate Viewpoint [1-4].
Ctrl + **[F1 - F4]**	Set Viewpoint [1-4].
Ctrl + **Shift** + **[F1 - F4]**	Delete Viewpoint [1-4].

Animation Editor

The following hotkeys are available in the Animation Editor.

Press:	To:
+ (plus)	Add key mode.
- (minus)	Delete key mode.
Esc	Cancel add/delete key mode.
Shift + **T**	Lock keyframe time.
Shift + **V**	Lock keyframe value.
Shift + **L**	Lock tangent length.
Shift + **O**	Lock tangent orientation.
Ctrl + **Shift** + **L**	Unify tangent length.
Ctrl + **Shift** + **O**	Unify tangent orientation.
Shift + **I**	Cycle interpolation mode keyframes.
Shift + **E**	Cycle extrapolation mode curves.
K	Set keyframe.
Shift + **K**	Set keyframe for marked attributes.
Delete	Delete keyframe.
Ctrl + **A**	Select all curves.
Ctrl + **Alt** + **A**	Select all keyframes on selected curves.
Ctrl + **Left Arrow**	Move to previous keyframe on selected curves.
Ctrl + **Right Arrow**	Move to next keyframe on selected curves.
Ctrl + **Shift** + **Left Arrow**	Add previous keyframe on selected curves, to selection.
Ctrl + **Shift** + **Right Arrow**	Add next keyframe on selected curves, to selection.

Press:	To:
Shift + plus sign (+)	Increase brush size in Delete Key mode.
Shift + minus sign (-)	Decrease brush size in Delete Key mode.

Player

The following hotkeys are available in the Player.

Press:	To:
Shift + **Z**	Set a region of interest (ROI). Click and drag to define an initial ROI, then use the cursor to adjust the size or location of the region.
F	Toggle full screen Player.
R	View the red channel of the target.
G	View the green channel of the target.
B	View the blue channel of the target.
A	View the alpha channel of the target.
C	View the channels of the target.
1	Display C1 Output (output of context point 1).
2	Display C2 Output (output of context point 2).
3	Display C3 Output (output of context point 3).
4	Display C4 Output (output of context point 4).
5	Display Tool Input (input of the currently selected tool).
6	Display Tool Output (output of the currently selected tool).
7	Display Composition (output of the composition's output node).
8	Display rendered version.
Shift + **8**	Display previous rendered version.

Press:	To:
9	Display rendered output.
Ctrl + **R**	Reset nodes.
Shift + **C**	Toggle Comparison tool.
Shift + **D**	Toggle Display Modifier tool.
0	Next stream (stereo)
Shift + **0**	Previous stream (stereo)

Separators

The following hotkey is available for manipulating UI separators.

Press:	To:
Ctrl + click	Reset to previous location.

Schematic

The following hotkeys are available in the Schematic view.

Press:	To:
[1-4] + click a node	Set a context point on that node. The number indicates the number of the context point. For example, pressing **1** + click sets context point 1. Pressing **3** + click sets context point 3.
[1-4] + click the background of Schematic	Clear the context point. The number indicates the number of the context point to clear. For example, pressing **2** + click clears context point 2, and pressing **4** + click clears context point 4.
Shift + drag a node into contact with another node	Connect the two nodes (Kiss). Release **Shift** and continue dragging to cancel the operation.
Alt + drag a node onto a connection	Insert the node between the two nodes joined by that connection. Release **Alt** and continue dragging to cancel the operation.

Press:	To:
Ctrl + drag through one or more connections	Disconnect the nodes joined by that connection or by those connections.
G	Group or ungroup the selected node(s).
Double-click	Edit the selected group.
Double-click (background)	Exit the group.
M	Mute or unmute nodes.
C	Clean up Schematic.
Delete	Delete nodes.
Ctrl + **R**	Reset nodes.
A + click (node)	Mark node for keyframe.
T	Toggle node thumbnails.
D (drag over node)	Display the details of a selected node without opening the node.
Shift + **D**	Show all node details.
E	Expand or collapse node.
Ctrl + **A**	Select all nodes and connections.
X + click	Select all upstream nodes. You can select multiple branches without clearing the selection.
L	layout selected.
Ctrl + **L**	Layout all.
Ctrl	Increase Auto-pan speed.
Alt	Decrease Auto-pan speed.
(outside of viewer)	Create boundary node.
Shift + **G**	Group nodes (visual).

Press:	To:
Shift + **O**	Open visual group.
Shift + **C**	Close visual group.
V + drag	Add to visual group.
Shift + **V** + drag	Remove from visual group.
C + drag	Copy node

Tools

The following describes hotkeys available in specific tools.

Garbage Mask (also applies to Remove Dust and Warp 2D tools)

The following hotkeys are available in the Garbage Mask tool.

Press:	To:
D	Create Freeform.
S	Create Rectangular.
W	Create Elliptical.
Esc	Finish shape.
E	Edit.
T	Transform.
Shift + **Left Arrow**	Nudge left.
Shift + **Right Arrow**	Nudge right.
Shift + **Up Arrow**	Nudge up.
Shift + **Down Arrow**	Nudge down.

Press:	To:
Ctrl + **Shift** + **Left Arrow**	Nudge left (large).
Ctrl + **Shift** + **Right Arrow**	Nudge right (large).
Ctrl + **Shift** + **Up Arrow**	Nudge up (large).
Ctrl + **Shift** + **Down Arrow**	Nudge down (large).
Enter	Close shape.
Ctrl + **A**	Select all control points.
Ctrl + **.**	Select next control point.
Ctrl + **,**	Select previous control point.
Alt + click (on curve)	Insert a control point.
Alt + click (on control point)	Delete a control point.
Ctrl + **Alt** + click + drag (on control point)	Move a control point in auto tangent mode.
Ctrl + click (on control point)	Add tangent handle. Repeat to remove tangent handle.
Alt + drag (the handle)	Adjust the length of both sides of a tangent handle.
Ctrl + click + drag (the handle)	Break a tangent handle.
Ctrl + **Alt** + drag (over location on a curve)	Move a curve between points.
Ctrl + click+ drag (on the curve)	Create an edge point.

Keyer

The following hotkeys are available in the Keyer tool.

Press:	To:
M	Sample matte.
Shift + **[1-9]**	Sample patch [1-9].
D	Sample degrain.
S	Spill suppress.
Shift + **B**	Adjust blend.

Master Keyer

The following hotkeys are available in the Master Keyer tool.

Press:	To:
M	Sample matte.
Shift + **[1-9]**	Sample patch [1-9].

Paint

The following hotkeys are available in the Paint tool.

Press:	To:
S + drag	Adjust brush size. Drag right to increase, drag left to decrease.
Ctrl + click	Clone brush.
Shift + click	Line brush.
X	Flip foreground/background colors.

Spill & Blend

The following hotkeys are available in the Spill & Blend tool.

Press:	To:
S	Spill suppress.
Shift + **B**	Adjust blend.

Reaction

The following hotkeys are available in the Reaction tool.

Press:	To:
Q	Orbit the camera.
Q + click	Orbit grasp.
W	Truck the camera.
W + click	Truck grasp.
E	Dolly the camera.
E + click	Dolly grasp.
M	Enter Move mode.
, (comma)	Enter Rotate mode.
. (period)	Enter Scale mode.

Cameras

The following hotkeys for the camera are available in the Player when using the Reaction tool.

Press:	To:
Shift + 1	Switch cameras. If you are using more than one camera, you will cycle through each camera.
Shift + 2	View from perspective camera.

Press:	To:
Shift + 3	View from front.
Shift + 4	View from Left.
Shift + 5	View from Top.
Shift + **Home**	Reset camera.

Resize

The following hotkeys are available in the Resize tool.

