Fusion Page Effects

PART 9 — CONTENTS

3D Nodes

This chapter covers, in great detail, the nodes used for creating 3D composites.

The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Alembic Mesh 3D [Abc]

The Alembic Mesh 3D node

Alembic Mesh Node Introduction

At times, you may need to import 3D geometry from applications like Blender, Cinema4D, or Maya. One of the formats you can use for importing 3D geometry is the Alembic file format. This file type is a 3D scene interchange format that contains baked animation with its geometry. In other words, it eliminates the animation calculation times by embedding fixed, uneditable animation with 3D geometry. The animation is typically embedded using a point cache, which saves the dynamic data such as velocity after it has been calculated. Alembic objects can contain mesh geometry, cameras, points, UVs, normals, and baked animation.

You can import Alembic files (.abc) into Fusion in two ways:

— Choose File > Import > Alembic Scene in Fusion or Fusion > Import > Alembic Scene in DaVinci Resolve's Fusion page.

— Add an AlembicMesh3D node to the Node Editor.

The first method is the preferred method; both Alembic and FBX nodes by themselves import the entire model as one object. However, the Import menu breaks down the model, lights, camera, and animation into a string of individual nodes. This makes it easy to edit and modify and use subsections of the imported Alembic mesh. Also, transforms in the file are read into Fusion splines and into the Transform 3D nodes, which get saved with the comp. Later, when reloading the comp, the transforms are loaded from the comp and not the Alembic file. Fusion handles the meshes differently, always reloading them from the Alembic file.

Arbitrary user data varies depending on the software creating the Alembic file, and therefore this type of metadata is mostly ignored.

Alembic Import Dialog

An Alembic Import dialog is displayed once you select the file to import

The top half of the Import dialog displays information about the selected file including the name of the plugin/application that created the Alembic file, the version of the Alembic software developer kit used during the export, the duration of the animation in seconds, if available, and the frame rate(s) in the file.
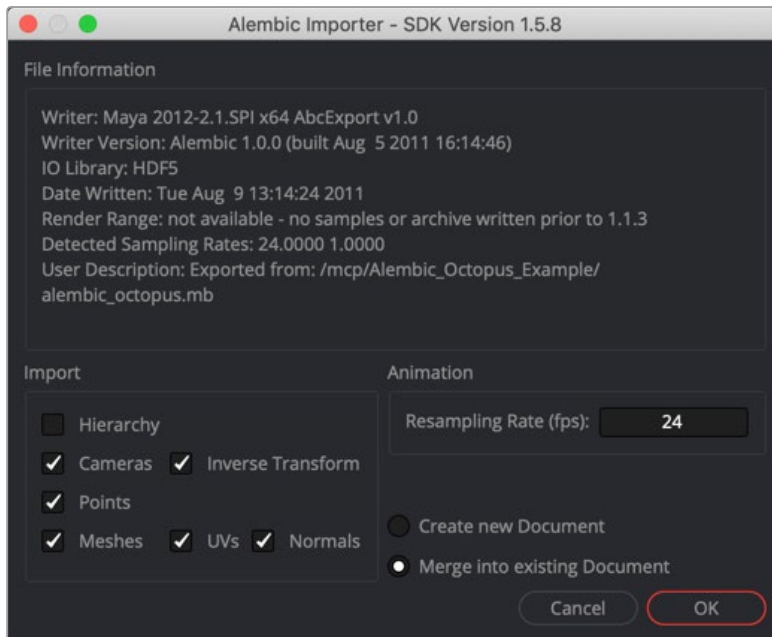
Alembic Import options

Various objects and attributes can be imported by selecting the checkboxes in the Import section.

— **Hierarchy:** When enabled, the full parenting hierarchy is recreated in Fusion using multiple Transform 3D nodes. When disabled, the transforms in the Alembic file are flattened down into the cameras and meshes. The flattening results in several meshes/cameras connected to a single Merge node in Fusion. It is best to have this disabled when the file includes animation. If enabled, the many rigs used to move objects in a scene will result in an equally large number of nodes in Fusion, so flattening will reduce the number of nodes in your node tree.

— **Orphaned Transforms:** When the hierarchy option is enabled, an Orphaned Transforms setting is displayed. Activating this Orphan Transforms setting imports transforms that parent a mesh or camera. For example, if you have a skeleton and associated mesh model, the model is imported as an Alembic mesh, and the skeleton as a node tree of Merge3Ds. If this is disabled, the Merge3Ds are not created.

— **Cameras:** When enabled, importing a file includes cameras along with Aperture, Angles of View, Plane of Focus, as well as Near and Far clipping plane settings. The resolution Gate Fit may be imported depending on whether the application used to export the file correctly tagged the resolution Gate Fit metadata. If your camera does not import successfully, check the setting for the Camera3D Resolution Gate Fit. Note that 3D Stereoscopic information is not imported.

— **InverseTransform:** Imports the Inverse Transform (World to Model) for cameras.

— **Points:** Alembic files support a Points type. This is a collection of 3D points with position information. Some 3D software exports particles as points. However, keep in mind that while position is included, the direction and orientation of the particles are lost.

— **Meshes:** This setting determines whether importing includes 3D models from the Alembic file. If it is enabled, options to include UVs and normals are displayed.

Animation

This section includes one option for the Resampling rate. When exporting an Alembic animation, it is saved to disk using frames per second (fps). When importing Alembic data into Fusion, the fps are detected and entered into the Resample Rate field unless you have changed it previously in the current comp. Ideally, you should maintain the exported frame rate as the resample rate, so your samples match up with the original. The Detected Sampling Rates information at the top of the dialog can give an idea of what to pick if you are unsure. However, using this field, you can change the frame rate to create effects like slow motion.

Not all objects and properties in a 3D scene have an agreed upon universal convention in the Alembic file format. That being the case, Lights, Materials, Curves, Multiple UVs, and Velocities are not currently supported when you import Alembic files.

Since the FBX file format does support materials and lights, we recommend the use of FBX for lights, cameras, and materials. Use Alembic for meshes only.

Inputs

The AlembicMesh3D node has two inputs in the Node Editor. Both are optional since the node is designed to use the imported mesh.

— **SceneInput:** The orange input can be used to connect an additional 3D scene or model. The imported Alembic objects combine with the other 3D geometry.
— **MaterialInput:** The optional green input is used to apply a material to the geometry by connecting a 2D bitmap image. It applies the connected image to the surface of the geometry in the scene.

Basic Node Setup

The AlembicMesh3D node is designed to be part of a larger 3D scene. Typically, when imported, a 3D geometry model is represented by one node, and any transforms are in another node. The nodes imported as part of the Alembic file connect into a Merge 3D node along with a camera, lights, and other elements that may be required for the scene.
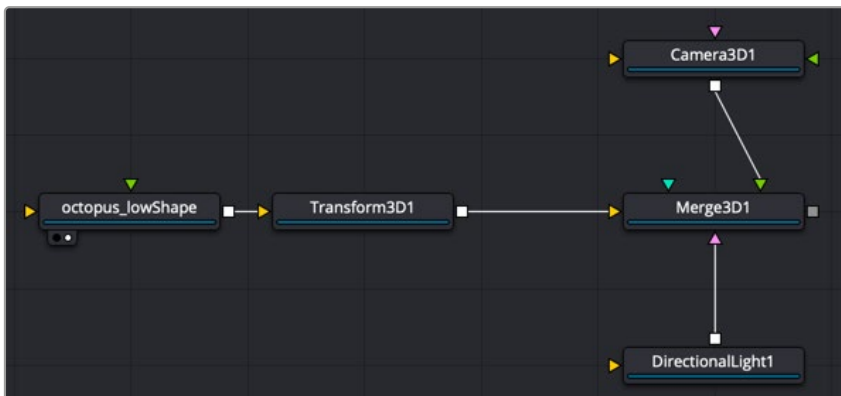
Alembic node structure

Inspector

Alembic mesh 3D controls

Controls Tab

The first tab in the Inspector is the Controls tab. It includes a series of unique controls specific to the Alembic Mesh 3D node as well as six groupings of controls that are common to most 3D nodes. "The Common Controls" section at the end of this chapter includes detailed descriptions of the common controls.

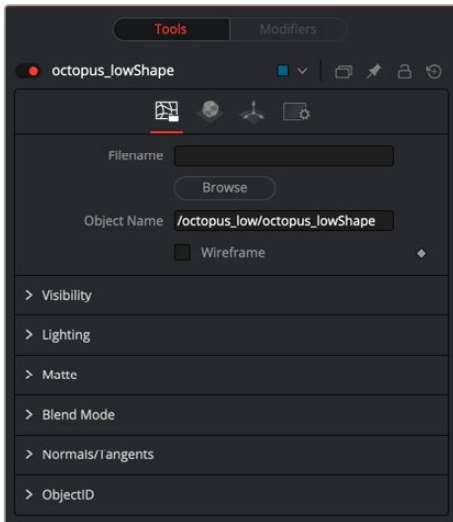Below are descriptions of the Alembic Mesh 3D specific controls.

Filename

The complete file path of the imported Alembic file is displayed here. This field allows you to change or update the file linked to this node.

Object Name

This text field shows the name of the imported Alembic mesh, which is also used to rename the Alembic Mesh 3D node in the Node Editor.

When importing with the Alembic Mesh 3D node, if this text field is blank, the entire contents of the Alembic geometry are imported as a single mesh. When importing geometry using File > Import > Alembic Scene, this field is set by Fusion.

Wireframe

Enabling this option causes the mesh to display only the wireframe for the object in the viewer. When enabled, there is a second option for wireframe anti-aliasing. You can also render these wireframes out to a file if the Renderer 3D node has the OpenGL render type selected.

Common Controls

Controls, Materials, Transform, and Settings Tabs

The controls for Visibility, Lighting, Matte, Blend Mode, Normals/Tangents, and Object ID in the Controls tab are common in many 3D nodes. The Materials tab, Transforms tab and Settings tab in the Inspector are also duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Bender 3D [3Bn]
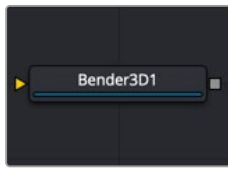
The Bender 3D node

Bender 3D Introduction

The Bender 3D node is used to bend, taper, twist, or shear 3D geometry based on the geometry's bounding box. It works by connecting any 3D scene or object to the orange input on the Bender 3D node, and then adjusting the controls in the Inspector. Only the geometry in the scene is modified. Any lights, cameras, or materials are passed through unaffected.

The Bender node does not produce new vertices in the geometry; it only alters existing vertices in the geometry. So, when applying the Bender 3D node to primitives, like the Shape 3D, or Text 3D nodes, increase the Subdivision setting in the primitive's node to get a higher-quality result.

Inputs

The following inputs appear on the Bender 3D node in the Node Editor.

— **SceneInput:** The orange scene input is the required input for the Bender 3D node. You use this input to connect another node that creates or contains a 3D scene or object.

Basic Node Setup

The Bender 3D node works by connecting a 3D node that contains geometry, like an image plane 3D, Shape 3D or Text 3D. The element you connect to the Bender 3D node will be distorted based on the controls in the Inspector. The Bender 3D node is designed to be part of a larger 3D scene, with the output typically connecting into a Merge 3D.
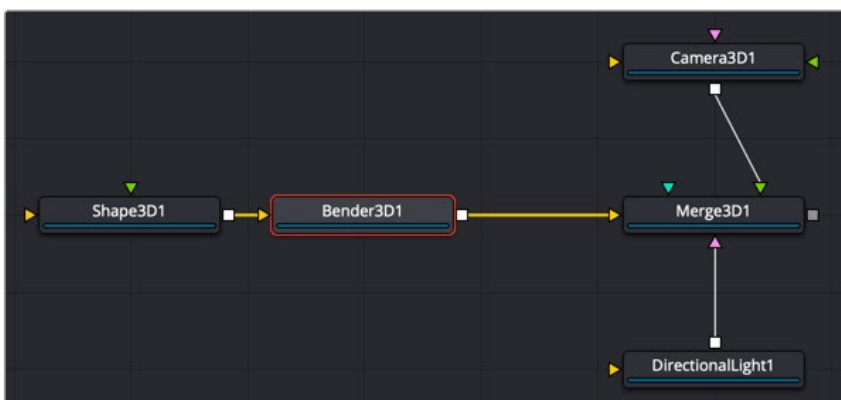
Bender 3D node structure

Inspector
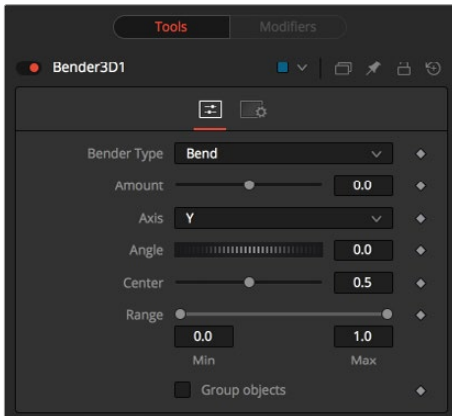
Bender 3D controls

Controls Tab

The first tab in the Inspector is the Controls tab. It includes all the controls for the Bender 3D node.

Bender Type

The Bender Type menu is used to select the type of deformation to apply to the geometry. There are four modes available: Bend, Taper, Twist, and Shear.

Amount

Adjusting the Amount slider changes the strength of the deformation.

Axis

The Axis control determines the axis along which the deformation is applied. It has a different meaning depending on the type of deformation. For example, when bending, this selects the elbow in conjunction with the Angle control. In other cases, the deform is applied around the specified axis.

Angle

The Angle thumbwheel control determines what direction about the axis a bend or shear is applied. It is not visible for taper or twist deformations.

Range

The Range control can be used to limit the effect of a deformation to a small portion of the geometry. The Range control is not available when the Bender Type is set to Shear.

Group Objects

If the input of the Bender 3D node contains multiple 3D objects, either through a Merge 3D or strung together, the Group Objects checkbox treats all the objects in the input scene as a single object, and the common center is used to deform the objects, instead of deforming each component object individually.

Common Controls

Settings

The Settings tab in the Inspector is common to all 3D nodes. This common tab is described in detail at the end of this chapter in "The Common Controls" section.

Camera 3D [3Cm]
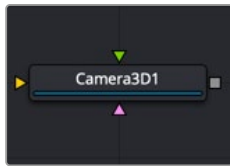
The Camera 3D node

Camera 3D Node Introduction

The Camera 3D node generates a virtual camera for viewing the 3D environment. It closely emulates the settings used in real cameras to make matching live-action or 3D-rendered elements as seamless as possible. Adding any cameras to a 3D composite allows you to frame the elements in a composite how you want and animate the camera during a scene to create moving camera shots.

Camera Projection

The Camera 3D node can also be used to perform Camera Projection by projecting a 2D image through the camera into 3D space. Projecting a 2D image can be done as a simple Image Plane aligned with the camera, or as an actual projection, similar to the behavior of the Projector 3D node, with the added advantage of being aligned precisely with the camera. The Image Plane, Projection, and Materials tabs do not appear until you connect a 2D image to the magenta image input on the Camera 3D node in the Node Editor.

Stereoscopic

The Camera node has built-in stereoscopic features. They offer control over eye separation and convergence distance. The camera for the right eye can be replaced using a separate camera node connected to the green left/right stereo camera input. Additionally, the plane of focus control for depth of field rendering is also available here.

If you add a camera by dragging the camera icon from the toolbar onto the 3D view, it automatically connects to the Merge 3D you are viewing. Also, the current viewer is set to look through the new camera.

Alternatively, it is possible to copy the current viewer to a camera (or spotlight or any other object) by selecting the Copy PoV To option in the viewer's contextual menu, under the Camera submenu.

Inputs

There are three optional inputs on the Camera 3D node in the Node Editor.

— **SceneInput:** The orange input is used to connect a 3D scene or object. When connected, the geometry links to the camera's field of view. It acts similarly to an image attached to the Image Plane input. If the camera's Projection tab has projection enabled, the image attached to the orange image input projects on to the geometry.

— **ImageInput:** The optional magenta input is used to connect a 2D image. When camera projection is enabled, the image can be used as a texture. Alternatively, when the camera's image plane controls are used, the parented planar geometry is linked to the camera's field of view.

— **RightStereoCamera:** The green input should be connected to another Camera 3D node when creating 3D stereoscopic effects. It is used to override the internal camera used for the right eye in stereoscopic renders and viewers.

Basic Node Setup

The output of a camera 3D node should be connected to a Merge 3D node. You then view the Merge 3D node and select the camera from the viewer's right-click menu or by right-clicking over the axis label in the viewer.
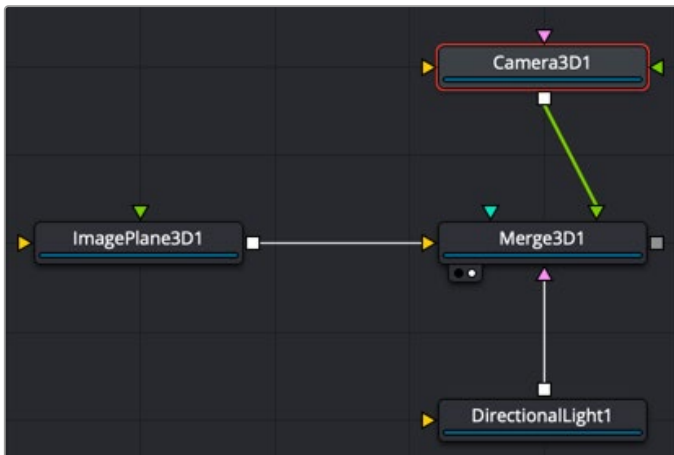
Camera node connected to and viewed through the Merge 3D

Displaying a camera node directly in the viewer shows only an empty scene; there is nothing for the camera to see. To view the scene through the camera, view the Merge 3D node where the camera is connected, or any node downstream of that Merge 3D. Then right-click on the viewer and select Camera > [Camera name] from the contextual menu. Right-clicking on the axis label found in the lower corner of each 3D viewer also displays the Camera submenu.

The aspect of the viewer may be different from the aspect of the camera, so the camera view may not match the actual boundaries of the image rendered by the Renderer 3D node. Guides can be enabled to represent the portion of the view that the camera sees and assist you in framing the shot. Right-click on the viewer and select an option from the Guides > Frame Aspect submenu. The default option uses the format enabled in the Composition > Frame Format preferences. To toggle the guides on or off, select Guides > Show Guides from the viewers' contextual menu, or use the Command-G (macOS) or Ctrl-G (Windows) keyboard shortcut when the viewer is active.
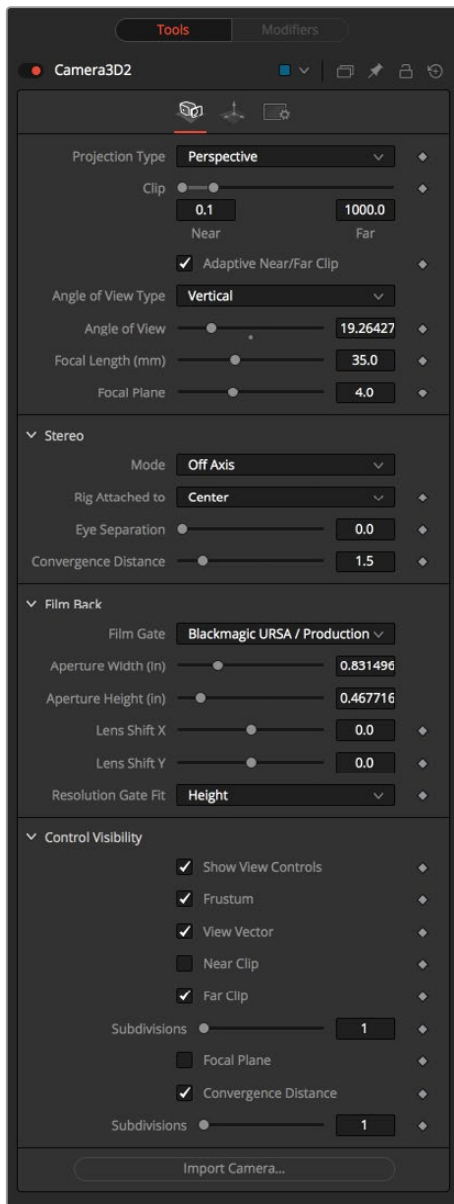
Inspector

Camera 3D controls

Controls Tab

The Camera3D Inspector includes six tabs along the top. The first tab, called the Controls tab, contains some of the most fundamental camera settings, including the camera's clipping plains, field of view, focal length, and stereoscopic properties. Some tabs are not displayed until a required connection is made to the Camera 3D node.

Projection Type

The Projection Type menu is used to select between Perspective and Orthographic cameras. Generally, real-world cameras are perspective cameras. An orthographic camera uses parallel orthographic projection, a technique where the view plane is perpendicular to the viewing direction. This produces a parallel camera output that is undistorted by perspective.

Orthographic cameras present controls only for the near and far clipping planes, and a control to set the viewing scale.

Near/Far Clip

The clipping planes are used to limit what geometry in a scene is rendered based on an object's distance from the camera's focal point. Clipping planes ensure objects that are extremely close to the camera, as well as objects that are too far away to be useful, are excluded from the final rendering.

The default perspective camera ignores this setting unless the Adaptive Near/Far Clip checkbox located under the Near/Far Clip control is disabled.

The clip values use units, so a far clipping plane of 20 means that any object more than 20 units from the camera is invisible to the camera. A near clipping plane of 0.1 means that any object closer than 0.1 units is also invisible.

> **NOTE:** A smaller range between the near and far clipping planes allows greater accuracy in all depth calculations. If a scene begins to render strange artifacts on distant objects, try increasing the distance for the Near Clip plane.

Adaptive Near/Far Clip

When selected, the renderer automatically adjusts the camera's near/far clipping plane to match the extents of the scene. This setting overrides the values of the Near and Far clip range controls described above. This option is not available for orthographic cameras.

Viewing Volume Size

When the Projection Type is set to Orthographic, the viewing volume size adjustment appears. It determines the size of the box that makes up the camera's field of view.

The Z-distance of an orthographic camera from the objects it sees does not affect the scale of those objects, only the viewing size does.

Angle of View Type

Use the Angle of View Type buttons to choose how the camera's angle of view is measured. Some applications use vertical measurements, some use horizontal, and others use diagonal measurements. Changing the Angle of View type causes the Angle of View control below to recalculate.

Angle of View

Angle of View defines the area of the scene that can be viewed through the camera. Generally, the human eye can see more of a scene than a camera, and various lenses record different degrees of the total image. A large value produces a wider angle of view, and a smaller value produces a narrower, or more tightly focused, angle of view.

Just as in a real-world camera, the angle of view and focal length controls are directly related. Smaller focal lengths produce a wider angle of view, so changing one control automatically changes the other to match.

Focal Length

In the real world, a lens' Focal Length is the distance from the center of the lens to the film plane. The shorter the focal length, the closer the focal plane is to the back of the lens. The focal length is measured in millimeters. The angle of view and focal length controls are directly related. Smaller focal lengths produce a wider angle of view, so changing one control automatically changes the other to match.

The relationship between focal length and angle of view is angle = 2 * arctan[aperture / 2 / focal_length].

Use the vertical aperture size to get the vertical angle of view and the horizontal aperture size to get the horizontal angle of view.

Plane of Focus (For Depth of Field)

Like a focal point on a real-world camera, this setting defines the distance from the camera to an object. It is used by the OpenGL renderer in the Renderer 3D node to calculate depth of field.

Stereo

The Stereo section includes options for setting up 3D stereoscopic cameras. 3D stereoscopic composites work by capturing two slightly different views, displayed separately to the left and right eyes. The mode menu determines if the current camera is a stereoscopic setup or a mono camera. When set to the default mono setting, the camera views the scene as a traditional 2D film camera. Three other options in the mode menu determine the method used for 3D stereoscopic cameras.

Toe In

In a toe-in setup, both cameras are rotating in on a single focal point. Though the result is stereoscopic, the vertical parallax introduced by this method can cause discomfort by the audience. Toe-in stereoscopic works for convergence around the center of the images but exhibits keystoning, or image separation, to the left and right edges. This setup is can be used when the focus point and the convergence point need to be the same. It is also used in cases where it is the only way to match a live-action camera rig.
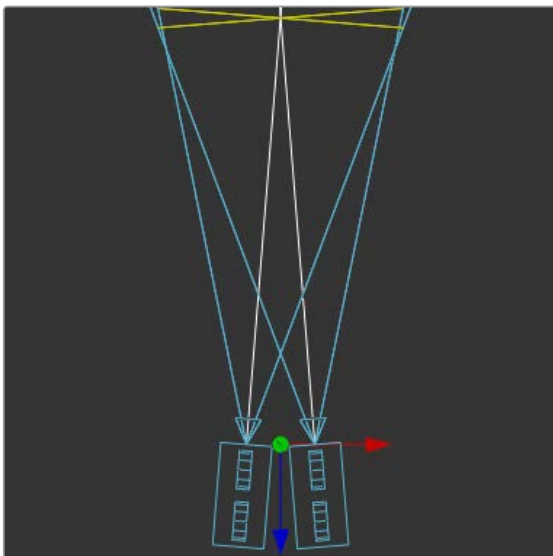
Toe In 3D camera setup

Off Axis

Regarded as the correct way to create stereo pairs, this is the default method in Fusion. Off Axis introduces no vertical parallax, thus creating stereo images with less eye strain. Sometimes called a skewed-frustum setup, this is akin to a lens shift in the real world. Instead of rotating the two cameras inward as in a toe-in setup, Off Axis shifts the lenses inward.
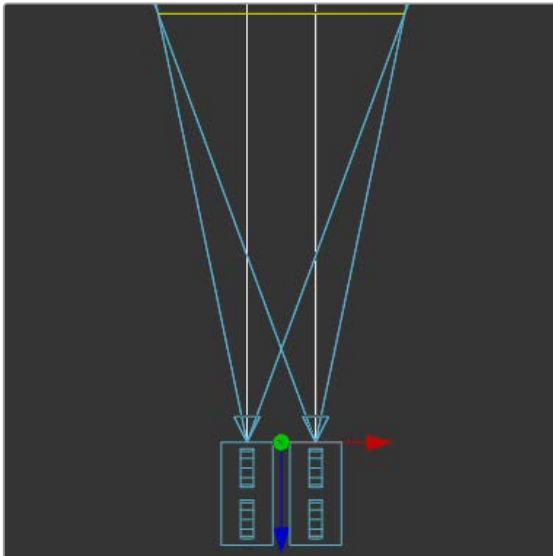
Off axis 3D camera setup

Parallel

The cameras are shifted parallel to each other. Since this is a purely parallel shift, there is no Convergence Distance control that limits your control over placing objects in front of or behind the screen. However, Parallel introduces no vertical parallax, thus creating less strain on the eyes.
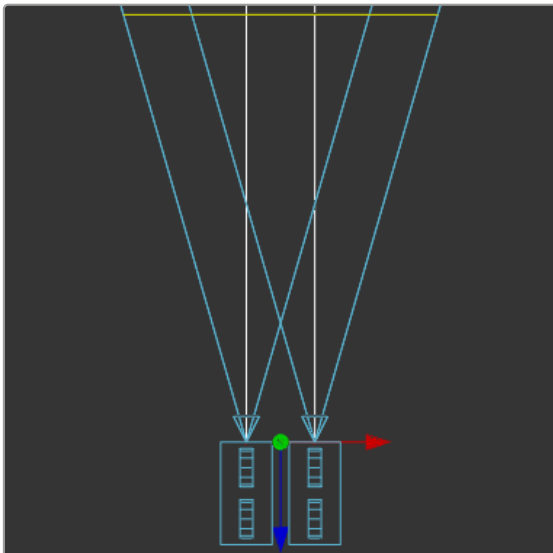
Parallel 3D camera setup

Rig Attached To

This drop-down menu allows you to control which camera is used to transform the stereoscopic setup. Based on this menu, transform controls appear in the viewer either on the right camera, left camera, or between the two cameras. The ability to switch the transform controls through rigging can assist in

matching the animation path to a camera crane or other live-action camera motion. The Center option places the transform controls between the two cameras and moves each evenly as the separation and convergence are adjusted. Left puts the transform controls on the left camera, and the right camera moves as the separation and convergence are adjusted. Right puts the transform controls on the right camera, and the left camera moves as adjustments are made to separation and convergence.

Eye Separation

Eye Separation defines the distance between both stereo cameras. Setting Eye Separation to a value larger than 0 shows controls for each camera in the viewer when this node is selected. Note that there is no Convergence Distance control in Parallel mode.

Convergence Distance

This control sets the stereoscopic convergence distance, defined as a point located along the Z-axis of the camera that determines where both left- and right-eye cameras converge. The Convergence Distance controls are only available when setting the Mode menu to Toe-In or Off Axis.

Film Back

Film Gate

The size of the film gate represents the dimensions of the aperture. Instead of setting the aperture's width and height, you can choose it using the list of preset camera types in the Film Gate menu. Selecting one of the options automatically sets the aperture width and aperture height to match.

Aperture Width/Height

The Aperture Width and Height sliders control the dimensions of the camera's aperture or the portion of the camera that lets light in on a real-world camera. In video and film cameras, the aperture is the mask opening that defines the area of each frame exposed. The Aperture control uses inches as its unit of measurement.

Resolution Gate Fit

Determines how the film gate is fitted within the resolution gate. This only has an effect when the aspect of the film gate is not the same aspect as the output image.

> **NOTE:** This setting corresponds to Maya's Resolution Gate. The modes Overscan, Horizontal, Vertical, and Fill correspond to Inside, Width, Height, and Outside.

— **Inside:** The image source defined by the film gate is scaled uniformly until one of its dimensions (X or Y) fits the inside dimensions of the resolution gate mask. Depending on the relative dimensions of image source and mask background, either the image source's width or height may be cropped to fit the dimension of the mask.
— **Width:** The image source defined by the film gate is scaled uniformly until its width (X) fits the width of the resolution gate mask. Depending on the relative dimensions of image source and mask, the image source's Y-dimension might not fit the mask's Y-dimension, resulting in either cropping of the image source in Y or the image source not covering the mask's height entirely.
— **Height:** The image source defined by the film gate is scaled uniformly until its height (Y) fits the height of the resolution gate mask. Depending on the relative dimensions of image source and mask, the image source's X-dimension might not fit the mask's X-dimension, resulting in either cropping of the image source in X or the image source not covering the mask's width entirely.

— **Outside:** The image source defined by the film gate is scaled uniformly until one of its dimensions (X or Y) fits the outside dimensions of the resolution gate mask. Depending on the relative dimensions of image source and mask, either the image source's width or height may be cropped or not fit the dimension of the mask.

— **Stretch:** The image source defined by the film gate is stretched in X and Y to accommodate the full dimensions of the generated resolution gate mask. This might lead to visible distortions of the image source.

Control Visibility

This section allows you to selectively activate the onscreen controls that are displayed along with the camera.

— **Show View Controls:** Displays or hides all camera onscreen controls in the viewers.

— **Frustum:** Displays the actual viewing cone of the camera.

— **View Vector:** Displays a white line inside the viewing cone, which can be used to determine the shift when in Parallel mode.

— **Near Clip:** The Near clipping plane. This plane can be subdivided for better visibility.

— **Far Clip:** The Far clipping plane. This plane can be subdivided for better visibility.

— **Focal Plane:** The plane based on the Plane of Focus slider explained in the Controls tab above. This plane can be subdivided for better visibility.

— **Convergence Distance:** The point of convergence when using Stereo mode. This plane can be subdivided for better visibility.

Import Camera

The Import Camera button displays a dialog to import a camera from another application.

It supports the following file types:

*LightWave Scene	`.lws`
*Max Scene	`.ase`
*Maya Ascii Scene	`.ma`
*dotXSI	`.xsi`

NOTE: FBX cameras can be imported using DaVinci Resolve's Fusion > Import > FBX Scene menu or File > Import > FBX Scene in Fusion Studio.

Image Tab

When a 2D image is connected to the magenta image input on the Camera3D node, an Image tab is created at the top of the inspector. The connected image is always oriented so it fills the camera's field of view.
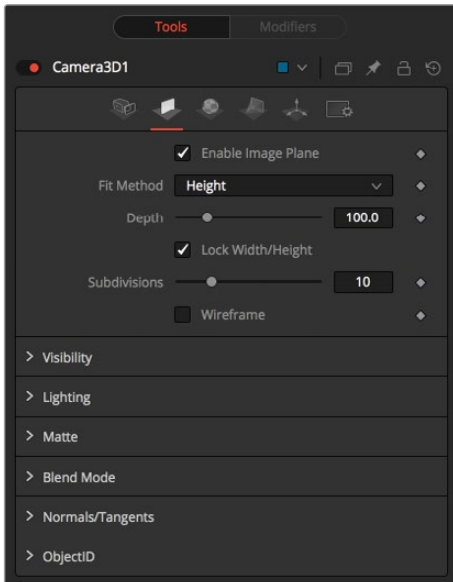
Camera 3D image plane tab

Except for the controls listed below, the options in this tab are identical to those commonly found in other 3D nodes. For more detail on visibility, lighting, matte, blend mode, normals/tangents, and Object ID, see "The Common Controls" section at the end of this chapter.

Enable Image Plane

Use this checkbox to enable or disable the usage of the Image Plane.

Fill Method

This menu configures how to scale the image plane if the camera has a different aspect ratio.

— **Inside:** The image plane is scaled uniformly until one of its dimensions (X or Y) fits the inside dimensions of the resolution gate mask. Depending on the relative dimensions of image source and mask background, either the image source's width or height may be cropped to fit the dimensions of the mask.

— **Width:** The image plane is scaled uniformly until its width (X) fits the width of the mask. Depending on the relative dimensions of image source and the resolution gate mask, the image source's Y-dimension might not fit the mask's Y-dimension, resulting in either cropping of the image source in Y or the image source not covering the mask's height entirely.

— **Height:** The image plane is scaled uniformly until its height (Y) fits the height of the mask. Depending on the relative dimensions of image source and the resolution gate mask, the image source's X-dimension might not fit the mask's X-dimension, resulting in either cropping of the image source in X or the image source not covering the mask's width entirely.

— **Outside:** The image plane is scaled uniformly until one of its dimensions (X or Y) fits the outside dimensions of the resolution gate mask. Depending on the relative dimensions of image source and mask, either the image source's width or height may be cropped or not fit the respective dimension of the mask.

— **Depth:** The Depth slider controls the image plane's distance from the camera.