Press:	To:
Ctrl + drag	Symmetrical resize.
Shift + drag	Maintain ratio.
Shift + **I** + drag	Maintain image input ratio.
Shift + **O** + drag	Maintain image output ratio.

Retimer

The following hotkeys are available in the Retimer tool.

Press:	To:
M	Cycle show vectors mode.

Tracker

The following hotkeys are available in the Tracker tool.

Press:	To:
Delete	Delete track point.
Backspace	Delete track point and step back.

Warping Tools

The following hotkeys are available in the Warping tools:

Press:	To:
M	Input transform manipulator.
'	Effect transform manipulator.
.	Output transform manipulator.
/	Crop.
Insert	Set Pivot (also for 2D Transform tool)
T	Twirl Rotation.

Comparison

The following hotkeys are available in the Comparison tool:

Press:	To:
Shift + **O**	Opacity.
Shift + **H**	Toggle tool visibility.
Shift + **R**	Swap reference.
Ctrl + click	Move pivot point or region center.
Player hotkeys **1** - **9** on reference side of manipulator	Cycle through inputs and outputs.
Player hotkeys **C**, **A** on reference side of manipulator	Cycle between channels and alpha.

Mac OS

The following are the Composite hotkeys for Mac OS.

Login Screen

The following hotkeys work in the login screen.

Press:	To:
Enter	Log in.
Cmd + **Q**	Exit Composite.

General

The following hotkeys work throughout the application.

Exit

The following hotkey is to exit the application.

Press:	To:
Cmd + **Q**	Exit Composite.

Context Menus

The following hotkeys are available for context, or right-click, menus.

Press:	To:
context menu + click or right-click	Display the context menu. If a keyboard includes the **context menu** key, it is typically located three keys to the right of the **Spacebar**.

Select, Edit, Undo/Redo

The following hotkeys are available to perform basic editing operations.

Press:	To:
Cmd + **X**	Cut the element.
Cmd + **C**	Copy the element.

Press:	To:
Cmd + **V**	Paste the element.
Ctrl + click	Add to, or remove from, the selection.
Shift + click	Select the range of elements between this one and the last one selected.
Cmd + **Z**	Undo the last operation.
Cmd + **Shift** + **Z**	Redo the last undo.

User Interface Elements

The following hotkeys are available for different interface elements.

Menubar and Taskbar

The following hotkeys are available for the menubar and taskbar.

Press:	To:
Cmd + **T**	Turn the taskbar on or off.
Ctrl + **M**	Turn the menubar on or off.

Layout Presets

The following hotkey is available for the layout presets.

Press:	To:
Cmd + **Backspace**	Previous layout preset.

Cursor

The following hotkey is available for the cursor.

Press:	To:
Ctrl + **Alt** + **Shift** + **C**	Reset the cursor

Tabs and Controls

The following hotkeys are available for tabs and controls.

Press:	To:
Shift + **Tab**	Go to previous control.
Tab	Go to next control
Alt + **Left Arrow**	Go to previous tab
Alt + **Right Arrow**	Go to next tab

Text Editor and Text Fields

The following hotkeys are available in the text editor and in any text field.

Press:	To:
Cmd + **Return**	Text editor: Add a line break. Text field: Accept text.
Double-click	Select current word.
Backspace	Delete previous character.
Delete	Delete next character.
Cmd + **Home**	Move to beginning of line.
Cmd + **End**	Move to end of line.
Left Arrow	Move cursor left.
Right Arrow	Move cursor right.
Cmd + **Shift** + **Home**	Select text from current position to beginning of line.
Cmd + **Shift** + **End**	Select text from current position to end of line.
Shift + **Left Arrow**	Select one character to the left of the cursor.
Shift + **Right Arrow**	Select one character to the right of the cursor.
Shift + **Up Arrow**	Text editor only. Select from current position backward to same position on previous line.

Press:	To:
Shift + **Down Arrow**	Text editor only. Select from current position forward to same position on next line.
Alt + **Left Arrow**	Move cursor left by one word.
Alt + **Right Arrow**	Move cursor right by one word.
Ctrl + **Shift** + **Left Arrow**	Select word to the left of the cursor.
Ctrl + **Shift** + **Right Arrow**	Select word to the right of the cursor.
Esc	Cancel text entry.
Alt + /	Auto Complete.

Numeric Fields (Value Editors)

The following hotkeys are available in numeric fields.

Press:	To:
Up Arrow	Increment (with autorepeat).
Down Arrow	Decrement (with autorepeat).
Ctrl + drag	Increment * 10.
Alt + drag	Increment / 10.
Ctrl + **Shift** + drag	Increment * 100.
Alt + **Shift** + drag	Increment / 100.
Enter	Edit text.
Ctrl + click	Reset to default.
Shift + click	Display the standard calculator.
Alt + **Shift** + click	Display the Espresso calculator if you can animate the field, or the standard calculator if you cannot.

Press:	To:
context menu + click or right-click	Display the pop-up menu.

Pop-up Dialogs

The following hotkeys are available in pop-up dialogs.

Press:	To:
Enter	Accept
Esc	Cancel

Browsers

The following hotkeys are available in any browser. The Browser view and the File workspace are browsers.

Press:	To:
Delete	Delete
F5	Refresh
Cmd + **A**	Select all.
Ctrl + drag	Copy
Ctrl + click	Add to, or remove from, the selection.
Shift + click	Select the range between the last element selected and this one.
Shift + click (on Sort area of the column)	Add the column to the sort criteria.
F2	Edit the text.
Esc	Cancel the text edit.
Down Arrow	Move to next row.
Up Arrow	Move to previous row.

Press:	To:
Right Arrow	Move to next column.
Left Arrow	Move to previous column.
Page Down	Move to next page.
Page Up	Move to previous page.
Home	Move to first item in list.
End	Move to last item in list.
Double-click	Open (File Browser).
Cmd + D	Bookmark path.

Floating Browsers

The following hotkeys are available for the Floating Browser.

Press:	To:
Cmd + I	Activate/deactivate Floating File Browser.
Alt + O	Activate/deactivate floating view
Cmd + O	Activate or deactivate the Floating File Browser.
Esc	Close the Floating Browser.

Composition Hotkeys

The following hotkeys are available when working with a composition.

Press:	To:
Cmd + S	Save composition
Cmd + P	Go to Pick List tab.
J	Toggle play backward.
L	Toggle play forward.

Press:	To:
Shift + **Left Arrow**	Move to the first frame.
Shift + **Right Arrow**	Move to the last frame.
Right Arrow	Move to the next frame.
Left Arrow	Move to the previous frame.
I	Set mark in.
O	Set mark out.
U	Set cue mark.
Ctrl + **Right Arrow**	Next cue mark
Ctrl + **Left Arrow**	Previous cue mark.
Shift + **T**	Toggle Timecode/Frame.
Alt + **Shift** + **Right Arrow**	Select downstream node.
Alt + **Shift** + **Left Arrow**	Select upstream node.
Alt + **Shift** + **Down Arrow**	Select next input/output.
Alt + **Shift** + **Up Arrow**	Select previous input/output.
Cmd + **N**	Create a new composition.
Ctrl + **Shift** + **Tab**	Previous view.
Ctrl + **Tab**	Next view.
F5	Refresh. This includes a refresh of the current composition if it is out of date (its composition icon is red).
Ctrl + **R**	Reset node.
Cmd + **W**	Close composition.

Press:	To:
Shift + **Tab**	Go to previous tool.
Tab	Go to next tool.
Shift + **Home**	Go to home tool.
Alt + **Right Arrow**	Go to next tab.
Alt + **Left Arrow**	Go to previous tab.
Shift + **A**	Toggle Marking State of Current Selection.
Alt + **Left Arrow**	Selection History back.
Alt + **Right Arrow**	Selection History forward.

Gate UI

The following hotkey is available for the Gate UI.

Press:	To:
Middle-click	Display the Gate UI.
(backquote key)	Display the Gate UI. The **backquote** key ( ) shares the same key with the tilde key (~) and is located below the **Esc** key on the keyboard.
Esc	Close the Gate UI.
Alt + swipe (through a gate)	Continue displaying the UI element of the gate you swiped through.

Calculator

The following hotkeys are available for the Calculator.

Press:	To:
Alt + **C**	Activate or deactivate the Calculator.
Esc	Close the Calculator.

Color Picker

The following hotkeys are available in the Color Picker.

Press:	To:
Ctrl + drag in the color spectrum	Perform an average (rectangle).
P	Use the eyedropper to pick a color without displaying the Color Picker.
Ctrl + click color pot	Pick a color for a color pot without displaying the Color Picker.
Alt + **P**	Activate or deactivate the pop-up Color Picker.
Ctrl + **Enter**	Accept.
Esc	Cancel.

Views

The following hotkeys are available in Schematic, Animation Editor, and Player views.

Press:	To:
Spacebar + drag	Pan
Home	Reset zoom and pan.
Up Arrow	Zoom in.
Down Arrow	Zoom out.
Cmd + **Up Arrow**	Integer zoom in.
Cmd + **Down Arrow**	Integer zoom out.
Cmd + **Spacebar** + drag	Zoom
Shift + **Spacebar** + drag	Zoom region.
Cmd + **Home**	Zoom selected items.

Press:	To:
Cmd + **Alt** + **Home**	Zoom all scene.
[F1 - F4]	Activate Viewpoint [1-4].
Cmd + **[F1 - F4]**	Set Viewpoint [1-4].
Cmd + **Shift** + **[F1 - F4]**	Delete Viewpoint [1-4].

Animation Editor

The following hotkeys are available in the Animation Editor.

Press:	To:
+ (plus)	Add key mode.
- (minus)	Delete key mode.
Esc	Cancel add/delete key mode.
Shift + **T**	Lock keyframe time.
Shift + **V**	Lock keyframe value.
Shift + **L**	Lock tangent length.
Shift + **O**	Lock tangent orientation.
Cmd + **Shift** + **L**	Unify tangent length.
Cmd + **Shift** + **O**	Unify tangent orientation.
Shift + **I**	Cycle interpolation mode keyframes.
Shift + **E**	Cycle extrapolation mode curves.
K	Set keyframe.
Shift + **K**	Set keyframe for marked attributes.
Delete	Delete keyframe.
Cmd + **A**	Select all curves.

Press:	To:
Cmd + **Alt** + **A**	Select all keyframes on selected curves.
Cmd + **Left Arrow**	Move to previous keyframe on selected curves.
Cmd + **Right Arrow**	Move to next keyframe on selected curves.
Cmd + **Shift** + **Left Arrow**	Add previous keyframe on selected curves, to selection.
Cmd + **Shift** + **Right Arrow**	Add next keyframe on selected curves, to selection.
Shift + plus sign (+)	Increase brush size in Delete Key mode.
Shift + minus sign (-)	Decrease brush size in Delete Key mode.

Player

The following hotkeys are available in the Player.

Press:	To:
Shift + **Z**	Set a region of interest (ROI). Click and drag to define an initial ROI, then use the cursor to adjust the size or location of the region.
F	Toggle full screen Player.
R	View the red channel of the target.
G	View the green channel of the target.
B	View the blue channel of the target.
A	View the alpha channel of the target.
Cmd + **Shift** + **C**	View the channels of the target.
1	Display C1 Output (output of context point 1).
2	Display C2 Output (output of context point 2).
3	Display C3 Output (output of context point 3).

Press:	To:
4	Display C4 Output (output of context point 4).
5	Display Tool Input (input of the currently selected tool).
6	Display Tool Output (output of the currently selected tool).
7	Display Composition (output of the composition's output node).
8	Display rendered version.
Shift + **8**	Display previous rendered version.
9	Display rendered output.
Cmd + **Shift** + **R**	Reset nodes.
Shift + **C**	Toggle Comparison tool.
Shift + **D**	Toggle Display Modifier tool.
0	Next stream (stereo)
Shift + **0**	Previous stream (stereo)

Separators

The following hotkey is available for manipulating UI separators.

Press:	To:
Ctrl + click	Reset to previous location.

Schematic

The following hotkeys are available in the Schematic view.

Press:	To:
[1-4] + click a node	Set a context point on that node. The number indicates the number of the context point. For example, pressing **1** + click sets context point 1. Pressing **3** + click sets context point 3.

Press:	To:
[1-4] + click the background of Schematic	Clear the context point. The number indicates the number of the context point to clear. For example, pressing **2** + click clears context point 2, and pressing **4** + click clears context point 4.
Shift + drag a node into contact with another node	Connect the two nodes (Kiss). Release **Shift** and continue dragging to cancel the operation.
Alt + drag a node onto a connection	Insert the node between the two nodes joined by that connection. Release **Alt** and continue dragging to cancel the operation.
Cmd + drag through one or more connections	Disconnect the nodes joined by that connection or by those connections.
G	Group or ungroup the selected node(s).
Double-click	Edit the selected group.
Double-click (background)	Exit the group.
M	Mute or unmute nodes.
C	Clean up Schematic.
Delete	Delete nodes.
Ctrl + **R**	Reset nodes.
A + click (node)	Mark node for keyframe.
T	Toggle node thumbnails.
D (drag over node)	Display the details of a selected node without opening the node.
Shift + **D**	Show all node details.
E	Expand or collapse node.
Cmd + **A**	Select all nodes and connections.
X + click	Select all upstream nodes. You can select multiple branches without clearing the selection.

Press:	To:
L	layout selected.
Cmd + **L**	Layout all.
Ctrl	Increase Auto-pan speed.
Alt	Decrease Auto-pan speed.
(outside of viewer)	Create boundary node.
Shift + **G**	Group nodes (visual).
Shift + **O**	Open visual group.
Shift + **C**	Close visual group.
V + drag	Add to visual group.
Shift + **V** + drag	Remove from visual group.
C + drag	Copy node

Tools

The following describes hotkeys available in specific tools.

Garbage Mask (also applies to Remove Dust and Warp 2D tools)

The following hotkeys are available in the Garbage Mask tool.

Press:	To:
D	Create Freeform.
S	Create Rectangular.
W	Create Elliptical.
Esc	Finish shape.
E	Edit.

Press:	To:
T	Transform.
Shift + **Left Arrow**	Nudge left.
Shift + **Right Arrow**	Nudge right.
Shift + **Up Arrow**	Nudge up.
Shift + **Down Arrow**	Nudge down.
Ctrl + **Shift** + **Left Arrow**	Nudge left (large).
Ctrl + **Shift** + **Right Arrow**	Nudge right (large).
Ctrl + **Shift** + **Up Arrow**	Nudge up (large).
Ctrl + **Shift** + **Down Arrow**	Nudge down (large).
Enter	Close shape.
Cmd + **A**	Select all control points.
Cmd + **.**	Select next control point.
Cmd + **Shift** + **.**	Select previous control point.
Alt + click (on curve)	Insert a control point.
Alt + click (on control point)	Delete a control point.
Ctrl + **Alt** + click + drag (on control point)	Move a control point in auto tangent mode.
Ctrl + click (on control point)	Add tangent handle. Repeat to remove tangent handle.
Alt + drag (the handle)	Adjust the length of both sides of a tangent handle.

Press:	To:
Ctrl + click + drag (the handle)	Break a tangent handle.
Ctrl + **Alt** + drag (over location on a curve)	Move a curve between points.
Ctrl + click+ drag (on the curve)	Create an edge point.

Keyer

The following hotkeys are available in the Keyer tool.