> **NOTE:** The Camera Z position has no effect on the image plane's distance from the camera.

Materials Tab

The options presented in the Materials tab are identical to those commonly found in other 3D nodes. For more detail on Diffuse, Specular, Transmittance, and Martial ID controls, see "The Common Controls" section at the end of this chapter.

Projection Tab

When a 2D image is connected to the camera node, a fourth projection tab is displayed at the top of the Inspector. Using this Projection tab, it is possible to project the image into the scene. A projection is different from an image plane in that the projection falls onto the geometry in the scene exactly as if there were a physical projector present in the scene. The image is projected as light, which means the Renderer 3D node must be set to enable lighting for the projection to be visible.
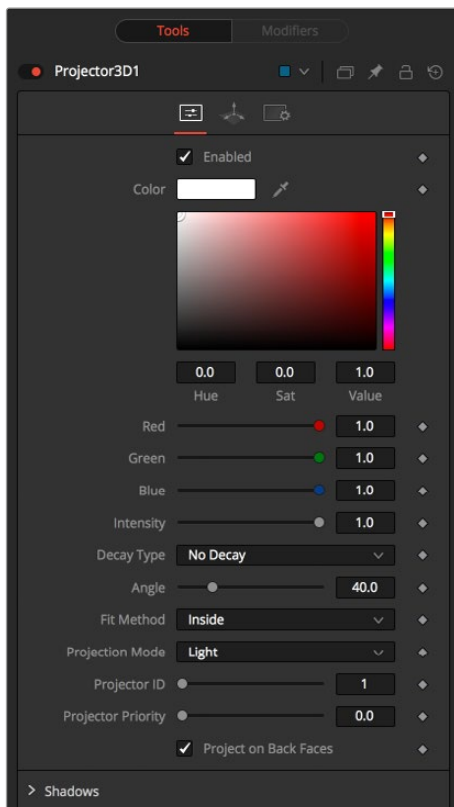
Camera 3D projection tab

Enable Camera Projection

Select this checkbox to enable projection of the 2D image connected to the magenta input on the Camera node.

Projection Fit Method

This menu can be used to select the method used to match the aspect of the projected image to the camera's field of view.

Projection Mode

— **Light:** Defines the projection as a spotlight.

— **Ambient Light:** Defines the projection as an ambient light.

— **Texture:** Allows a projection that can be relighted using other lights. Using this setting requires a Catcher node connected to the applicable inputs of the specific material.

Common Controls

Transform and Settings Tabs

The options presented in the Transform and Settings tabs are commonly found in other 3D nodes. For more detail on the controls found in these tabs, see "The Common Controls" section at the end of this chapter.

Tips for Camera 3D

Camera Projection: When importing a camera from a 3D application that is also used as a projector, make sure that the Fit Resolution Gate options on the Controls tab as well as the Projection tab are in sync. Only the first one automatically sets to what the 3D app was using. The latter might have to be adjusted manually.

Image Plane: The camera's image plane isn't just a virtual guide for you in the viewers. It's actual geometry that you can also project on to. To use a different image on the image plane, you need to insert a Replace Material node after your Camera node.

Parallel Stereo: There are three ways you can achieve real Parallel Stereo mode:

— Connect an additional external (right) camera to the green Right Stereo Camera input of your camera.

— Create separate left and right cameras.

— When using Toe-In or Off Axis, set the Convergence Distance slider to a very large value of 999999999.

Rendering Overscan: If you want to render an image with overscan, you also must modify your scene's Camera3D. Since overscan settings aren't exported along with camera data from 3D applications, this is also necessary for cameras you've imported via .fbx or .ma files. The solution is to increase the film back's width and height by the factor necessary to account for extra pixels on each side.

Cube 3D [3Cb]

The Cube 3D node

Cube 3D Node Introduction

The Cube 3D node is a basic primitive geometry type capable of generating a simple cube.

The node also provides six additional image inputs that can be used to map a texture onto the six faces of the cube. Cubes are often used as shadow casting objects and for environment maps. For other basic primitives, see the Shape 3D node in this chapter.

Inputs

The following are optional inputs that appear on the Cube3D node in the Node Editor:

— **SceneInput:** The orange scene input is used to connect another node that creates or contains a 3D scene or object. The additional geometry gets added to the Cube3D.

— **NameMaterialInput:** These six inputs are used to define the materials applied to the six faces of the cube. You can connect either a 2D image or a 3D material to these inputs. Textures or materials added to the Cube3D do not get added to any 3D objects connected to the Cube's SceneInput.

Basic Node Setup

The output of a Cube 3D node typically connects to a Merge 3D node, integrating it into a larger scene. When 3D tracking, the Cube 3D is often used as a placeholder for proper geometry that is not available at the current time.
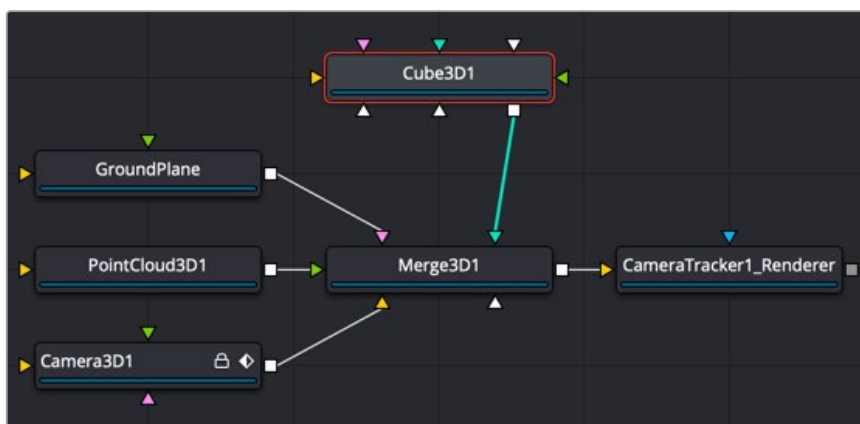
Cube 3D node connected to a 3D scene exported
from the Camera Tracker node

Inspector

Cube 3D controls

Controls Tab

The first tab in the Inspector is the Controls tab. It includes the primary controls for determining the overall size and shape of the Cube 3D node.

Lock Width/Height/Depth

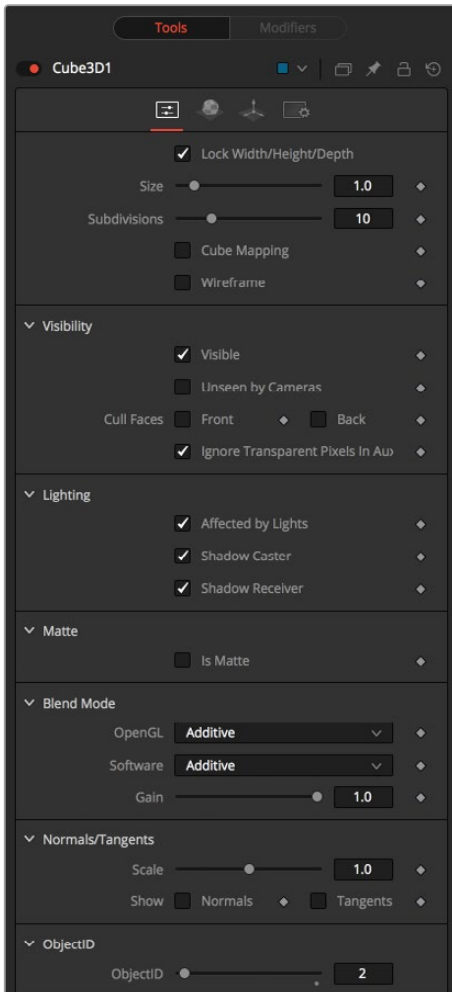This checkbox locks the Width, Height, and Depth dimensions of the cube together. When selected, only a Size control is displayed; otherwise, separate Width, Height, and Depth sliders are shown.

Size or Width/Height/Depth

If the Lock checkbox is selected, then only the Size slider is shown; otherwise, separate sliders are displayed for Width, Height, and Depth. The Size and Width sliders are the same control renamed, so any animation applied to Size is also applied to Width when the controls are unlocked.

Subdivision Level

Use the Subdivision Level slider to set the number of subdivisions used when creating the image plane.

The 3D viewers and renderer use vertex lighting, meaning all lighting is calculated at the vertices on the 3D geometry and then interpolated from there. Therefore, the more subdivisions in the mesh, the more vertices are available to represent the lighting. For example, make a sphere and set the

subdivisions to be small so it looks chunky. With lighting on, the object looks like a sphere but has some amount of fracturing resulting from the large distance between vertices. When the subdivisions are high, the vertices are closer and the lighting becomes more even. So, increasing subdivisions can be useful when working interactively with lights.

Cube Mapping

Enabling the Cube Mapping checkbox causes the cube to wrap its first texture across all six faces using a standard cubic mapping technique. This approach expects a texture laid out in the shape of a cross.

Wireframe

Enabling this checkbox causes the mesh to render only the wireframe for the object when rendering with the OpenGL renderer in the Renderer 3D node.

Common Controls

Controls, Materials, Transform, and Settings Tabs

The remaining controls for Visibility, Lighting, Matte, Blend Mode, Normals/Tangents, and Object ID are common to many 3D nodes. The same is true of the Materials, Transform, and Settings tabs. Their descriptions can be found in "The Common Controls" section at the end of this chapter.

Custom Vertex 3D [3CV]
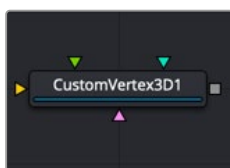
The Custom Vertex 3D node

Custom Vertex 3D Node Introduction

The Custom Vertex 3D node is an advanced custom node for 3D geometry that performs per vertex manipulations. If you have moderate experience with scripting or C++ programming, you should find the structure and terminology used by the Custom node familiar.

Using scripting math functions and lookup tables from images, you can move vertex positions on 3D geometry. Vertices can be more than just positions in 3D space. You can manipulate normals, texture coordinates, vectors, and velocity.

For example, Custom Vertex 3D can be used to make a flat plane wave like a flag, or create spiral models.

Besides providing a 3D scene input and three image inputs, the Inspector includes up to eight number fields and as many as eight XYZ position values from other controls and parameters in the node tree.

> **NOTE:** Modifying the X, Y, and Z positions of a 3D object does not modify the normals/tangents. You can use a ReplaceNormals node afterward to recompute the normals/tangents.

> **TIP:** Not all geometry has every attribute. For example, most Fusion geometry does not have vertex colors, with the exception of particles and some imported FBX/Alembic meshes. No geometry currently has environment coordinates, and only particles have velocities. If an attribute is not present on the input geometry, it is assumed to have a default value.

Inputs

The Custom Vertex 3D node includes four inputs. The orange scene input is the only one of the four that is required.

— **SceneInput:** The orange scene input takes 3D geometry or a 3D scene from a 3D node output. This is the 3D scene or geometry that is manipulated by the calculations in the Custom Vertex 3D node.

— **ImageInput1, ImageInput2, ImageInput3:** The three image inputs using green, magenta, and teal colors are optional inputs that can be used for compositing.

> **NOTE:** Missing attributes on the input geometry are created if the expression for an attribute is nontrivial. The values for the attributes are given as in the above point. For example, if the input geometry does not have normals, then the values of (nx, ny, nz) is always (0,0,1). To change this, you could use a ReplaceNormals node beforehand to generate them.

Basic Node Setup

The object you want to manipulate connects to the orange scene input of the Custom Vertex 3D node. The output typically connects to a Merge 3D node, integrating it into a larger scene.
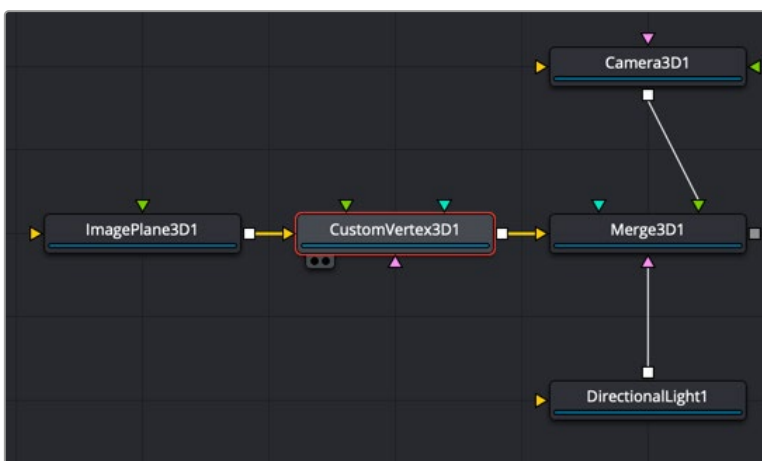
Custom Vertex 3D node manipulating an Image Plane 3D node

Inspector

Custom Vertex 3D node Vertex tab

Vertex Tab

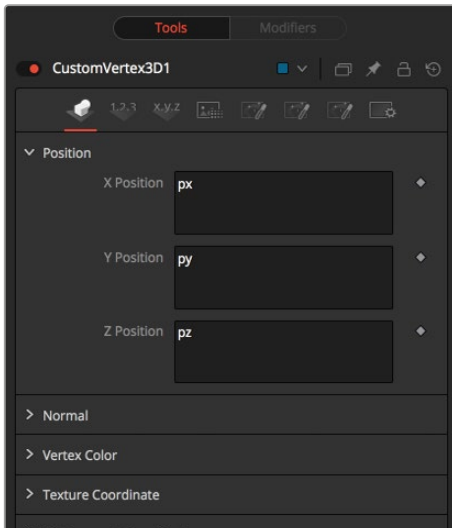Using the fields in the Vertex tab, vertex calculations can be performed on the Position, Normals, Vertex Color, Texture Coordinates, Environment Coordinates, UV Tangents, and Velocity attributes.

The vertices are defined by three XYZ Position values in world space as px, py, pz. Normals, which define as a vector the direction the vertex is pointing as nx, ny, nz.

Vertex color is the Red, Green, Blue, and Alpha color of the point as vcr, vcg, vcb, vca.

Numbers Tab
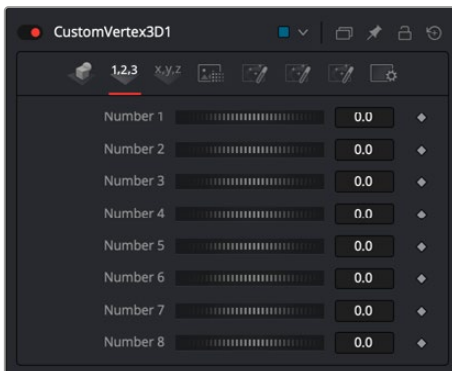
Custom Vertex 3D node Numbers tab

Numbers 1-8

Numbers are variables with a dial control that can be animated or connected to modifiers exactly as any other control might. The numbers can be used in equations on vertices at current time: n1, n2, n3, n4,... or at any time: n1_at(float t), n2_at(float t), n3_at(float t), n4_at(float t), where t is the time you want. The values of these controls are available to expressions in the Setup and Intermediate tabs. They can be renamed and hidden from the viewer using the Config tab.

Points Tab

Custom Vertex 3D node Points tab

Points 1-8

The point controls represent points in the Custom Vertex 3D tool, not the vertices. These eight point controls include 3D X,Y,Z position controls for positioning points at the current time: (p1x, p1y, p1z, p2x, p2y, p2z) or at any time: p1x_at(float t), p1y_at(float t), p1z_at(float t), p2x_at(float t), p2y_at(float t), p2z_at(float t), where t is the time you want. For example, you can use a point to define a position in 3D space to rotate the vertices around. They can be renamed and hidden from the viewer using the Config tab. They are normal positional controls and can be animated or connected to modifiers as any other node might.

LUT Tab
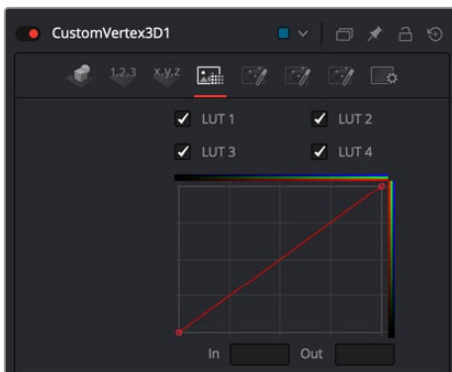
Custom Vertex 3D node LUT tab

LUTs 1-4

The Custom Vertex 3D node provides four LUT splines. A LUT is a lookup table that will return a value from the height of the LUT spline. For example, getlut1(float x), getlut2(float x),… where x = 0 … 1 accesses the LUT values.

The values of these controls are available to expressions in the Setup and Intermediate tabs using the getlut# function. For example, setting the R, G, B, and A expressions to getlut1(r1), getlut2(g1), getlut3(b1), and getlut4(a1) respectively, would cause the Custom Vertex 3D node to mimic the Color Curves node.

These controls can be renamed using the options in the Config tab to make their meanings more apparent, but expressions still see the values as lut1, lut2,...lut8.