Press:	To:
M	Sample matte.
Shift + **[1-9]**	Sample patch [1-9].
D	Sample degrain.
S	Spill suppress.
Shift + **B**	Adjust blend.

Master Keyer

The following hotkeys are available in the Master Keyer tool.

Press:	To:
M	Sample matte.
Shift + **[1-9]**	Sample patch [1-9].

Paint

The following hotkeys are available in the Paint tool.

Press:	To:
S + drag	Adjust brush size. Drag right to increase, drag left to decrease.
Ctrl + click	Clone brush.
Shift + click	Line brush.
X	Flip foreground/background colors.

Spill & Blend

The following hotkeys are available in the Spill & Blend tool.

Press:	To:
S	Spill suppress.
Shift + **B**	Adjust blend.

Reaction

The following hotkeys are available in the Reaction tool.

Press:	To:
Q	Orbit the camera.
Q + click	Orbit grasp.
W	Truck the camera.
W + click	Truck grasp.
E	Dolly the camera.
E + click	Dolly grasp.
M	Enter Move mode.
, (comma)	Enter Rotate mode.

Press:	To:
. (period)	Enter Scale mode.

Cameras

The following hotkeys for the camera are available in the Player when using the Reaction tool.

Press:	To:
Shift + 1	Switch cameras. If you are using more than one camera, you will cycle through each camera.
Shift + 2	View from perspective camera.
Shift + 3	View from front.
Shift + 4	View from Left.
Shift + 5	View from Top.
Shift + **Home**	Reset camera.

Resize

The following hotkeys are available in the Resize tool.

Press:	To:
Ctrl + drag	Symmetrical resize.
Shift + drag	Maintain ratio.
Shift + **I** + drag	Maintain image input ratio.
Shift + **O** + drag	Maintain image output ratio.

Retimer

The following hotkeys are available in the Retimer tool.

Press:	To:
M	Cycle show vectors mode.

Tracker

The following hotkeys are available in the Tracker tool.

Press:	To:
Delete	Delete track point.
Backspace	Delete track point and step back.

Warping Tools

The following hotkeys are available in the Warping tools:

Press:	To:
M	Input transform manipulator.
'	Effect transform manipulator.
.	Output transform manipulator.
/	Crop.
Insert	Set Pivot (also for 2D Transform tool)
T	Twirl Rotation.

Comparison

The following hotkeys are available in the Comparison tool:

Press:	To:
Shift + **O**	Opacity.

Press:	To:
Shift + **H**	Toggle tool visibility.
Shift + **R**	Swap reference.
Ctrl + click	Move pivot point or region center.
Player hotkeys **1** - **9** on reference side of manipulator	Cycle through inputs and outputs.
Player hotkeys **C**, **A** on reference side of manipulator	Cycle between channels and alpha.

Composite Executables

B

Topics in this chapter:

Working with the Composite Executables

The Composite executable files let you verify stored information, edit settings, and perform certain tasks. The Composite executable files reside in the Composite *program* directory or in the Composite*bin* directories for Linux and Mac.

NOTE As the behavior of Composite is the same on a Linux or Mac workstation as on a Windows workstation, the screen shots in this chapter were taken from Composite running on a Windows workstation.

You can access the executables via a shell.

NOTE There are no file extensions on Linux and MAC OS workstations. Therefore, the executables don't have the "*.exe*" extension as they do on Windows. However, the functionality of the executables as well the list of command line parameters are identical on all platforms.

You can get a list of command line parameters for any Composite executable by running the executables without any parameters. In general, this provides some form of usage/help message.

To access the Composite Command Prompt:

➤ In the Windows Start menu, choose Programs > Autodesk >Autodesk Composite 2011> Composite Command Prompt.

composite Executable

The *composite* executable is used to start the interactive application.

To start Composite from a command line:

➤ In a shell, enter **`composite`**.

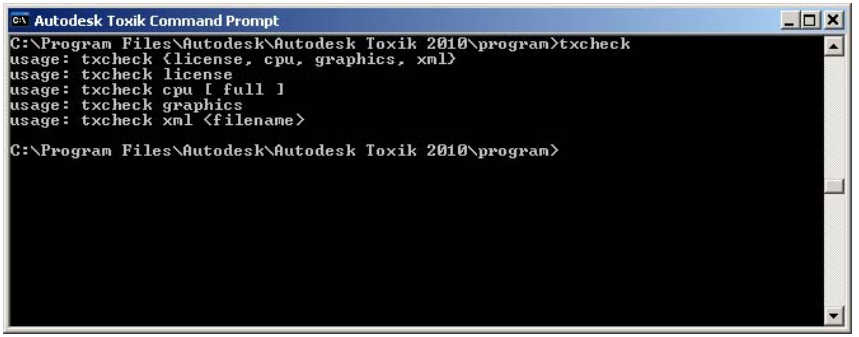
To set the Composite command line arguments,initialization options and environement variables—see Initialization Variables on page 873.

txcheck Executable

The *txcheck* executable checks the installed Composite license for each Composite feature, checks the CPU type, configuration, and supported processor features, checks the graphics card and driver, and parses an *.xml* file (for example, Composite UI resource and config files, Composite composition files, (.txcomposition), Composite project (.txproject), and user (.txuser) preference files) and checks whether it is valid or not.

To show the usage of the txcheck executable:

➤ In a shell, enter **`txcheck`**.

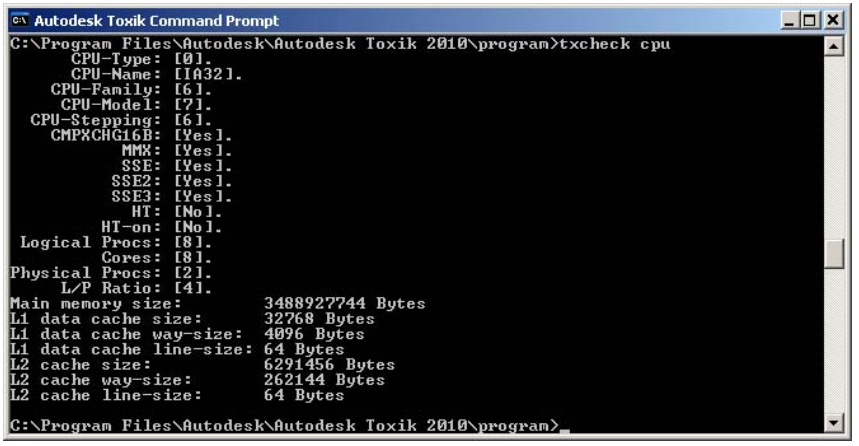


To check the Composite license installed on a workstation:

➤ In a shell, enter **`txcheck license`**.

To check the cpu:

➤ In a shell, enter **`txcheck cpu`**.