Setup Tab
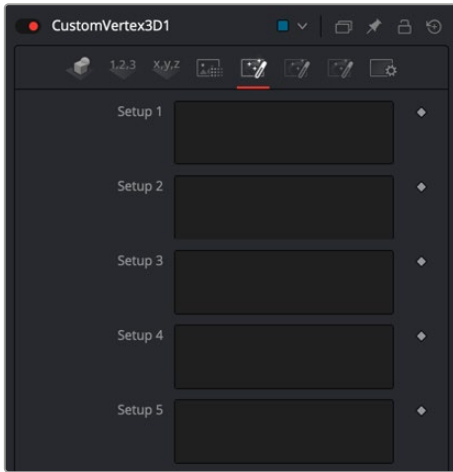
Custom Vertex 3D node Setup tab

Setups 1-8

Up to eight separate expressions can be calculated in the Setup tab of the Custom Vertex 3D node. The Setup expressions are evaluated once per frame, before any other calculations are performed. The results are then made available to the other expressions in the node as variables s1, s2, s3, and s4.

Think of them as global setup scripts that can be referenced by the intermediate and channel scripts for each vertex.

For example, Setup scripts can be used to transform vertex from model to world space.

> **NOTE:** Because these expressions are evaluated once per frame only and not for each pixel, it makes no sense to use per-pixel variables like X and Y or channel variables like r1, g1, b1, and so on. Allowable values include constants, variables like n1...n8, time, W and H, and so on, and functions like sin() or getr1d().

Intermediate Tab
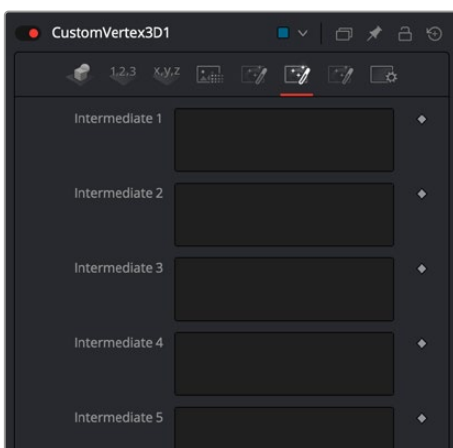
Custom Vertex 3D Node Intermediate tab

Intermediates 1-8

An additional eight expressions can be calculated in the Intermediate tab. The Intermediate expressions are evaluated once per vertex, after the Setup expressions are evaluated. Results are available as variables i1, i2, i3, i4, i5, i6, i7, i8, which can be referenced by channel scripts. Think of them as "per vertex setup" scripts.

For example, you can run the script to produce the new vertex (i.e., new position, normal, tangent, UVs, etc.) or transform from world space back to model space.

Config Tab
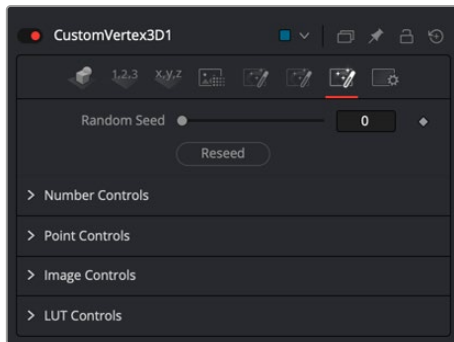
Custom Vertex 3D node Config tab

Random Seed

Use this to set the seed for the rand() and rands() functions. Click the Reseed button to set the seed to a random value. This control may be needed if multiple Custom Vertex 3D nodes are required with different random results for each.

Number Controls

There are eight sets of Number controls, corresponding to the eight sliders in the Numbers tab. Disable the Show Number checkbox to hide the corresponding Number slider, or edit the Name for Number text field to change its name.

Point Controls

There are eight sets of Point controls, corresponding to the eight controls in the Points tab. Disable the Show Point checkbox to hide the corresponding Point control and its crosshair in the viewer. Similarly, edit the Name for Point text field to change the control's name.

Common Controls

Settings Tab

The Settings tab controls are common to many 3D nodes, and their descriptions can be found in "The Common Controls" section at the end of this chapter.

Displace 3D [3Di]

The Displace 3D node

Displace 3D Node Introduction

The Displace 3D node is used to displace the vertices of an object along their normals based on a reference image. The texture coordinates on the geometry are used to determine where to sample the image.

When using Displace 3D, keep in mind that it only displaces existing vertices and does not subdivide surfaces to increase detail. To obtain a more detailed displacement, increase the subdivision amount for the geometry that is being displaced. Note that the pixels in the displacement image may contain negative values.

> **TIP:** Passing a particle system through a Displace 3D node disables the Always Face Camera option set in the pEmitter. Particles are not treated as point-like objects; each of the four particle vertices are individually displaced, which may or may not be the preferred outcome.

Inputs

The following two inputs appear on the Displace 3D node in the Node Editor:

— **SceneInput:** The orange scene input is the required input for the Displace 3D node. You use this input to connect another node that creates or contains a 3D scene or object.

— **Input:** This green input is used to connect a 2D image that is used to displace the object connected to the Scene input. If no image is provided, this node effectively passes the scene straight through to its output. So, although not technically a required input, there isn't much use for adding this node unless you connect this input correctly.

Basic Node Setup

The output of a Displace 3D node typically connects to a Merge 3D node, integrating it into a larger scene. The 3D geometry you want to displace is connected to the orange input, and in this example, a Fast Noise node is used to displace the geometry.
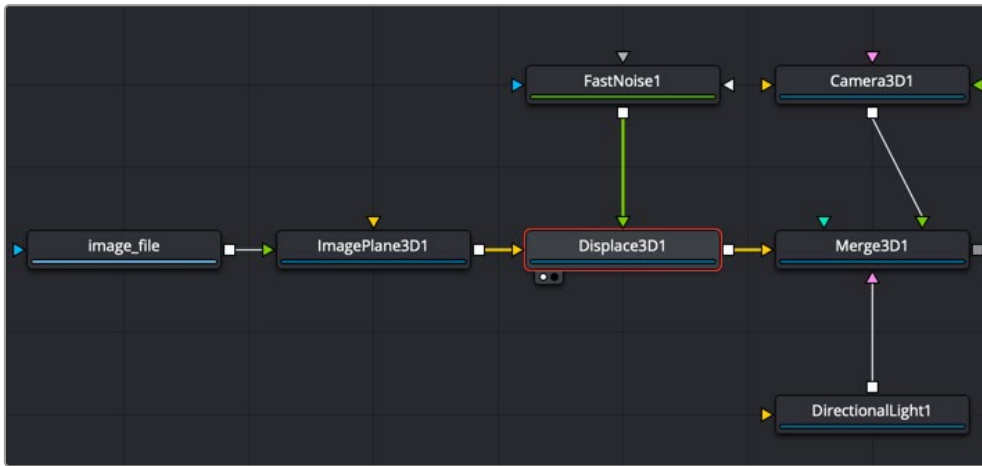
An image on an Image Plane 3D is displaced by a Fast Noise node

Inspector
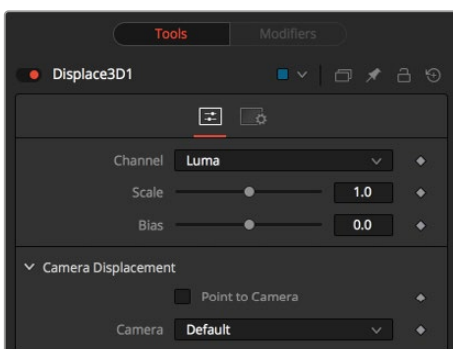
Displace 3D controls

Controls Tab

The Displace 3D Inspector includes two tabs along the top. The primary tab, called the Controls tab, contains the dedicated Displace 3D controls.

Channel

Determines which channel of the connected input image is used to displace the geometry.

Scale and Bias

Use these sliders to scale (magnify) and bias (offset) the displacement. The bias is applied first and the scale afterward.

Camera Displacement

— **Point to Camera:** When the Point to Camera checkbox is enabled, each vertex is displaced toward the camera instead of along its normal. One possible use of this option is for displacing a camera's image plane. The displaced camera image plane would appear unchanged when viewed through the camera but is deformed in 3D space, allowing one to comp-in other 3D layers that correctly interact in Z.

— **Camera:** This menu is used to select which camera in the scene is used to determine the camera displacement when the Point to Camera option is selected.

Common Controls

Settings Tab

The Settings tab controls are common to many 3D nodes, and their descriptions can be found in "The Common Controls" section at the end of this chapter.

Duplicate 3D [3Dp]
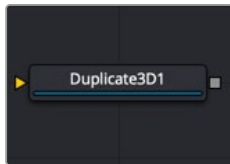
The Duplicate 3D node

Duplicate 3D Node Introduction

Similar to the 2D version called the Duplicate node, the Duplicate 3D node can be used to duplicate any geometry in a scene, applying a successive transformation to each, and creating repeating patterns and complex arrays of objects. The options in the Jitter tab allow non-uniform transformations, such as random positioning or sizes.

Inputs

The Duplicate 3D node has a single input by default where you connect a 3D scene. An optional Mesh input appears based on the settings of the node.

— **SceneInput:** The orange Scene Input is a required input. The scene or object you connect to this input is duplicated based on the settings in the Control tab of the Inspector.

— **MeshInput:** A green optional mesh input appears when the Region's tab Region menu is set to mesh. The mesh can be any 3D model, either generated in Fusion or imported.

Basic Node Setup

The output of a Duplicate 3D node typically connects to a Merge 3D node, integrating it into a larger scene. The 3D geometry you want duplicated, in this case a Cube 3D, is connected to the orange input.
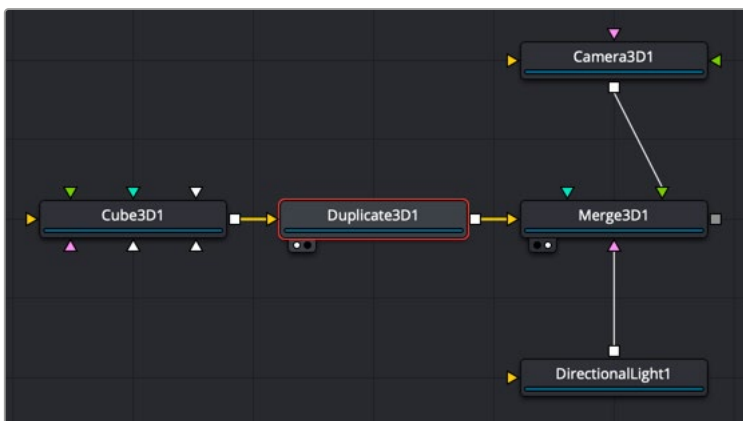
A Cube 3D is duplicated

Inspector
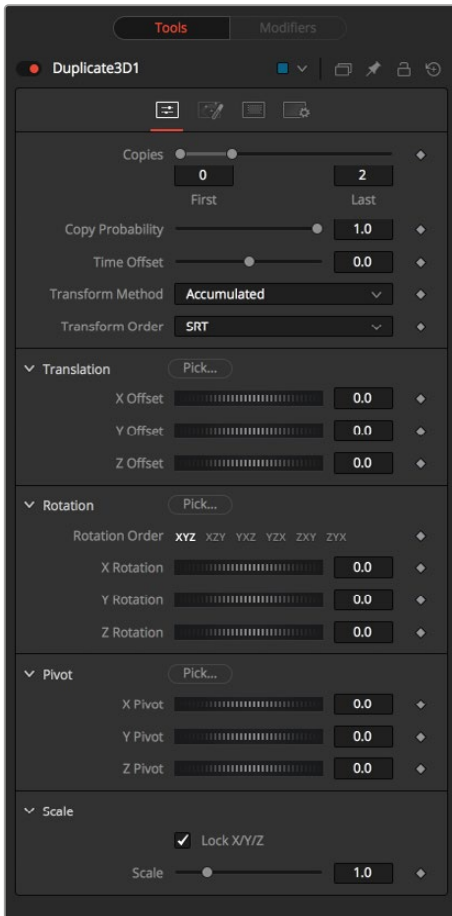
Duplicate 3D controls

Controls Tab

The Controls tab includes all the parameters you can use to create, offset, and scale copies of the object connected to the scene input on the node.

Copies

Use this range control to set the number of copies made. Each copy is a copy of the last copy, so if this control is set to [0,3], the parent is copied, then the copy is copied, then the copy of the copy is copied, and so on. This allows some interesting effects when transformations are applied to each copy using the controls below.

Setting the First Copy to a value greater than 0 excludes the original object and shows only the copies.

Time Offset

Use the Time Offset slider to offset any animations that are applied to the source geometry by a set amount per copy. For example, set the value to -1.0 and use a cube set to rotate on the Y-axis as the source. The first copy shows the animation from a frame earlier; the second copy shows animation from a frame before that, etc. This can be used with great effect on textured planes—for example, where successive frames of a clip can be shown.

Transform Method

— **Linear:** When set to Linear, transforms are multiplied by the number of the copy, and the total scale, rotation, and translation are applied in turn, independent of the other copies.

— **Accumulated:** When set to Accumulated, each object copy starts at the position of the previous object and is transformed from there. The result is transformed again for the next copy
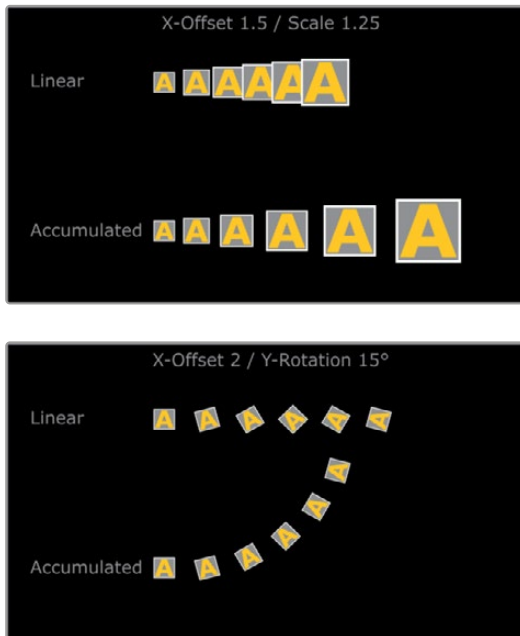
Transform Order

With this menu, the order in which the transforms are calculated can be set. It defaults to Scale-Rotation-Transform (SRT).

Using different orders results in different positions of your final objects.

Translation

The X, Y, and Z Offset sliders set the offset position applied to each copy. An X offset of 1 would offset each copy 1 unit along the X-axis from the last copy.

Rotation

The buttons along the top of this group of rotation controls set the order in which rotations are applied to the geometry. Setting the rotation order to XYZ would apply the rotation on the X-axis first, followed by the Y-axis rotation, then the Z-axis rotation.

The three Rotation sliders set the amount of rotation applied to each copy.

Pivot

The pivot controls determine the position of the pivot point used when rotating each copy.

Scale

— **Lock:** When the Lock XYZ checkbox is selected, any adjustment to the duplicate scale is applied to all three axes simultaneously. If this checkbox is disabled, the Scale slider is replaced with individual sliders for the X, Y, and Z scales.

— **Scale:** The Scale controls tell Duplicate how much scaling to apply to each copy.

Jitter Tab

The options in the Jitter tab allow you to randomize the position, rotation, and size of all the copies created in the Controls tab.
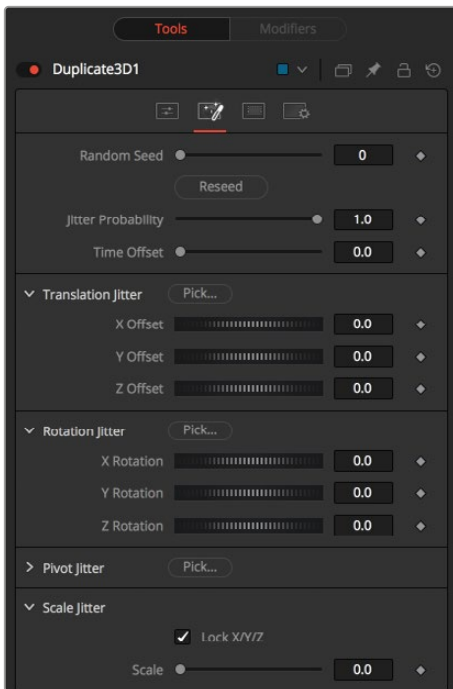
Duplicate 3D Jitter tab

Random Seed

The Random Seed slider is used to generate a random starting point for the amount of jitter applied to the duplicated objects. Two Duplicate nodes with identical settings but different random seeds produce two completely different results.

Randomize

Click the Randomize button to auto generate a random seed value.

Jitter Probability

Adjusting this slider determines the percentage of copies that are affected by the jitter. A value of 1.0 means 100% of the copies are affected, while a value of 0.5 means 50% are affected.

Time Offset

Use the Time Offset slider to offset any animations that are applied to the source geometry by a set amount per copy. For example, set the value to –1.0 and use a cube set to rotate on the Y-axis as the source. The first copy shows the animation from a frame earlier; the second copy shows animation from a frame before that, etc. This can be used with great effect on textured planes—for example, where successive frames of a clip can be shown.

Translation Jitter

Use these three controls to adjust the amount of variation in the X, Y, and Z translation of the duplicated objects.

Rotation Jitter

Use these three controls to adjust the amount of variation in the X, Y, and Z rotation of the duplicated objects.

Pivot Jitter

Use these three controls to adjust the amount of variation in the rotational pivot center of the duplicated objects. This affects only the additional jitter rotation, not the rotation produced by the Rotation settings in the Controls tab.

Scale Jitter

Use this control to adjust the amount of variation in the scale of the duplicated objects. Disable the Lock XYZ checkbox to adjust the scale variation independently on all three axes.

Region Tab

The options in the Region tab allow you to define an area in the viewer where the copies can appear or are prevented from appearing. Like most parameters in Fusion, this area can be animated to cause the copied object to pop on and off the screen based on the region's shape and setting.
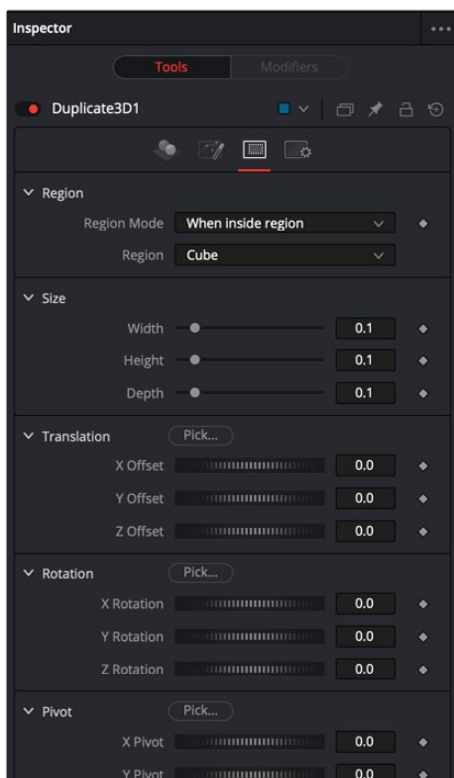
Duplicate 3D Region tab

Region Tab

The Region section includes two settings for controlling the shape of the region and the affect the region has on the duplicate objects.

— **Region Mode:** There are three options in the Region Mode menu. The default, labeled "Ignore region" bypasses the node entirely and causes no change to the copies of objects from how they are set in the Controls and Jitter tabs. The menu option labeled "When inside region" causes the copied objects to appear only when their position falls inside the region defined in this tab. The last menu option, "When not Inside region" causes the copied objects to appear only when their position falls outside the region defined in this tab.

— **Region:** The Region menu determines the shape of the region. The five options include cube, sphere, and rectangle primitive shapes. The mesh option allows you to connect a 3D model into the green mesh input on the node. The green input appears only after the Region menu is set to Mesh. The All setting refers to the entire scene. This allows the copies to pop on and off if the Region mode is animated. When the Region menu is set to Mesh, four other options are displayed. These are described below.

— **Winding Rule:** Using four common techniques, the Winding Rule menu determines how the mesh of polygons is determined as an area of volume and consequently how copies locate the vertices in the mesh. Complex overlapping regions of a mesh can cause an irregular fit. Trying a different technique from this menu can sometimes create a better match between the mesh and how the copies interpret the mesh shape.

— **Winding Ray Direction:** A 3D model is a mesh of vertices made up of flat polygons. When making this a volume for a region, the Winding Ray Direction is used to determine in which direction the volume of each polygon (like depth extrude) is aligned.

— **Limit by Object ID:** When a scene with multiple meshes is connected to the green Mesh input on the node, all the meshes are used as the region. Enabling this checkbox allows you to use the Object ID slider to select the ID for the mesh you want to use as the Region.

— **Object ID:** When the Limit by Object ID checkbox is enabled, this slider selects the number ID for the mesh object you want to use for the Region.

Common Controls

Settings Tab

The Settings tab controls are common to many 3D nodes, and their descriptions can be found in "The Common Controls" section at the end of this chapter.

FBX Exporter 3D [FBX]

The FBX Exporter node

FBX Exporter Node Introduction

The FBX Exporter node provides a method of exporting a Fusion 3D scene to the FBX scene interchange format. Each node in Fusion is a single object in the exported file. Objects, lights, and cameras use the name of the node that created them. The node can be set to export a single file for the entire scene, or to output one frame per file.

Setting the Preferences > Global > General > Auto Clip Browse option in the Fusion Studio application, or the Fusion > Fusion Settings > General > Auto Clip Browse option in DaVinci Resolve to Enabled (default), and then adding this node to a composition automatically displays a file browser allowing you to choose where to save the file.

Once you have set up the node, the FBX Exporter is used similarly to a Saver node: clicking the Render button in the toolbar renders out the file.

Besides the FBX format, this node can also export to the 3D Studio's .3ds, Collada's .dae, Autocad's .dxf, and the Alias .obj formats.

Inputs

The FBX Exporter node has a single orange input.

— **Input:** The output of the 3D scene that you want to export connects to the orange input on the FBX Exporter node.

Basic Node Setup

The input to the FBX Exporter 3D node is any 3D scene you want to export. Below, the node is placed as a separate branch off of the Duplicate 3D node. Only the objects generated by the Duplicate 3D node are exported.
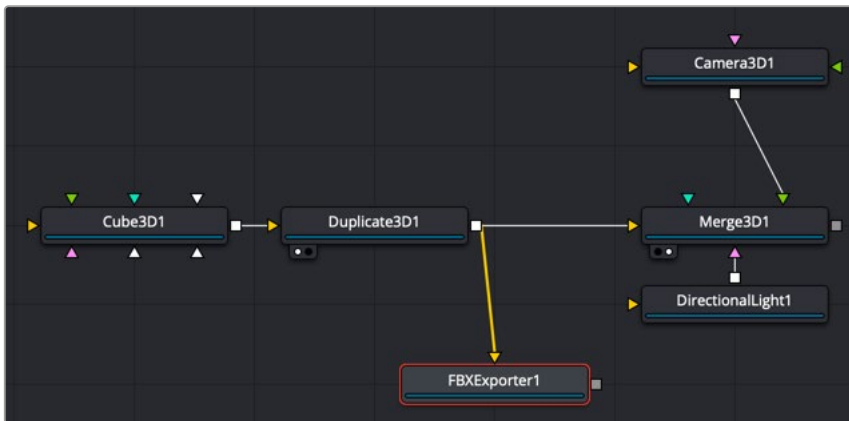
An FBX Exporter 3D branched off from the Duplicate 3D node

Inspector
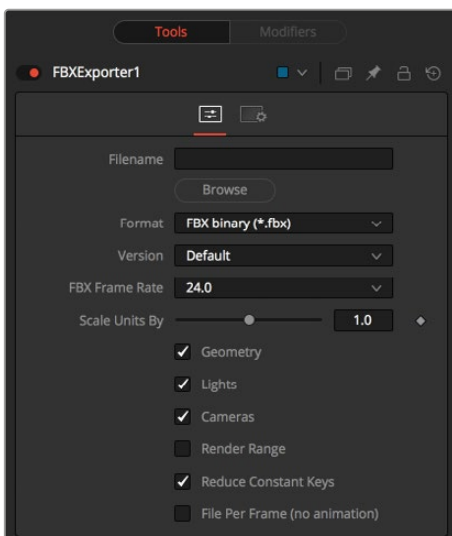
FBX Exporter controls

Controls Tab

The Controls tab includes all the parameters you used to decide how the FBX file is created and what elements in the scene get exported.

Filename

This Filename field is used to display the location and file that is output by the node. You can click the Browse button to open a file browser dialog and change the location where the file is saved.

Format

This menu is used to set the format of the output file.

Not all features of this node are supported in all file formats. For example, the .obj format does not handle animation.

Version

The Version menu is used to select the available versions for the chosen format. The menu's contents change dynamically to reflect the available versions for that format. If the selected format provides only a single option, this menu is hidden.

Choosing Default for the FBX formats uses FBX2011.

Frame Rate

This menu sets the frame rate that is in the FBX scene.

Scale Units By

This slider changes the working units in the exported FBX file. Changing this can simplify workflows where the destination 3D software that you have uses a different scale.

Geometry/Lights/Cameras

These three checkboxes determine whether the node attempts to export the named scene element. For example, deselecting Geometry and Lights but leaving Cameras selected would output only the cameras currently in the scene.

Render Range

Enabling this checkbox saves the Render Range information in the export file, so other applications know the time range of the FBX scene.

Reduce Constant Keys

Enabling this option automatically removes keyframes if the adjacent keyframes have the same value.

File Per Frame (No Animation)

Enabling this option forces the node to export one file per frame, resulting in a sequence of numbered files. This disables the export of animation. Enable this checkbox to reveal the Sequence Start Frame control where you can set the first frame in the sequence to a custom value.

Sequence Start Frame

Enabling this checkbox displays a thumbwheel control to set a specific start frame for the number sequence applied to the rendered filenames. For example, if Global Start is set to 1 and frames 1–30 are rendered, files are normally numbered 0001–0030. If the Sequence Start frame is set to 100, the rendered output is numbered from 100–131.

Common Controls

Settings Tab

The Settings tab controls are common to many 3D nodes, and their descriptions can be found in "The Common Controls" section at the end of this chapter.

FBX Mesh 3D [FBX]

The FBX Mesh node

FBX Mesh 3D Node Introduction

The FBX Mesh 3D node is used to import polygonal geometry from scene files that are saved in the FilmBox (FBX) format. It is also able to import geometry from OBJ, 3DS, DAE, and DXF scene files. This provides a method for working with more complex geometry than is available using Fusion's built-in primitives.

When importing geometry with this node, all the geometry in the FBX file is combined into one mesh with a single pivot and transformation. The FBX Mesh node ignores any animation applied to the geometry.

Alternatively, in Fusion Studio, the File > Import > FBX Scene or in DaVinci Resolve, the Fusion > Import > FBX Scene menu can be used to import an FBX scene. This option creates individual nodes for each camera, light, and mesh in the file. This menu option can also be used to preserve the animation of the objects.

Setting the Preferences > Global > General > Auto Clip Browse option in Fusion Studio, or the Fusion > Fusion Settings > General > Auto Clip Browse option in DaVinci Resolve to Enabled (default), and then adding this node to a composition automatically displays a file browser allowing you to choose the file to import.

Inputs

— **SceneInput:** The orange scene input is an optional connection if you wish to combine other 3D geometry nodes with the imported FBX file.

— **Material Input:** The green input is the material input that accepts either a 2D image or a 3D material. If a 2D image is provided, it is used as a diffuse texture map for the basic material tab in the node. If a 3D material is connected, then the basic material tab is disabled.

Basic Node Setup

The FBX Mesh 3D node can be used as a stand-alone node without any other nodes connected to it. The output is connected to a Merge 3D, integrating the FBX model into a lager scene. Below, the FBX Mesh 3D node also has a chrome material connected to its material input.
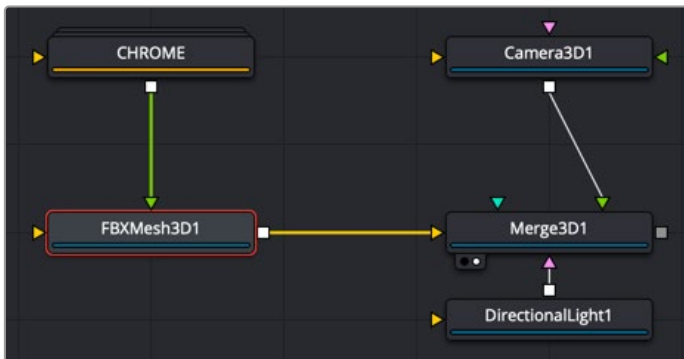
An FBX Mesh 3D node with chrome material applied

Inspector
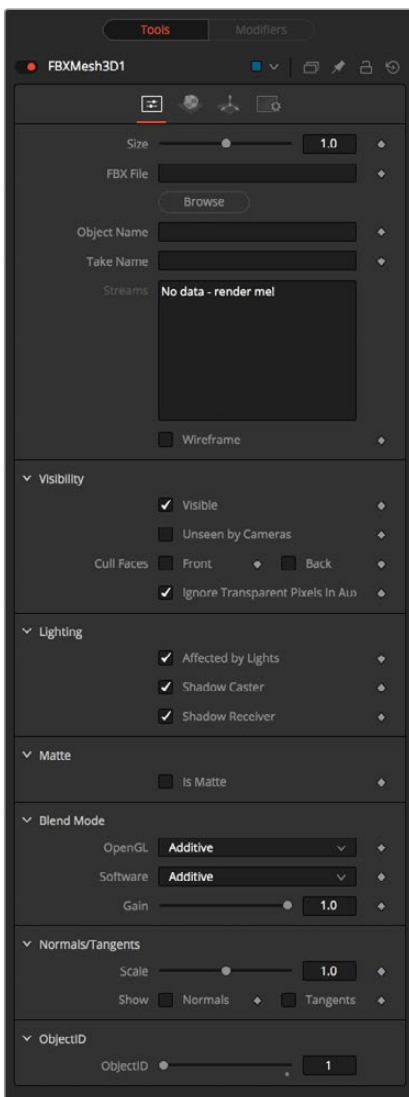
FBX Mesh 3D controls

Controls Tab

Most of the Controls tab is taken up by common controls. The FBX-specific controls included on this tab are primarily information and not adjustments.

Size

The Size slider controls the size of the FBX geometry that is imported. FBX meshes have a tendency to be much larger than Fusion's default unit scale, so this control is useful for scaling the imported geometry to match the Fusion environment.

FBX File

This field displays the filename and file path of the currently loaded FBX mesh. Click the Browse button to open a file browser that can be used to locate a new FBX file. Despite the node's name, this node is also able to load a variety of other formats.

FBX ascii	`(*.fbx)`
FBX 5.0 binary	`(*.fbx)`
Autocad DXF	`(*.dxf)`
3D Studio 3Ds	`(*.3ds)`
Alias OBJ	`(*.obj)`
Collada DAE	`(*.dae)`

Object Name

This input shows the name of the mesh from the FBX file that is being imported. If this field is blank, then the contents of the FBX geometry are imported as a single mesh. You cannot edit this field; it is set by Fusion when using the File > Import > FBX Scene menu.

Take Name

FBX files can contain multiple instances of an animation, called Takes. This field shows the name of the animation take to use from the FBX file. If this field is blank, then no animation is imported. You cannot edit this field; it is set by Fusion when using the File > Import > FBX Scene menu.

Wireframe

Enabling this checkbox causes the mesh to render only the wireframe for the object. Only the OpenGL renderer in the Renderer 3D node supports wireframe rendering.

Common Controls

Controls, Materials, Transform, and Settings Tabs

The remaining controls for Visibility, Lighting, Matte, Blend Mode, Normals/Tangents, and Object ID are common to many 3D nodes. The same is true of the Materials, Transform, and Settings tabs. Their descriptions can be found in "The Common Controls" section at the end of this chapter.

Fog 3D [3Fo]

The Fog 3D node

Fog 3D Node Introduction

The Fog 3D node applies fog to the scene based on a depth cue. It is the 3D version of the Fog node in the Deep Pixel category. It is designed to work completely in 3D space and takes full advantage of anti-aliasing and depth of field effects during rendering.

The Fog 3D node essentially retextures the geometry in the scene by applying a color correction based on the object's distance from the camera. An optional density texture image can be used to apply variation to the correction.

Inputs

The Fog 3D node has two inputs in the Node Editor, only one of which is required for the Fog 3D to project onto a 3D scene.

— **SceneInput:** The required orange-colored input accepts the output of a 3D scene on which the fog is "projected."

— **DensityTexture:** This optional green-colored input accepts a 2D image. The color of the fog created by this node is multiplied by the pixels in this image. When creating the image for the density texture, keep in mind that the texture is effectively projected onto the scene from the camera.

Basic Node Setup

The Fog 3D node is placed after the Merge 3D node that contains the scene. Viewing the Fog node will show the fog applied to the objects in the 3D scene based on their Z position.
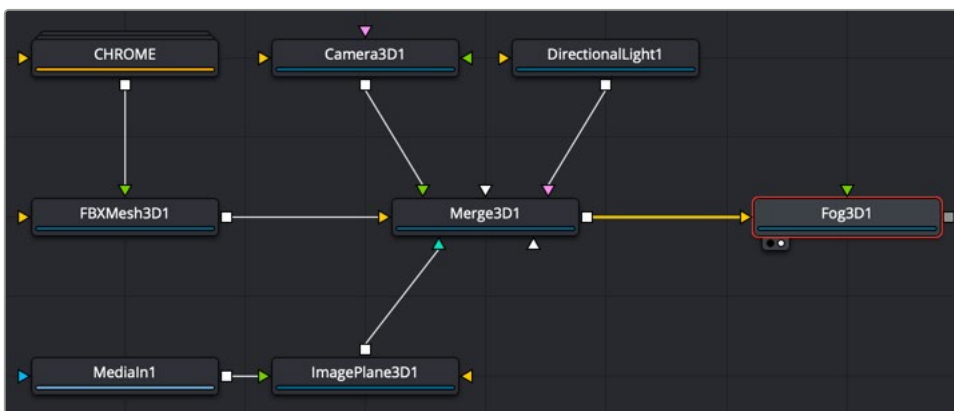
A Fog 3D node placed after the Merge 3D scene

Inspector
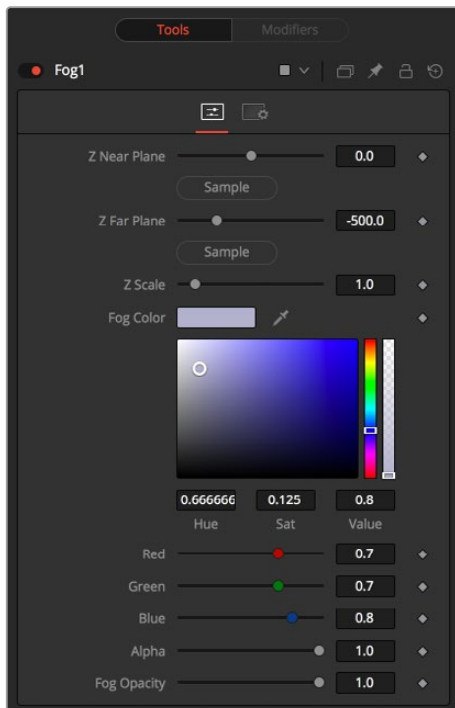
Fog node controls

Controls Tab

The Controls tab includes all the parameters you use to decide how the Fog looks and projects onto the geometry in the scene.