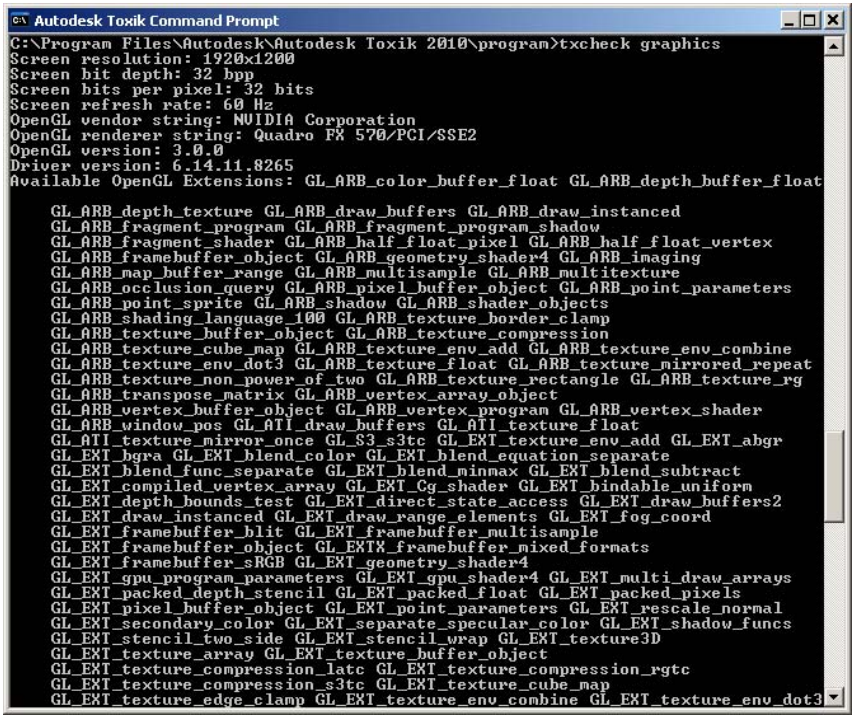


Summarized information regarding the CPU type, configuration, and supported processor features is listed. To get a full report on a cp, in a shell, enter the following: **`txcheck cpu full`**.

To check the graphics:

➤ In a shell, enter **`txcheck graphics`**.

A detailed check list is displayed. The list includes information on the graphics card, driver version, screen resolution, bit depth, refresh rate, OpenGL® version, vendor string, and available OpenGL extensions.



To check the *xml:*

➤ Enter a single parameter to parse the given Composite *xml* file and check it's validity. For example, from the Composite installation folder:*txcheck xml/resources/toolUi/crop.xml*.

txperf Executable

The *txperf* executable is used to test the performance of various Composite subsystems. *txperf* is built by the Composite application layer as a separate executable which depends on several Composite libraries. It runs a number

of performance tests either individually or simultaneously and covers the following subsystems:

- Disk I/O
- File System
- Content Server I/O
- Memory Transfers

Each test subsystem can be configured with various parameters. Any number of subsystems can be tested simultaneously but each subsystem will only have one configuration during the execution of the performance tool. The profiling tool will not run simultaneously with Composite or any Composite process including itself.

Disk IO

The disk IO tests are invoked by the *-disk* switch. These tests consist of reading and writing bytes through single threaded calls.

The behavior of disk IO tests are controlled by the following suboptions:

- size
- read block
- write block
- file
- iterations

size=**<n>** where **<n>** is the number of bytes the operation read/write. This suboption measures the speed at which bytes can be read/written depending of the size of the file. This measures the effect of file fragmentation depending on file size.

rblock=**<b>** where **<b>** is the size of each read. This suboption indicates that the file created will be read in chunk of **<l>** bytes.

rblock=**<b>** where **<b>** is the size of each write. This suboption indicates that the file created will be read in chunk of **<l>** bytes.

file=**<filepath>** where **<filepath>** indicates where the file will be created.

iteration=**<i>** where **<i>** indicates the number of times the test is repeated.

File System

The file system tests are invoked by the *-fs* switch. These tests consist of opening files, closing files, listing directories, verifying the existence of files, creating files, and deleting files.

The behavior of file-system tests is controlled by the following sub-options:

- files
- subdir
- location

files=**<n>** where **<n>** is the number of files to create. This suboption measures the speed at which files can be created and the number of creations which can be done per second, then verifies the existence of each file to measure the performance of determining a file's existence. Once all files are verified, each file is opened, then all files are closed and timing data for closing files is collected. The last three operations—verifying the existence of files, opening files, and closing files—are be performed in various orders (sequential, reverse, random, butterfly). The default is 256 files.

subdir=**<l>** where **<l>** is the number of subdirectory levels. This suboption indicates that files created by the test will be spread across a directory tree of depth **<l>**. The default is to create a flat directory structure. Valid range is from 1 to 16.

loc=**<path>** where **<path>** indicates where files will be created. A directory will be created below **<path>**. That directory will contain all the files created during the execution of the tool. When the tool finishes its execution, the created subdirectory will be destroyed along with all its contents. The default is specified by the TMP environment variable.

Content Server

The content server tests is invoked with the *-content* switch. Content server will measure read and write operations using the Composite content server. Collected data on I/O will include average throughput, sustained throughput and average/min/max latency.

The behavior of disk I/O tests is controlled by the following sub-options:

- Mode
- Size
- Blocks
- Queue

mode=**<ro|wo|rw>** where **ro** indicates that only read performance is measured, **rw** indicates that both read and write performance is measured and **wo** that only writing performance is measured. The default mode is **ro**.

time=**<s>** where **<s>** is the number of seconds devoted to each transfer test. The default is 5. For each of read and write, performance measurements will be accumulated over **<s>** seconds.

size=**<s>** where **<s>** is the number of kilobytes of each file used for testing I/O performance. The default size is 4096.

blocks=**<s>** where **<s>** is the size of I/O blocks, in kilobytes used for testing I/O performance. I/O will be performed in blocks of this size. The default will be 512.

queue=**<q>** where **<q>** is the number of outstanding requests. The default will be 32.

Memory

The memory performance tests are controlled by the *-mem* switch. Using this switch will cause memory-to-memory transfers to be measured for performance. The goal of these tests is to allow the FSB to be used during disk and cache tests so that FSB contention can be detected.

The behavior of memory tests is controlled by the following sub-options:

- Time
- Blocks
- Memory

time=**<s>** where **<s>** is the number of seconds devoted to the memory tests. The default is 5. Performance numbers will be accumulated over **<s>** seconds.

blocks=**<s>** where **<s>** is the size of blocks copied from memory-to-memory. The default is 512K.

freq=**<f>** where **<f>** is the frequency in hertz of memory transfers. The default is 96Hz. If 0 is specified than memory transfers will be performed continuously.

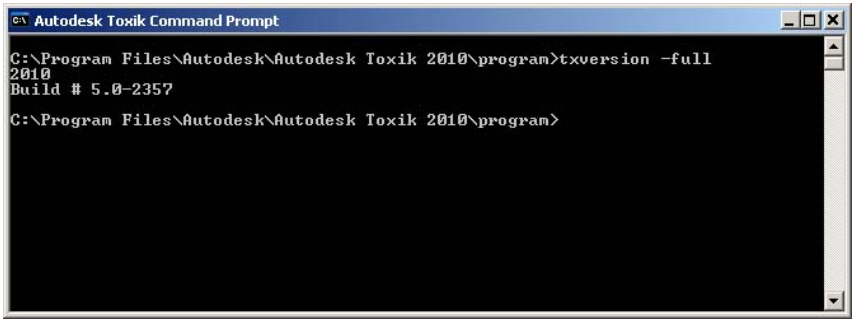
txversion Executable

The *txversion* executable reports the version of Composite. *txversion* supports one command line parameters called **-full**, which returns the extended version information.

To verify the version of Composite:

➤ In a shell, do one of the following:

- Enter **txversion**
- Enter **txversion -full**



Python Scripting

C

Topics in this chapter:

About Python Scripting

Scripting in Composite is implemented by extending the Python language (*http://www.python.org*). Composite is written in the C++ programming language, so some libraries, known as Python extension modules, allow for a script to invoke the Composite code through Python functions. The result is the Composite Python API.

NOTE The Composite Python API documentation can be accessed through the Help, via the Scripting menu (the HTML files are located in the documentation/scripting directory of the Composite installation).

With Python and the Composite Python API installed, you can write scripts to automatically perform certain repetitive managerial tasks and creative work on a project, such as creating compositions, adding nodes to it, importing material, versioning, rendering, and so on.

Composite ships with a number of sample scripts for performing common tasks. These sample scripts are located in the

resources/scripts/examplessubfolder of the Composite installation directory.

NOTE Example Python scripts are provided with no guarantee of effectiveness or efficiency.

There are also scripts that Composite itself invokes (archiving, rendering, pre-comp import, etc.) in the **resource/scripts**sufolder of the Composite installation directory. You may find these scripts helpful both as examples and as a starting point for your own scripts. In some cases, the script command that Composite launches to accomplish a given task is specified in the project preferences. For example, the "Render Actions" tab in the project preferences contains the "Render Executable" setting that specifies the Python command line executed each time you render a composition in Composite. By default, that command line launches the "render.py" script. You can edit the command line to launch a different script or change the "render.py script (located in **resources/scripts**) to modify the default rendering behavior, like performing post-render tasks, for example.

How to Run a Composite Python Script

To successfully run a Python script that makes use of the Composite Python API (or more specifically, a script that imports Composite Python modules), it is important to run it in the right environment, in order for the Python interpreter to find the Composite Python modules and libraries, and for the Composite code to execute properly afterwards.

There are two contexts in which a script can be run. First, in the Composite embedded Python interpreter, i.e. within the Composite interactive application. And secondly, in an external Python interpreter, i.e. within a separate Python process, typically from a shell.

Setting Up the Environment

Setting up the environment involves either an embedded or external interpreter.

Embedded

In the case of the embedded interpreter, the environment is already set up by Composite for a Python script to run properly.

Externally

An external interpreter can either be set up manually or automatically.

Manual Setup

In the case of a script ran in an external Python interpreter, the shell from which the Composite Python script is invoked must have the following environment variables set (except on Mac OS which doesn't require any). In the following paths,**<INSTALL_DIR>**refers to the path of the Composite installation directory:

On all operating systems, except Mac OS:

➤ **DL_INSTALL_DIR=<INSTALL_DIR>**

On Windows

1. PYTHONHOME = <INSTALL_DIR>/python
2. PATH = <INSTALL_DIR>/program;<INSTALL_DIR/plugins

On Linux

1. PYTHONHOME = <INSTALL_DIR>
2. LD_LIBRARY_PATH = <INSTALL_DIR>lib<INSTALL_DIR>/plugins<INSTALL_DIR>/lib/python2.5/site-packages/autodesk_toxik/libs

Automatic Setup

The procedure to automatically set up the environment required to run a Composite Python script from a shell can be found here: Working with the Composite Executables on page 859.

Running the Script

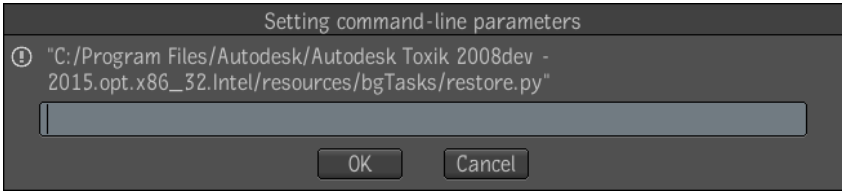
Running the script can be done with an embedded Python interpreter or externally.

Embedded

To run a python script in the Composite embedded Python interpreter, do the following:

1. Locate the script in a Composite "File Browser" ("Ctrl-I" (Windows and Linux) or “Cmd+ I” (Mac OS) to bring up a floating browser for example). Note that in order for Composite to recognize a file as a Python script, it must have the “.py” extension. Composite identifies Python scripts with a thumbnail that represents the Python logo:

2. Right-click on the script.
3. To run the script without any parameter choose “Run”.
4. To run the script with parameters choose "Run with Parameters" and type in the script arguments as you would in a shell.



Externally

WARNING To run a Composite Python script, it is strongly recommended to use the Python distribution shipped with Composite (deployed under the Composite installation directory). Running a Composite Python script with any other Python distribution might not work properly and is very likely to result in a crash.

On Windows:

➤ <INSTALL_DIR>/program/python.exe myScript.py

On Linux

➤ <INSTALL_DIR>/bin/python myScript.py

On Mac OS

➤ <INSTALL_DIR>/Contents/Frameworks/Python.framework/Versions/Current/bin/python myScript.py

Scripted Actions

Custom actions can be implemented as Python scripts that can be invoked from the UI in the same way as native Composite actions.

Action scripts can be located anywhere on the local file system or on the network. Two environment variables (and toxik.ini options) control the loading and searching of action scripts:

DL_SCRIPTED_ACTIONS_LOAD_ON_STARTUP (toxik::scripting::scriptedActionsLoadOnStartup)

1. Set to 1 or 0 to control whether to load and register the scripted actions during the Composite startup.
2. The default behavior is to register action scripts on startup.
3. Action scripts can be loaded (or reloaded) at any time through the (Re)Load Scripts entry of the Scripts top menu.

DL_SCRIPTED_ACTIONS_PATH (toxik::scripting::scriptedActionsPath)

1. Search path for action scripts; a list of directory paths separated by the character conventionally used by the operating system to separate search path components (":" on Linux and Mac OS, ";" on Windows).
2. By default, the search path is empty, but the following directory of the Composite installation is always searched for action scripts: **`resources/scripts/actions`**.

For more information on how to write scripted actions and examples, please refer to the main page of the scripting API documentation.

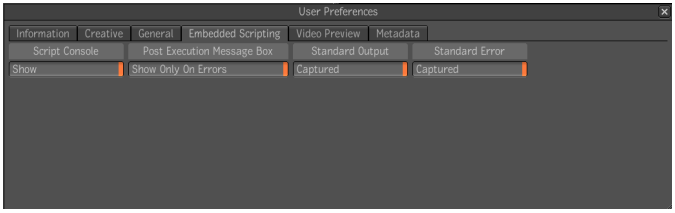
Viewing Output and Error Messages

Some scripts may output some information or error messages during execution. All output explicitly printed by the script writer goes to the script console, which is a pop-up window with one line of text. New text overrides any

previous text. When the scripts complete, the window closes. If there were errors during the script execution, another window is displayed, containing information or the stack trace that was generated when the error occurred. Information and error messages are displayed by default, but you can turn them off in the user preferences.

To turn off feedback while running scripts:

1. In the taskbar, click your user name to open the User Preferences window.
2. Select the Embedded Scripting tab.



3. Deselect any of the following:
   - **Script Console (Show)** Controls whether the script console is displayed when the script executes.
   - **Post Execution Message Box (Show Only on Errors)** In addition to errors, displays all outputs that were produced.
   - **Standard Output (Captured)** Normal messages are captured. If capture is disabled, messages go to the shell on Linux. On Windows, messages are lost.
   - **Standard Error (Captured)** Error messages are captured. If capture is disabled, messages go to the shell on Linux. On Windows, messages are lost.

Initialization Variables and String Substitutions

D

Topics in this chapter:

Initialization Variables

During installation, the toxik.ini file is created. These files can be found in *C:\Program Files\Autodesk\Autodesk Composite 2011\etc*.You can disable any of these files by adding the pound sign (#) before the toxik.ini file name.

The initialization variables are listed in a table as they appear in the Composite initialization file. The table is available from the Help menu via the The Menu Bar on page 19. If you specify the command line argument, the argument takes priority over the environment variable and the initialization variable. If you specify the environment variable, then it takes priority over the initialization variable.

To edit the Command Line switches via the Window's shortcut:

1. In the Windows Start menu, choose Programs > Autodesk > Composite 2011, then right-click the Composite menu entry and choose Properties.

2 In the Properties dialog that appears, click in the Target field and add the desired startup option to the end of the line.

   The option must be placed after the quotation mark.

3 Click OK and close the dialog.

   The changes will take effect the next time you start Composite.

To view the environment variables settings on Windows:

1 Right-click the My Computer icon on the desktop and choose Properties.

   If Composite is running or My Computer is not shown on the desktop, hold down the Windows key and press **E** to start Windows Explorer, *then* right-click My Computer and choose Properties.