Enable

Use this checkbox to enable or disable parts of the node from processing. This is not the same as the red switch in the upper-left corner of the inspector. The red switch disables the tool altogether and passes the image on without any modification. The Enable checkbox is limited to the effect part of the tool. Other parts like scripts in the Settings tab still processes as normal.

Show Fog in View

By default, the fog created by this node is visible only when the scene is viewed using a camera node. When this checkbox is enabled, the fog becomes visible in the scene from all points of view.

Color

This control can be used to set the color of the fog. The color is also multiplied by the density texture image, if one is connected to the green input on the node.

Radial

By default, the fog is created based on the perpendicular distance to a plane (parallel with the near plane) passing through the eye point. When the Radial option is checked, the radial distance to the eye point is used instead of the perpendicular distance. The problem with perpendicular distance fog is that when you move the camera about, as objects on the left or right side of the frustum move into the center, they become less fogged although they remain the same distance from the eye. Radial fog fixes this. Radial fog is not always desirable, however.

For example, if you are fogging an object close to the camera, like an image plane, the center of the image plane could be unfogged while the edges could be fully fogged.
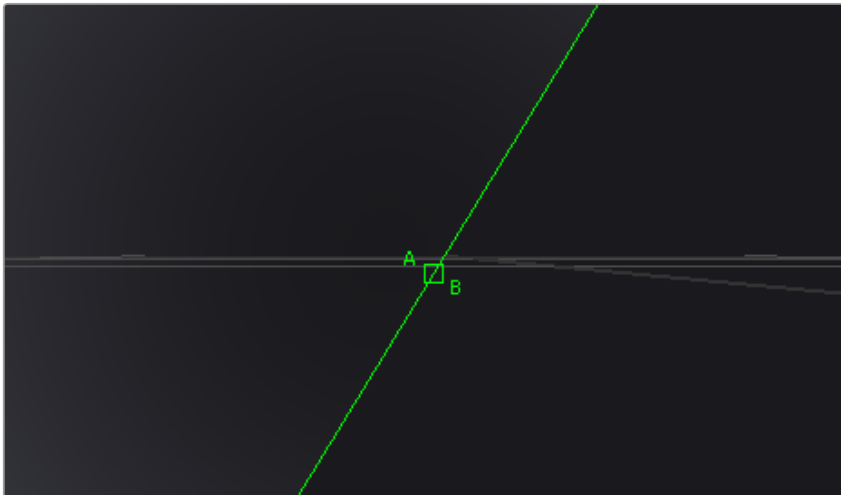
Radial vs. Perpendicular fog effect

Type

This control is used to determine the type of falloff applied to the fog.

— **Linear:** Defines a linear falloff for the fog.
— **Exp:** Creates an exponential nonlinear falloff.
— **Exp2:** Creates a stronger exponential falloff.

Near/Far Fog Distance

This control expresses the range of the fog in the scene as units of distance from the camera. The Near Distance determines where the fog starts, while the Far Distance sets the point where the fog has its maximum effect. Fog is cumulative, so the farther an object is from the camera, the thicker the fog should appear.

Common Controls

Settings Tab

The Settings tab controls are common to many 3D nodes, and their descriptions can be found in "The Common Controls" section at the end of this chapter.

Image Plane 3D [3Im]

The Image Plane 3D node

Image Plane 3D Node Introduction

The Image Plane node produces 2D planar geometry in 3D space. The node is most commonly used to represent 2D images on "cards" in the 3D space. The aspect of the image plane is determined by the aspect of the image connected to the material input. If you do not want the aspect ratio of the image to modify the "card" geometry, then use a Shape 3D node instead.

Inputs

Of the two inputs on this node, the material input is the primary connection you use to add an image to the planar geometry created in this node.

— **SceneInput:** This orange input expects a 3D scene. As this node creates flat, planar geometry, this input is not required.

— **MaterialInput:** The green-colored material input accepts either a 2D image or a 3D material. It provides the texture and aspect ratio for the rectangle based on the connected source such as a Loader node in Fusion Studio or a MediaIn node in DaVinci Resolve. The 2D image is used as a diffuse texture map for the basic material tab in the Inspector. If a 3D material is connected, then the basic material tab is disabled.

Basic Node Setup

The Image Plane 3D node is primarily used to bring a video clip into a 3D composite. The MediaIn or Loader node is connected to the Image Plane 3D node, and the Image Plane 3D is then connected to a Merge 3D node. Viewing the Merge 3D node will show all the Image Plane 3D nodes and other elements connected to it.
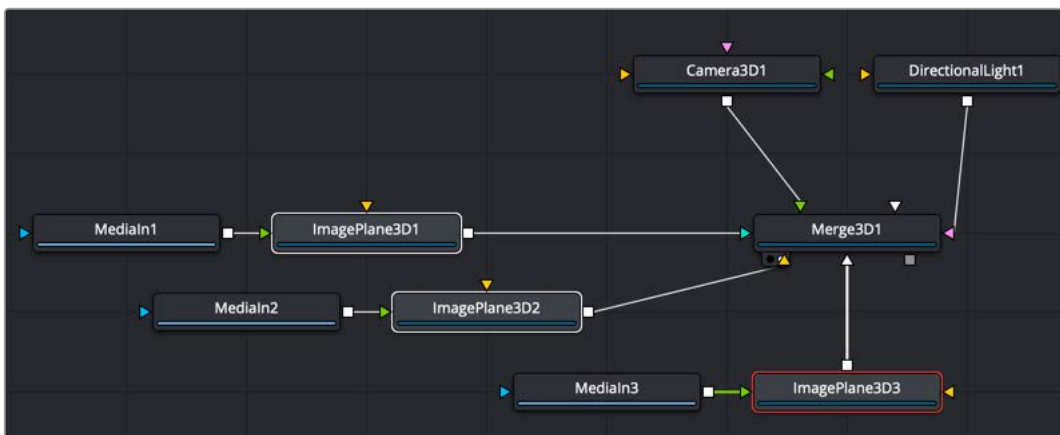
Multiple Image Plane 3D nodes connected to a Merge 3D

Inspector
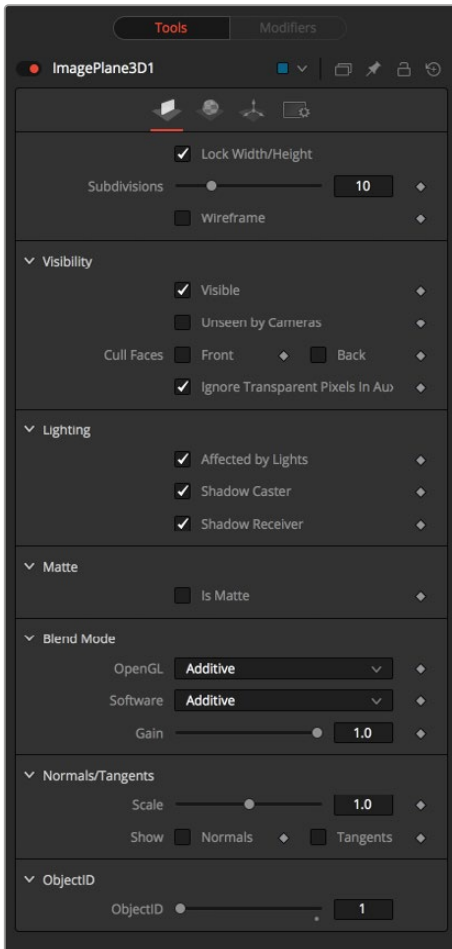
Image Plane 3D controls

Controls Tab

Most of the Controls tab is taken up by common controls. The Image Plane specific controls at the top of the Inspector allow minor adjustments.

Lock Width/Height

When checked, the subdivision of the plane is applied evenly in X and Y. When unchecked, there are two sliders for individual control of the subdivisions in X and Y. This defaults to on.

Subdivision Level

Use the Subdivision Level slider to set the number of subdivisions used when creating the image plane. If the Open GL viewer and renderer are set to Vertex lighting, the more subdivisions in the mesh, the more vertices are available to represent the lighting. So, high subdivisions can be useful when working interactively with lights.

Wireframe

Enabling this checkbox causes the mesh to render only the wireframe for the object when using the OpenGL renderer.

Common Controls

Controls, Materials, Transform, and Settings Tabs

The remaining controls for Visibility, Lighting, Matte, Blend Mode, Normals/Tangents, and Object ID are common to many 3D nodes. The same is true of the Materials, Transform, and Settings tabs. Their descriptions can be found in "The Common Controls" section at the end of this chapter.

Locator 3D [3Lo]
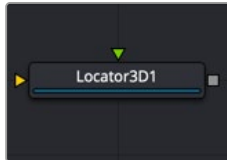
The Locator 3D node

Locator 3D Node Introduction

The Locator 3D node's purpose is to transform a point in 3D space to 2D coordinates that other nodes can use as part of expressions or modifiers.

When the Locator is provided with a camera and the dimensions of the output image, it transforms the coordinates of a 3D control into 2D screen space. The 2D position is exposed as a numeric output that can be connected to/from other nodes. For example, to connect the center of an ellipse to the 2D position of the Locator, right-click on the Mask center control and select Connect To > Locator 3D > Position.

Inputs

Two inputs accept 3D scenes as sources. The orange scene input is required, while the green Target input is optional.

— **SceneInput:** The required orange scene input accepts the output of a 3D scene. This scene should contain the object or point in 3D space that you want to covert to 2D coordinates.

— **Target:** The optional green target input accepts the output of a 3D scene. When provided, the transform center of the scene is used to set the position of the Locator. The transformation controls for the Locator become offsets from this position.

Basic Node Setup

The scene provided to the Locator's input must contain the camera through which the coordinates are projected. So, the best practice is to place the Locator after the Merge that introduces the camera to the scene.

If an object is connected to the Locator node's target input, the Locator is positioned at the object's center, and the Transformation tab's translation XYZ sliders function in the object's local coordinate space instead of global scene space. This is useful for tracking an object's position despite any additional transformations applied further downstream.
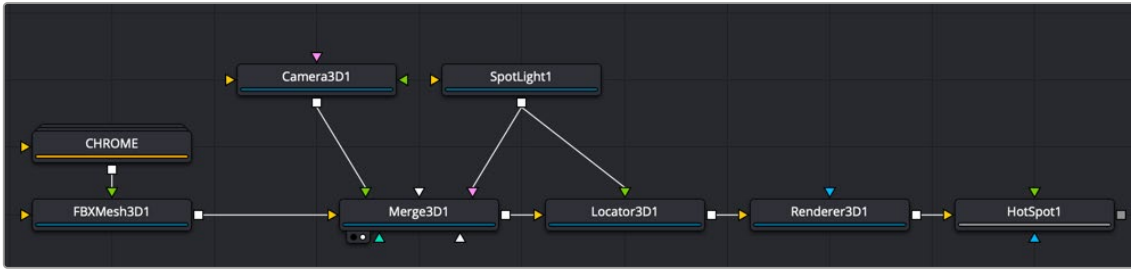
Locator 3D connected after a Merge 3D with the SpotLight as the target

Inspector
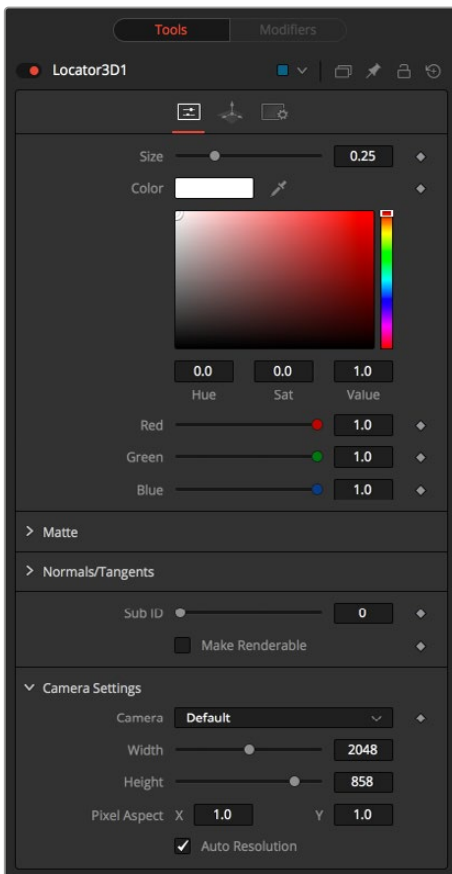
Locator 3D controls

Controls Tab

Most of the controls for the locator 3D are cosmetic, dealing with how the locator appears and whether it is rendered in the final output. However, the Camera Settings are critical to getting the results you're looking for.

Size

The Size slider is used to set the size of the Locator's onscreen crosshair.

Color

A basic Color control is used to set the color of the Locator's onscreen crosshair.

Matte

Enabling the Is Matte option applies a special texture to this object, causing this object to not only become invisible to the camera, but also making everything that appears directly behind the camera invisible as well. This option overrides all textures. For more information, *see Chapter 85, "3D Compositing Basics,"* in the DaVinci Resolve Reference Manual, or Chapter 23 in the Fusion Reference Manual.

— **Is Matte:** When activated, objects whose pixels fall behind the matte object's pixels in Z do not get rendered.
— **Opaque Alpha:** Sets the Alpha value of the matte object to 1. This checkbox is visible only when the Is Matte option is enabled.
— **Infinite Z:** Sets the value in the Z-channel to infinity. This checkbox is visible only when the Is Matte option is enabled.

Sub ID

The Sub ID slider can be used to select an individual subelement of certain geometry, such as an individual character produced by a Text 3D node or a specific copy created by a Duplicate 3D node.

Make Renderable

Defines whether the Locator is rendered as a visible object by the OpenGL renderer. The software renderer is not currently capable of rendering lines and hence ignores this option.

Unseen by Camera

This checkbox control appears when the Make Renderable option is selected. If the Unseen by Camera checkbox is selected, the Locator is visible in the viewers but not rendered into the output image by the Renderer 3D node.

Camera

This drop-down control is used to select the Camera in the scene that defines the screen space used for 3D to 2D coordinate transformation.

Use Frame Format Settings

Select this checkbox to override the width, height, and pixel aspect controls, and force them to use the values defined in the composition's Frame Format preferences instead.

Width, Height, and Pixel Aspect

In order for the Locator to generate a correct 2D transformation, it must know the dimensions and aspect of the image. These controls should be set to the same dimensions as the image produced by a renderer associated with the camera specified above. Right-clicking on these controls displays a contextual menu containing the frame formats configured in the composition's preferences.

Common Controls

Transform and Settings tabs

The remaining Transform and Settings tabs are common to many 3D nodes. Their descriptions can be found in "The Common Controls" section at the end of this chapter.

Merge 3D [3Mg]

The Merge 3D node

Merge 3D Introduction

The Merge 3D node is the primary node in Fusion that you use to combine separate 3D elements into the same 3D environment.

For example, in a scene created with an image plane, a camera, and a light, the camera would not be able to see the image plane and the light would not affect the image plane until all three objects are introduced into the same environment using the Merge 3D node.

The Merge provides the standard transformation controls found on most nodes in Fusion's 3D suite. Unlike those nodes, changes made to the translation, rotation, or scale of the Merge affect all the objects connected to the Merge. This behavior forms the basis for all parenting in Fusion's 3D environment.

Inputs

The Merge node displays only two inputs initially, but as each input is connected a new input appears on the node, assuring there is always one free to add a new element into the scene.

— **SceneInput[#]:** These multicolored inputs are used to connect image planes, 3D cameras, lights, entire 3D scenes, as well as other Merge 3D nodes. There is no limit to the number of inputs this node can accept. The node dynamically adds more inputs as needed, ensuring that there is always at least one input available for connection.

Basic Node Setup

The Merge 3D is the hub of a 3D composite. All elements in a 3D scene connect into a Merge 3D.

Multiple Merge 3D nodes can be strung together to control lighting or for neater organization. The last Merge 3D in a string must connect to a Renderer 3D to be output as a 2D image.
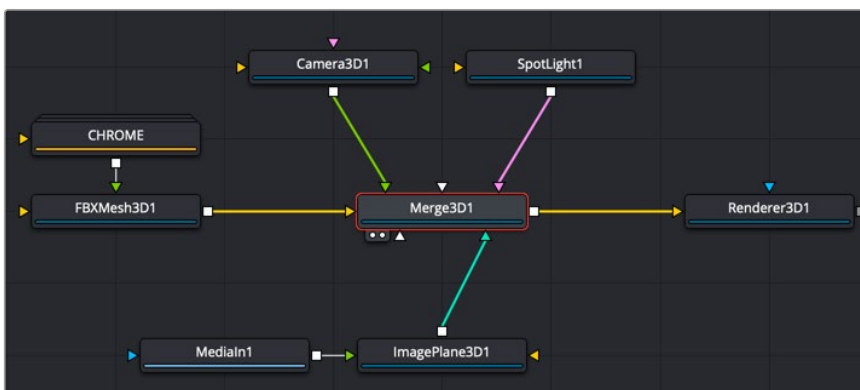
Merge 3D with a connected Image Plane, FBX Mesh object, SpotLight, and camera

Inspector
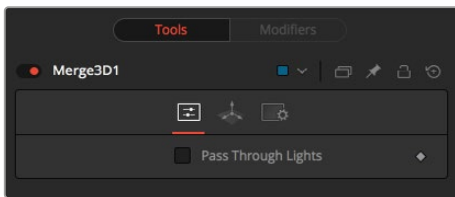
Merge 3D controls

Controls Tab

The Controls tab is used only to pass through any lights connected to the Merge 3D node.