   The System Properties dialog appears.

2 On the Advanced tab, click the Environment Variables button.

   The Environment Variables dialog appears, displaying system variables in the lower part of the dialog and variables defined for the current user in the top part.

To set the environment variables on Linux:

➤ Issue the following command in a tcsh shell:

```
# setenv <variable name> <value>
```

OR

In a bash shell:

```
# export <variable name> <value>
```

To view the environment variable settings on Linux:

➤ Issue the following command in a shell:

```
# env
```

String Substitutions

You can perform string substitutions in various Composite components to change the default execution settings. A table which lists the strings that can be edited is available from the Help menu via the The Menu Bar on page 19.

NOTE Two of the tokens, <Time> and <Date>, will return different results, depending when they are evaluated.

Glossary

E

List of Terms

This glossary contains industry-standard terms and terms that apply to Autodesk Composite.

1D LUT A 1D Look-up Table (LUT) is generated from one measure of gamma (white, gray, and black) or a series of measures for each color channel. With a pair of 1D LUTs, the first converts logarithmic data to linear data, and the second converts the linear data back to logarithmic data to print to film.

1-point stabilizing A technique for removing the horizontal or vertical jitter motion from a camera.

1-point tracking Tracking or stabilizing a single feature in motion, then applying that motion to a second element, so that the two appear to be locked together.

2-point stabilizing A technique for removing the motion or changes in scale/rotation (zoom/roll).

2-point tracking A technique for tracking two features, using the relationship between the two to track scale/rotation.

3D LUT A type of LUT for converting from one color space to another. It applies a transformation to each value of a color cube in RGB space. 3D LUTs use a more sophisticated method of mapping color values from different color spaces. A 3D LUT provides a way to represent arbitrary color space transformations, as opposed to the 1D LUT where a component of the output color is determined only from the corresponding component of the input color. In essence, the 3D LUT allows cross-talk, i.e. a component of the output color is computed from all components of the input color providing the 3D LUT tool with more power and flexibility than the 1D LUT tool. See also *1D LUT*.

4-point tracking Tracks four features, also referred to as corner pinning. With four-point tracking, four trackers are used to generate tracking data for anchoring the four corners of a bilinear surface to background footage.

alpha The name given to a fourth channel matte when bundled together with a three-channel color image. Also known as a *matte channel*. An alpha channel can be further described as either: 1. A TV signal that defines a portion of an image or clip to be keyed, or overlaid with a second image. A grayscale representation of an image used to create a mask for keying images.2. A matte or mask that defines a portion of an image or clip. The portion of a four-channel image that is used to store transparency information. 3. A raster channel defined by a paint or brush stroke.

animation The process of creating the illusion of moving images by displaying sequential images in rapid succession. In each successive image, two or more values are changed over time, and the items drawn or recorded in the images appear to move.

animation curve Also known as *function curve* or *fcurve*. It is a graphical representation of an animated parameter. The time and value of the animated parameter is displayed on two axes: the vertical axis representing the value, and the horizontal axis representing the time.

AutoKey A mode that, when activated, sets a keyframe whenever you change the selected object's properties.

background The composition background defines the rendering plane and sets its format after which other layers can be created.

bit depth 1. The number of bits used to represent the color of a pixel. Black and white images use a bit depth of 1; 16-color images use a bit depth of 4, 256-color images use 8, and so on. 2. The number of bits used to represent an audio sample. The more bits there are, the more accurately you can represent the amplitude at that point in time.

blue screen A special effects procedure in which the subject is photographed in front of a uniformly illuminated blue or green background. A new background image can be substituted for the blue or green during the shoot or in post production through the use of chroma key. See also keying.

Boolean A system of logic that lets you combine objects using logical operators, such as And, Or, Xor, Min, and Max. Boolean operators can be used for combining mattes in a single layer.

browsers A generic term which refers to the program or tool used for information searching in a window.

camera A device for viewing scenes from a different angle or "vantage point".

capture 1. To digitally transfer audio or video material from an external device, such as a videotape recorder, to a shared storage location or the local disk storage on your workstation. 2. To convert analog video and audio signals to digital signals.

channel 1. A physical audio input or output. 2. One of several color components that combine to define a color image. An RGB image is made up of red, green and blue color channels. In color correction, you can redefine color channels by blending color components in different proportions.

chrominance An image processing property that defines the hue and saturation of a pixel.

color A means of identifying colors in a source according to its component parts of RGB, hue, saturation, chrominance, luminance, sharpness, softness, tolerance or threshold, and computer-generated imagery.

color correction The process of adjusting the color characteristics of video material to achieve an accurate representation of color and consistency of color from one clip in a sequence to another. The term generally refers to adjustment changes made as part of a single effect.

color curves Color curves lets you remove color spill by either suppressing a sampled color, by modifying the red, green, blue, or hue, by modifying the saturation, or by modifying the luminance of the selected color, or by shifting the hue of a color range to a sampled hue shift target. Each curve in color curves is a hue gradient. When you change the shape of a curve, the colors along the curve's gradient change to reflect the result. For example, if your hue shift target is magenta, as you raise the green portion of the default hue shift curve, it gradually becomes magenta, becoming fully magenta when you reach the full value (75%).

comparison operators Operators that use Boolean algebra to compare two values with each other, if the comparison is true the result is 1, if the comparison is false the result is 0.

composition Composition is media imported into the application that can be used to insert, merge, or link to or from another composition.

computer-generated imagery (CG) Images created or generated with a computer.

conform A compound object created by projecting the vertices of one object, called the Wrapper, onto the surface of another object, called the Wrap-To.

context point A specific tool node you mark as the target for a Player view. Context points are especially useful when you want to see how changes to a tool node that precedes the node with a context point affect the context point.

corner pinning A technique used to pin the four corners of one static image (usually) onto another static image.

cue marks High-level keyframe controls that let you associate a keyframe with a mark. Instead of viewing a clip over and over again in order to evaluate and record important timing points, you can view footage and add cue marks.

dependency graph A set of connected nodes consisting of one or more inputs, an output, and any number of effects or compositing operations.

depth of field The range of acceptable focus in front of and behind the primary focus setting. It is a function not only of the specific lens used but also on the distance from the lens to the primary focal plane, and of the chosen aperture. Larger apertures will narrow the depth of field, smaller apertures will increase it.

Domain of Definition (DOD) A usually rectangular region that defines the maximum boundaries of useful information in an image. Generally, everything outside of the DOD will have a value of 0 in all channels of the image. The DOD is usually determined automatically, as opposed to a Region of Interest. See also Region of Interest.

expressions Mathematical formulas that let you control any parameter that can be animated, such as translation, rotation, scaling, material, or texture.