Pass Through Lights

When the Pass Through Lights checkbox is selected, lights are passed through the Merge into its output to affect downstream elements. Normally, the lights are not passed downstream to affect the rest of the scene. This is frequently used to ensure projections are not applied to geometry introduced later in the scene.

Common Controls

Transform and Settings Tabs

The remaining controls for the Transform and Settings tabs are common to most 3D nodes. Their descriptions can be found in "The Common Controls" section at the end of this chapter.

Override 3D [3Ov]
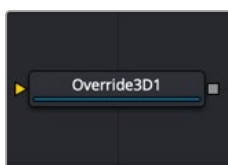
The Override 3D node

Override 3D Node Introduction

The Override node lets you change object-specific options for every object in a 3D scene simultaneously. This is useful, for example, when you wish to set every object in the input scene to render as a wireframe. Additionally, this node is the only way to set the wireframe, visibility, lighting, matte, and ID options for 3D particle systems and the Text 3D node.

Inputs

— **SceneInput:** The orange Scene input accepts the output of a Merge 3D node or any node creating a 3D scene.

Basic Node Setup

The Override 3D node is frequently used in conjunction with the Replace Material node to produce isolated passes. For example, in the node tree below, a scene branches out to an Override node that turns off the Affected by Lights property of each node, then connects to a Replace Material node that applies a Falloff shader to produce a falloff pass of the scene.
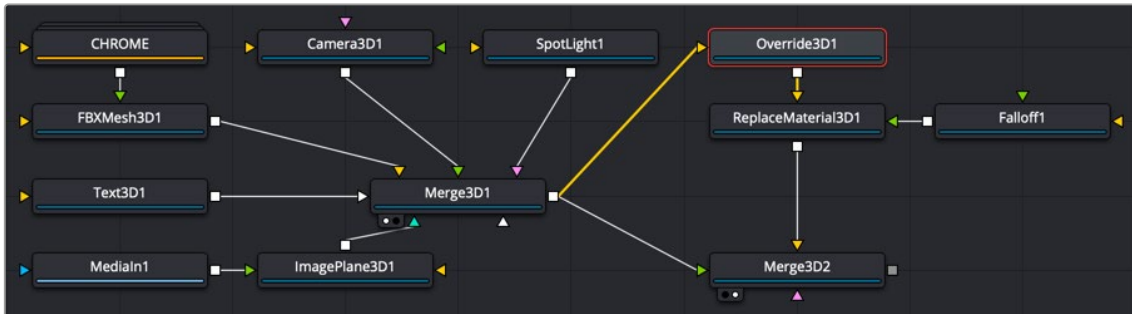
Override 3D connected to a Replace Material node

Inspector
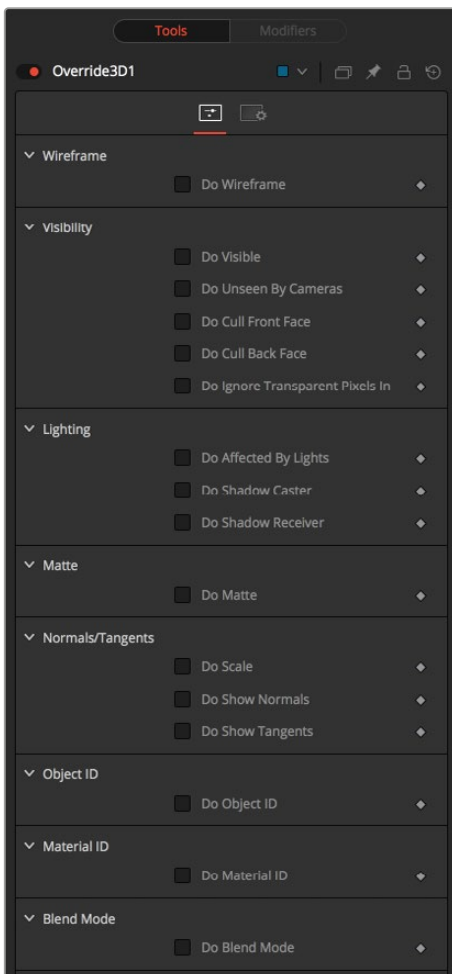
Override 3D controls

Controls Tab

The function of the controls found in the Controls tab is straightforward. First, you select the option to override using the Do [Option] checkbox. That reveals a control that can be used to set the value of the option itself. The individual options are not documented here; a full description of each can be found in any geometry creation node in this chapter, such as the Image Plane, Cube, or Shape nodes.

Do [Option]

Enables the override for this option.

[Option]

If the Do [Option] checkbox is enabled, then the control for the property itself becomes visible. The control values of the properties for all upstream objects are overridden by the new value.

Common Controls

Settings Tabs

The Settings tab includes controls common to most 3D nodes. Their descriptions can be found in "The Common Controls" section at the end of this chapter.

Point Cloud 3D [3PC]
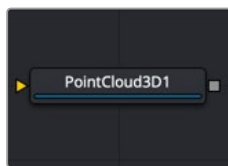
The PointCloud 3D node

Point Cloud 3D Node Introduction

A Point Cloud is generally many null objects created by 3D tracking or modeling software.

When produced by 3D tracking software, the points typically represent each of the patterns tracked to create the 3D camera path. These point clouds can be used to identify a ground plane and to orient other 3D elements with the tracked image. The Point Cloud 3D node creates a point cloud either by importing a file from a 3D tracking application or generating it when you use the Camera Tracker node.

> **NOTE:** A null object is an invisible 3D object that has all the same transform properties of a visible 3D object.

Inputs

The Point Cloud has only a single input for a 3D scene.

— **SceneInput:** This orange input accepts a 3D scene.

Basic Node Setup

The Point Cloud 3D node is viewed and connected through a Merge 3D node, integrating it into the larger 3D scene.
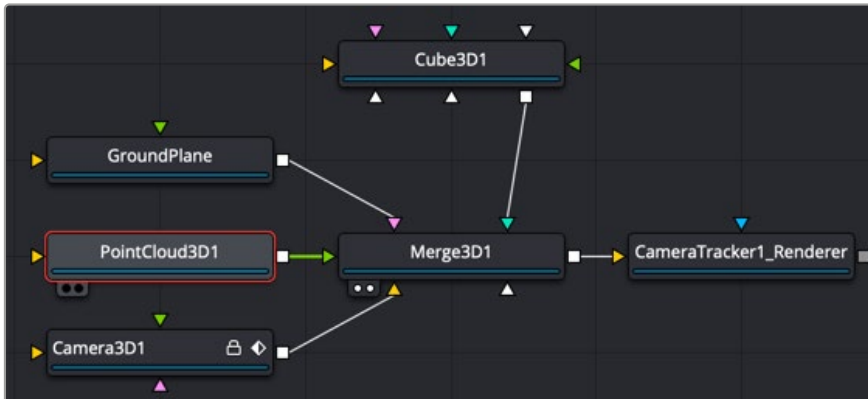
Point Cloud 3D connected and viewed through a Merge 3D

Inspector
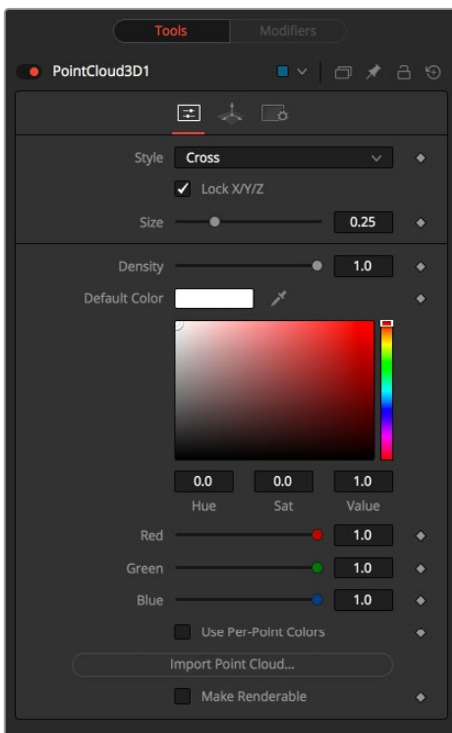
Point Cloud 3D controls

Controls Tab

The Controls tab is where you can import the point cloud from a file and controls its appearance in the viewer.

Style

The Style menu allows you to display the point cloud as cross hairs or points in the viewer.

Lock X/Y/Z

Deselect this checkbox to provide individual control over the size of the X, Y, and Z arms of the points in the cloud.

Size X/Y/Z

These sliders can be used to increase the size of the onscreen crosshairs used to represent each point.

Density

This slider defines the probability of displaying a specific point. If the value is 1, then all points are displayed. A value of 0.2 shows only every fifth point.

Color

Use the standard Color control to set the color of onscreen crosshair controls.

Import Point Cloud

The Import Point Cloud button displays a dialog to import a point cloud from another application.

Supported file types are:

Alias's Maya	`.ma`
3DS Max ASCII Scene Export	`.ase`
NewTek's LightWave	`.lws`
Softimage XSI's	`.xsi.`

Make Renderable

Determines whether the point cloud is visible in the OpenGL viewer and in final renderings made by the OpenGL renderer. The software renderer does not currently support rendering of visible crosshairs for this node.

Unseen by Camera

This checkbox control appears when the Make Renderable option is selected. If the Unseen by Cameras checkbox is selected, the point cloud is visible in the viewers but not rendered into the output image by the Renderer 3D node.

Common Controls

Transform and Settings Tabs

The remaining Transform and Settings tabs are common to many 3D nodes. Their descriptions can be found in "The Common Controls" section at the end of this chapter.

Onscreen Contextual Menu
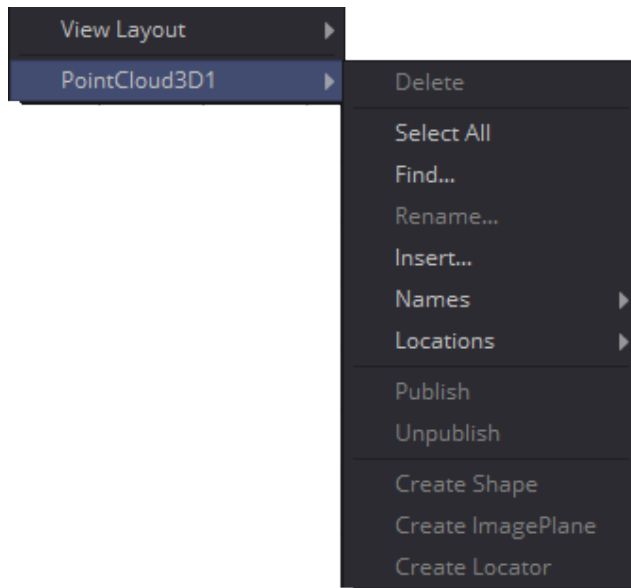
The Point Cloud 3D contextual menu options

Frequently, one or more of the points in an imported point cloud is manually assigned to track the position of a specific feature. These points usually have names that distinguish them from the rest of the points in the cloud. To see the current name for a point, hover the mouse pointer directly over a point, and after a moment a small tooltip appears with the name of the point.

When the Point Cloud 3D node is selected, a submenu is added to the viewer's contextual menu with several options that make it simple to locate, rename, and separate these points from the rest of the point cloud.

The contextual menu contains the following options:

— **Find:** Selecting this option from the viewer contextual menu opens a dialog to search for and select a point by name. Each point that matches the pattern is selected.

— **Rename:** Rename any point by selecting Rename from the contextual menu. Type the new name into the dialog that appears and press Return. The point now has that name, with a four-digit number added to the end. For example, the Name window is window0000, and multiple points would be window0000, window0001, and so on. Names must be valid Fusion identifiers (i.e., no spaces allowed, and the name cannot start with a number).

— **Delete:** Selecting this option deletes the currently selected points.

— **Publish:** Normally, the exact position of a point in the cloud is not exposed. To expose the position, select the points, and then select the Publish option from this contextual menu. This adds a coordinate control to the control panel for each published point that displays the point's current location.

Additional Toolbar and Shortcuts
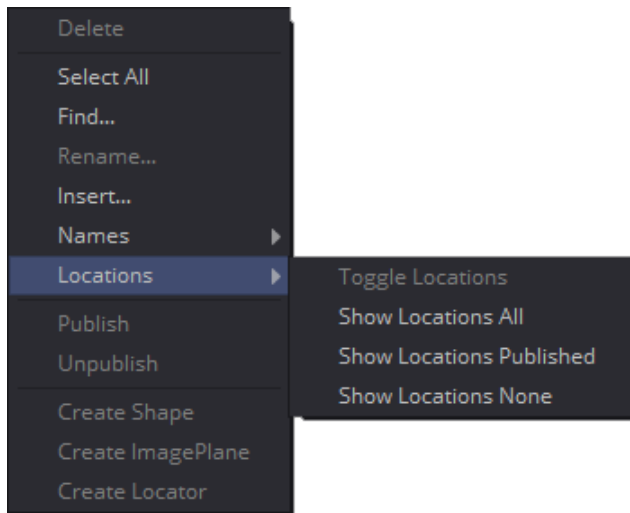
Delete Selected Points	**Del**
Select All	**Shift+A**
Find Points	**Shift+F**
Rename Selected Points	**F2**
Create New Point	**Shift+C**
Toggle Names on None/Selected/Published/All Points	**Shift+N**
Toggle Locations on None/Selected/Published/All Points	**Shift+L**
Publish Selected Points	**Shift+P**
Unpublish Selected Points	**Shift+U**
Create a Shape at Selected Points	**Shift+S**
Create and Fit an ImagePlane to Selected Points	**Shift+I**
Create a Locator at Selected Points	**Shift+O**

Projector 3D [3Pj]
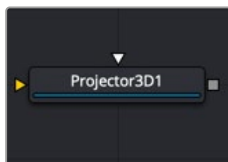
The Projector 3D node

Projector 3D Node Introduction

The Projector 3D node is used to project an image upon 3D geometry. This can be useful in many ways: texturing objects with multiple layers, applying a texture across multiple separate objects, projecting background shots from the camera's viewpoint, image-based rendering techniques,

and more. The Projector node is just one of several nodes capable of projecting images and textures. Each method has advantages and disadvantages. For more information, *see Chapter 85, "3D Compositing Basics,"* in the DaVinci Resolve Reference Manual, or Chapter 23 in the Fusion Reference Manual.

Projected textures can be allowed to "slide" across the object if the object moves relative to the Projector 3D, or, alternatively, by grouping the two with a Merge 3D so they can be moved as one and the texture remains locked to the object.

The Projector 3D node's capabilities and restrictions are best understood if the Projector is considered to be a variant on the SpotLight node. The fact that the Projector 3D node is actually a light has several important consequences when used in Light or Ambient Light projection mode:

— Lighting must be turned on for the results of the projection to be visible.
— The light emitted from the projector is treated as diffuse/specular light. This means that it is affected by the surface normals and can cause specular highlights. If this is undesirable, set the Projector 3D to project into the Ambient Light channel.
— Enabling Shadows causes Projector 3D to cast shadows.
— Just as with other lights, the light emitted by a Projector 3D only affects objects that feed into the first Merge 3D that is downstream of the Projector 3D node in the node tree.
— Enabling Merge 3D's Pass Through Lights checkbox allows the projection to light objects further downstream.
— The light emitted by a Projector 3D is controlled by the Lighting options settings on objects and the Receives Lighting options on materials.
— Alpha values in the projected image do not clip geometry in Light or Ambient Light mode. Use Texture mode instead.
— If two projections overlap, their light contributions are added.

To project re-lightable textures or textures for non-diffuse color channels (like Specular Intensity or Bump), use the Texture projection mode instead:

— Projections in Texture mode only strike objects that use the output of the Catcher node for all or part of the material applied to that object.
— Texture mode projections clip the geometry according to the Alpha channel of the projected image.

See the section for the Catcher node for additional details.

Camera Projection vs. Projection 3D Node

The Camera 3D node also provides a projection feature, and should be used when the projection is meant to match a camera, as this node has more control over aperture, film back, and clip planes. The Projector 3D node was designed to be used as a custom light in 3D scenes for layering and texturing. The projector provides better control over light intensity, color, decay, and shadows.

Inputs

The Projector 3D has two inputs: one for the scene you are projecting on to and another for the projected image.

— **SceneInput:** The orange scene input accepts a 3D scene. If a scene is connected to this input, then transformations applied to the spotlight also affect the rest of the scene.

— **ProjectiveImage:** The white input expects a 2D image to be used for the projection. This connection is required.