Expresso calculator An extension of the standard calculator that allows you to create expressions that generate values used in numerical fields.

external matte A matte taken from another clip. See also matte, garbage matte, static matte and traveling matte.

garbage mask Garbage masks can key undesired elements in an image and to do rotoscope work. A garbage mask affects the image's alpha channel, which is where the transparency information is stored. You can use a mask to create the alpha channel, or you can use a mask to add to an alpha channel.

garbage matte A rough simple matte that you create in an image's alpha channel that isolates unwanted elements from the primary element in an image. Garbage mattes are also referred to as garbage masks. See also matte, external matte, static matte, and traveling matte.

hue A specific color from the color spectrum, disregarding its saturation or value.

intermediate results The results rendered at any node in the dependency graph, other than the output node.

interpolation A mode that automatically defines the shape of an animation curve between keyframes in the curve. Infers a point which does not actually exist by averaging real points near it, either in space or time. See also extrapolation.

keyframe The value of an animated parameter that you set at a given point in time. Parameter values between keyframes are obtained by interpolating between keyframe values.

keyframing The simplest form of animating an object. It is based on the notion that an object has a beginning state, or condition, and changes over time in position, form, color, luminosity, or any other property to some different, final state. Keyframing takes the stance that we only need to show the key frames or conditions that describe the transformation of the object, and that all other intermediate positions can be figured out from these.

keying The process of isolating a region of an image by selecting pixels of a particular color and making those pixels transparent, or creating an alpha channel where those pixels are located.

layer An object that you can composite over another layer using a matte. A layer composed of a source input that you assign to a geometrical support, or 3D geometry, and a material node that is automatically generated.

layers Layers are composed of three separate components: a surface (3D geometry) object and local axis, a multi-channel material object, and a layer element.

linear color space An abbreviation Linear color space. A color space in which the relationship between a pixel's digital value and its visual brightness remains constant (linear) across the full gamut of black to white.

lights An object (algorithm) used to illuminate your scene and apply lighting effects to surfaces. Different types of lights include: omni, directional, and spot.

log color space An abbreviation for Logarithmic color space. A nonlinear color space whose conversion function is similar to the curve produced by the logarithmic equation.

luminance An image processing property that defines the brightness of a pixel. Expressed in percentages, 0% luminance is black and 100% luminance is white. Values between 0 and 100% define a range of grays. Colored pixels have a luminance value equivalent to the mean of their RGB values.

LUT A Look-up Table (LUT) is an ASCII file that describes specific pixel values used for image data conversion both on import and export. For example, you can use a LUT to convert a 10-bit logarithmic film image to a 16-bit linear image on import. You can use two kinds of LUTs: 1D LUTs and 3D LUTs. See 1D LUT, and 3D LUT.

masking The process of hiding a region of an image. You can use masks to remove the area outside the mask shape and keep the area inside the mask shape, or you can use masks to remove the area inside the mask shape and keep the area outside the mask shape. Pixel Masking lets you temporarily isolate specific areas of the footage. You can apply effects to the selected area of an image without affecting the rest.

material A material defines how a layer interacts with lights to define how a scene is rendered. Each layer must have a material associated with it, but many layers can share a single material.

matte An image used to define or control the transparency of another image. When you pull a key, a matte that defines the transparency of the front source is generated. There are several types of mattes. See also keying, external matte, garbage matte, static matte and traveling matte.

metadata Information embedded in the image attained through tags that further describes the image.

motion blur A digital effect that simulates the blur of fast-moving objects.

multiple-point tracking Multi-point tracking tracks as many features as you want. You can apply the stored motion to various elements, such as the center point of an object. For example, you could track a halo over the head of a person in a clip. Assume that in the clip a woman tilts her head and advances toward the camera. You would want the halo to reflect the position, scaling, and rotational changes of the tracked features on the woman's head.

mute To turn off a node.

node An element for connecting a composition in a dependency graph. There are several types of nodes, including input, output, group, tool, and super tool.

NTSC National Television Standards Committee. The committee that established the color transmission system used in the United States, Canada, Mexico and Japan. Also the broadcast standard for North America. The NTSC standard uses 525 horizontal lines per frame with two fields per frame of 262.5 lines. Each field refreshes at 59.94 Hz. See also PAL, and SECAM.

object Any element in a 3D scene, such as lights, camera, and surfaces.

PAL Phase Alternate Line. The color broadcasting standard used in most of Europe, China, Australia, New Zealand, the Middle East, and parts of Africa. The PAL standard uses a total of 625 lines per frame scanned at a rate of 25 fps, with two fields per frame of 312.5 lines each. Each field refreshes at 50 Hz. See also NTSC, and SECAM.

Pick list A list of frequently used tools and other elements that you can pick and assemble.

pinning With corner pinning (also known as four-point tracking), you use four trackers to generate tracking data for anchoring the four corners of a bilinear surface to background footage. See also 4-point tracking.

premultiply An image whose red, green, and blue channels have been multiplied by a matte that is usually stored as the alpha channel of this image.

preset A saved configuration of views and settings appropriate for a particular task.

process To create the final image or sequence of images using effects, transitions, composite, or container clips in your sequence.

proxy A low resolution copy of a high resolution image, one for each frame. Proxies are used to provide real-time playback of processed results and to provide low resolution stand in clips in a compositing environment so composites can be quickly created. See also composition thumbnails.

Region of Definition (ROD) In Composite each image has a size and position within a 2D reference plane also called the Image Reference Frame (IRF). The Region of Definition (ROD) of an image is the rectangle enclosing that image in the IRF. This rectangular region is always aligned with the main X & Y axes of the IRF.

Region of Interest (ROI) A region (usually rectangular) that is determined by the user in order to limit certain calculations. See also Domain of Definition (DOD).

render The process of creating a final image or sequence of images from a scene.

resolution The amount and degree of detail contained in the video image, measured along both the horizontal and vertical axes by dots or lines.

RGB A color model that defines pixels according to red, green, and blue channel values.

ROD See Region of Definition.

ROI See Region of Interest. See also Domain of Definition.

safe action area, safe title area An area in the center of the video image considered safe to place a title or other graphic, or capturing movement, so that none of it is cut off at the edges of the TV screen. The safe action area is 10% percent smaller than the maximum image size, and the safe title area is 20% smaller.

saturation 1. Saturation is the brilliance or purity of a given color. For example, the difference between a pastel and a pure color is defined by the amount of saturation. 2. The intensity of colors. 3. The degree by which the eye perceives a color as departing from a gray or white scale of the same brightness.

Schematic A view in which the artist builds a dependency graph for a composition. See also dependency graph.

SECAM Séquentiel couleur à Mémoire. A color television standard developed in France and used throughout Eastern Europe. The SECAM standard uses 625 lines per frame. See also NTSC, and PAL.

shader A program that defines the look of objects in a scene. Shaders are part of the rendering pipeline.

sharpness Sharpness can be defined as the visual sense of the abruptness of an edge. It highlights the detail in an image or image sequence.

SMPTE Society of Motion Picture and Television Engineers. A frame numbering system used for electronic editing and timing of video productions. Each frame of video is assigned a number. Timecode denotes the hours: minutes: seconds: frames (00:00:00:00) elapsed on videotape. There are two types of timecode: drop frame and non-drop frame.

softness Colors that fall within the softness range are partially transparent. These areas appear gray (a dark gray to light gray range) in the matte. For example, softness makes the transition between the foreground and background of a composition more convincing by softening what would otherwise look like an abrupt cutout. Softness can also be used to retain transparent key-in materials or shadows that you want to appear in the result composition.

stabilizing To remove motion jitter and unwanted camera movement from a clip.

super tool A tool comprised of a set of tools that includes Reaction, Garbage Mask, 2D Compositor, Image Import, Keyer, Tracker, and Paint.

surface A geometric object onto which a source is mapped to create a layer for a composition. Geometric types include: bilinear, frustum, box, plane, and sphere.

tangent A line that extends from a control point that controls the direction of a Bézier curve. Tangent handles also control key points on function curves when the curve point's type is set to Bézier.

task group A set of one or more preset UI configurations grouped according to task.

tolerance Tolerance or Threshold is the colors that fall within the tolerance or threshold range is transparent. These areas appear black in the matte.

tracking Tracking allows the capture and storage of motion that's applied to a second element, so that the two appear to be locked together. Tracking of one, two, four or multiple features is possible using 1-point, 2-point, 4-point or multiple-point tracking.

traveling matte A matte that changes position or shape over time. See also matte, external matte, garbage matte and static matte.

X axis The horizontal axis in a three-dimensional system.

Y axis The vertical axis in a three-dimensional system.

Z axis The axis that is perpendicular to the X and Y axes in a three-dimensional system.

Index

A

B

C

D

E

F

G

H

I

K

L

M

N

O

P

Q

R

S

T

U

V

W

X

Z