Basic Node Setup

As an example, the Projector 3D node below is used to project a texture (MediaIn2) onto 3D primitives as a way to create a simple 3D set. All the set elements are connected into a Merge 3D, which outputs the projected set into a larger scene with camera, lights, and other elements. As an alternative, the Projector 3D node could be inserted between the two Merge 3D nodes; however, the transform controls in the Projector 3D node would then affect the entire scene.
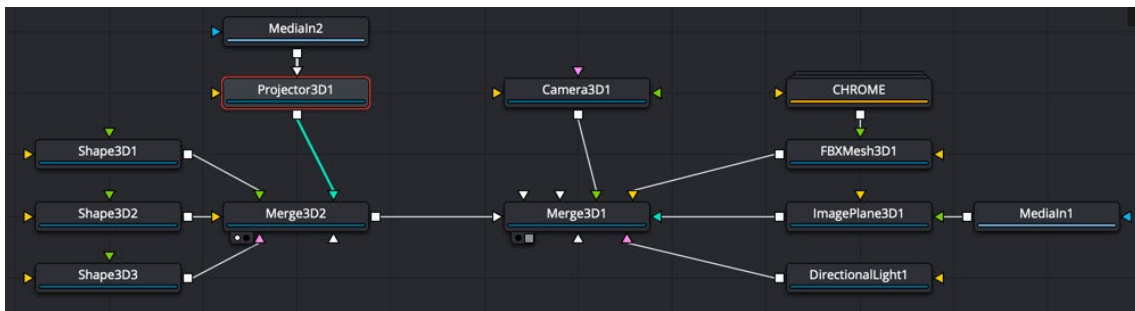
Projector 3D texturing groups of shapes to construct a set

Inspector
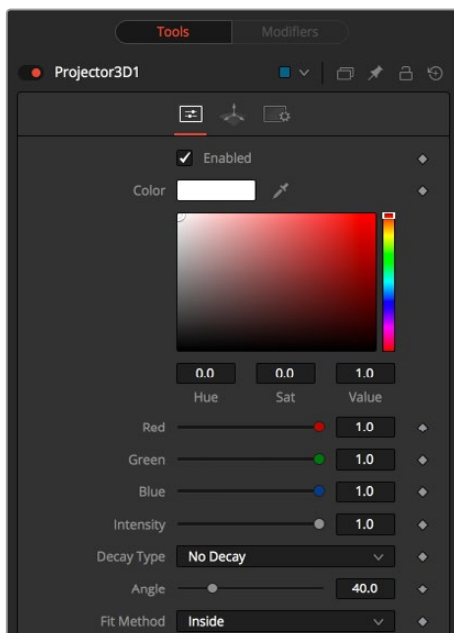
Projector 3D controls

Controls Tab

Enabled

When this checkbox is enabled, the projector affects the scene. Disable the checkbox to turn off the projector. This is not the same as the red switch in the upper-left corner of the Inspector. The red switch disables the tool altogether and passes the image on without any modification. The Enabled checkbox is limited to the effect part of the tool. Other parts, like scripts in the Settings tab still process as normal.

Color

The input image is multiplied by this color before being projected into the scene.

Intensity

Use this slider to set the Intensity of the projection when the Light and Ambient Light projection modes are used. In Texture mode, this option scales the Color values of the texture after multiplication by the color.

Decay Type

A projector defaults to No Falloff, meaning that its light has equal intensity on geometry, despite the distance from the projector to the geometry. To cause the intensity to fall off with distance, set the Decay type to either Linear or Quadratic modes.

Angle

The Cone Angle of the node refers to the width of the cone where the projector emits its full intensity. The larger the angle, the wider the cone angle, up to a limit of 90 degrees.

Fit Method

The Fit Method determines how the projection is fitted within the projection cone.

The first thing to know is that although this documentation may call it a "cone," the Projector 3D and Camera 3D nodes do not project an actual cone; it's more of a pyramid of light with its apex at the camera/projector. The Projector 3D node always projects a square pyramid of light—i.e., its X and Y angles of view are the same. The pyramid of light projected by the Camera 3D node can be non-square depending on what the Film Back is set to in the camera. The aspect of the image connected into the Projector 3D/Camera 3D does not affect the X/Y angles of the pyramid, but rather the image is scaled to fit into the pyramid based upon the fit options.

When both the aspect of the pyramid (AovY/AovX) and the aspect of the image (height * pixelAspectY)/(width * pixelAspectX) are the same, there is no need for the fit options, and in this case the fit options all do the same thing. However, when the aspect of the image and the pyramid (as determined by the Film Back settings in Camera 3D) are different, the fit options become important.

For example, Fit by Width fits the width of the image across the width of the Camera 3D pyramid. In this case, if the image has a greater aspect ratio than the aspect of the pyramid, some of the projection extends vertically outside of the pyramid.

There are five options:

— **Inside:** The image is uniformly scaled so that its largest dimension fits inside the cone. Another way to think about this is that it scales the image as big as possible subject to the restriction that the image is fully contained within the pyramid of the light. This means, for example, that nothing outside the pyramid of light ever receives any projected light.

— **Width:** The image is uniformly scaled so that its width fits inside the cone. Note that the image could still extend outside the cone in its height direction.

— **Height:** The image is uniformly scaled so that its height fits inside the cone. Note that the image could still extend outside the cone in its width direction.

— **Outside:** The image is uniformly scaled so that its smallest dimension fits inside the cone. Another way to think about this is that it scales the image as small as possible subject to the restriction that the image covers the entire pyramid (i.e., the pyramid is fully contained within the image). This means that any pixel of any object inside the pyramid of light always gets illuminated.

— **Stretch:** The image is non-uniformly scaled, so it exactly covers the cone of the projector.

Projection Mode

— **Light:** Projects the texture as a diffuse/specular light.

— **Ambient Light:** Uses an ambient light for the projection.

— **Texture:** When used in conjunction with the Catcher node, this mode allows re-lightable texture projections. The projection strikes only objects that use the catcher material as part of their material shaders.

One useful trick is to connect a Catcher node to the Specular Texture input on a 3D Material node (such as a Blinn). This causes any object using the Blinn material to receive the projection as part of the specular highlight. This technique can be used in any material input that uses texture maps, such as the Specular and Reflection maps.

Shadows

Since the projector is based on a spotlight, it is also capable of casting shadows using shadow maps. The controls under this reveal are used to define the size and behavior of the shadow map.

— **Enable Shadows:** The Enable Shadows checkbox should be selected if the light is to produce shadows. This defaults to selected.

— **Shadow Color:** Use this standard Color control to set the color of the shadow. This defaults to black (0, 0, 0).

— **Density:** The Shadow Density determines the transparency of the shadow. A density of 1.0 produces a completely transparent shadow, whereas lower values make the shadow transparent.

— **Shadow Map Size:** The Shadow Map Size control determines the size of the bitmap used to create the shadow map. Larger values produce more detailed shadow maps at the expense of memory and performance.

— **Shadow Map Proxy:** The Shadow Map Proxy determines the size of the shadow map used for proxy and auto proxy calculations. A value of 0.5 would use a 50% shadow map.

— **Multiplicative/Additive Bias:** Shadows are essentially textures applied to objects in the scene, so there is occasionally Z-fighting, where the portions of the object that should be receiving the shadows render over the top of the shadow instead.

— **Multiplicative and Additive Bias:** Bias works by adding a small depth offset to move the shadow away from the surface it is shadowing, eliminating the Z-fighting. Too little bias and the objects can self-shadow themselves. Too much bias and the shadow can become separated from the surface. Adjust the multiplicative bias first, then fine tune the result using the additive bias control.

— **Force All Materials Non-Transmissive:** Normally, an RGBAZ shadow map is used when rendering shadows. By enabling this option, you are forcing the renderer to use a Z-only shadow map. This can lead to significantly faster shadow rendering while using a fifth as much memory. The disadvantage is that you can no longer cast "stained-glass"-like shadows.

— **Shadow Map Sampling:** Sets the quality for sampling of the shadow map.

— **Softness:** Soft edges in shadows are produced by filtering the shadow map when it is sampled. Fusion provides three separate filtering methods that produce different effects when rendering shadows.

  — **None:** Shadows have a hard edge. No filtering of the shadow map is done at all. The advantage of this method is that you only have to sample one pixel in the shadow map, so it is fast.

  — **Constant:** Shadow edges have a constant softness. A filter with a constant width is used when sampling the shadow map. Adjusting the Constant Softness slider controls the size of the filter. Note that the larger you make the filter, the longer it takes to render the shadows. If the Softness is set to constant, then a Constant slider appears. It can be used to set the overall softness of the shadow.

  — **Variable:** The softness of shadow edges grows the farther away the shadow receiver is from the shadow caster. The variable softness is achieved by changing the size of the filter based on the distance between the receiver and caster. When this option is selected, the Softness Falloff, Min Softness and Max Softness sliders appear.

Softness Falloff	The Softness Falloff slider appears when the Softness is set to variable. This slider controls how fast the softness of shadow edges grows with distance. More precisely, it controls how fast the shadow map filter size grows based on the distance between shadow caster and receiver. Its effect is mediated by the values of the Min and Max Softness sliders.
Min Softness	The Min Softness slider appears when the Softness is set to variable. This slider controls the Minimum Softness of the shadow. The closer the shadow is to the object casting the shadow, the sharper it is up to the limit set by this slider.
Max Softness	The Max Softness slider appears when the Softness is set to variable. This slider controls the Maximum Softness of the shadow. The farther the shadow is from the object casting the shadow, the softer it is up to the limit set by this slider.

Common Controls

Transform and Settings Tabs

The remaining Transform and Settings tabs are common to many 3D nodes. Their descriptions can be found in "The Common Controls" section at the end of this chapter.

Renderer 3D [3Rn]

The Renderer 3D node

Renderer 3D Node Introduction

The Renderer 3D node converts the 3D environment into a 2D image using either a default perspective camera or one of the cameras found in the scene. Every 3D scene in a composition terminates with at least one Renderer 3D node. The Renderer node includes a software and OpenGL render engine to produce the resulting image. Additional render engines may also be available via third-party plugins.

The software render engine uses the system's CPU only to produce the rendered images. It is usually much slower than the OpenGL render engine, but produces consistent results on all machines, making it essential for renders that involve network rendering. The Software mode is required to produce soft shadows, and generally supports all available illumination, texture, and material features.

The OpenGL render engine employs the GPU processor on the graphics card to accelerate the rendering of the 2D images. The output may vary slightly from system to system, depending on the exact graphics card installed. The graphics card driver can also affect the results from the OpenGL renderer. The OpenGL render engines speed makes it possible to provide customized supersampling and realistic 3D depth of field options. The OpenGL renderer cannot generate soft shadows. For soft shadows, the software renderer is recommended.

Like most nodes, the Renderer's motion blur settings can be found under the Common Controls tab. Be aware that scenes containing particle systems require that the Motion Blur settings on the pRender nodes exactly match the settings on the Renderer 3D node.

Otherwise, the subframe renders conflict producing unexpected (and incorrect) results.

> **NOTE:** The Open GL renderer respects the Color Depth option in the Image tab of the Renderer 3D node. This can cause slowdowns on certain graphics cards when rendering to int16 or float32.

Inputs

The Renderer 3D node has two inputs on the node. The main scene input takes in the Merge 3D or other 3D nodes that need to be converted to 2D. The effect mask limits the Renderer 3D output.

— **SceneInput:** The orange scene input is a required input that accepts a 3D scene that you want to convert to 2D.

— **EffectMask:** The blue effects mask input uses a 2D image to mask the output of the node.

Basic Node Setup

All 3D scenes must end with a Renderer 3D node. The Renderer 3D node is used to convert a 3D scene into a 2D image. Below, the Renderer 3D node takes the output of a Merge 3D node, and renders the 3D scene into a 2D image.
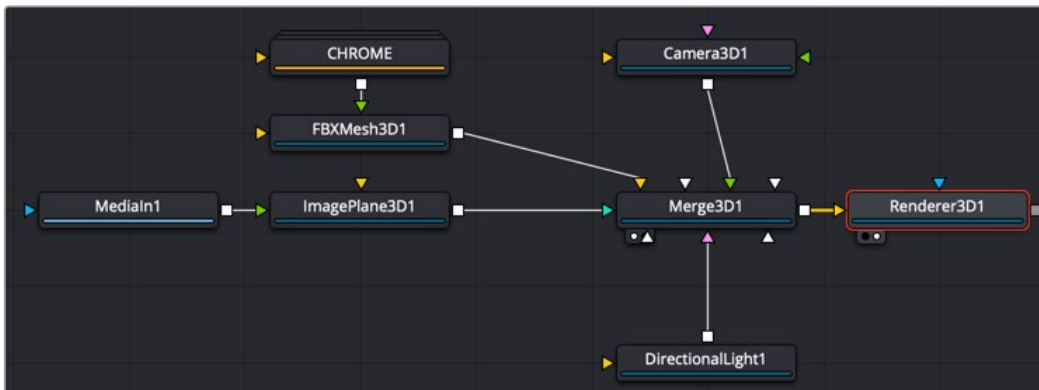
Renderer 3D connected directly after a Merge 3D, rendering the 3D scene to a 2D image

Inspector
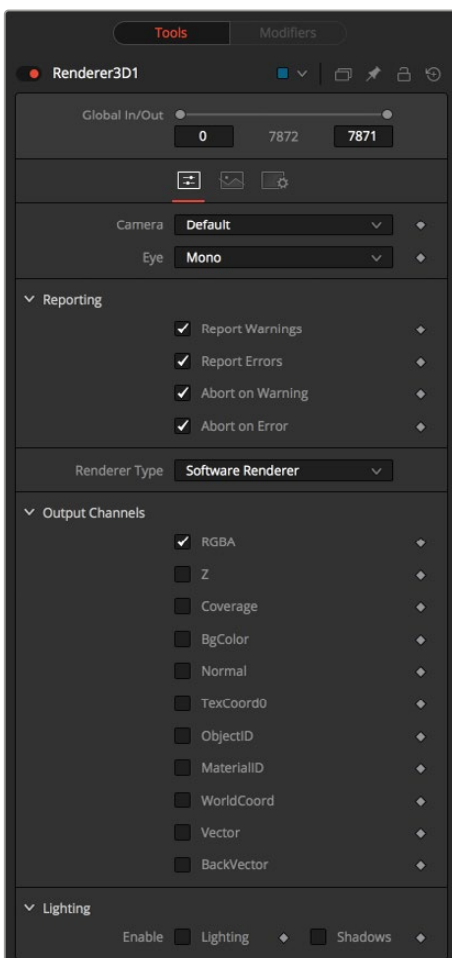
Render 3D controls

Controls Tab

Camera

The Camera menu is used to select which camera from the scene is used when rendering. The Default setting uses the first camera found in the scene. If no camera is located, the default perspective view is used instead.

Eye

The Eye menu is used to configure rendering of stereoscopic projects. The Mono option ignores the stereoscopic settings in the camera. The Left and Right options translate the camera using the stereo Separation and Convergence options defined in the camera to produce either left- or right-eye outputs. The Stacked option places the two images one on top of the other instead of side by side.

Reporting

The first two checkboxes in this section can be used to determine whether the node prints warnings and errors produced while rendering to the console. The second set of checkboxes tells the node whether it should abort rendering when a warning or error is encountered. The default for this node enables all four checkboxes.

Renderer Type

This menu lists the available render engines. Fusion provides three: the software renderer, OpenGL renderer, and the OpenGL UV render engine. Additional renderers can be added via third-party plugins.

All the controls found below this drop-down menu are added by the render engine. They may change depending on the options available to each renderer. So, each renderer is described in its own section below.

Software Controls

Output Channels

Besides the usual Red, Green, Blue, and Alpha channels, the software renderer can also embed the following channels into the image. Enabling additional channels consumes additional memory and processing time, so these should be used only when required.

— **RGBA:** This option tells the renderer to produce the Red, Green, Blue, and Alpha color channels of the image. These channels are required, and they cannot be disabled.

— **Z:** This option enables rendering of the Z-channel. The pixels in the Z-channel contain a value that represents the distance of each pixel from the camera. Note that the Z-channel values cannot include anti-aliasing. In pixels where multiple depths overlap, the frontmost depth value is used for this pixel.

— **Coverage:** This option enables rendering of the Coverage channel. The Coverage channel contains information about which pixels in the Z-buffer provide coverage (are overlapping with other objects). This helps nodes that use the Z-buffer to provide a small degree of anti-aliasing. The value of the pixels in this channel indicates, as a percentage, how much of the pixel is composed of the foreground object.

— **BgColor:** This option enables rendering of the BgColor channel. This channel contains the color values from objects behind the pixels described in the Coverage channel.

— **Normal:** This option enables rendering of the X, Y, and Z Normals channels. These three channels contain pixel values that indicate the orientation (direction) of each pixel in the 3D space. A color channel containing values in a range from [–1,1] represents each axis.

— **TexCoord:** This option enables rendering of the U and V mapping coordinate channels. The pixels in these channels contain the texture coordinates of the pixel. Although texture coordinates are processed internally within the 3D system as three-component UVW, Fusion images store only UV components. These components are mapped into the Red and Green color channel.

— **ObjectID:** This option enables rendering of the ObjectID channel. Each object in the 3D environment can be assigned a numeric identifier when it is created. The pixels in this floating-point image channel contain the values assigned to the objects that produced the pixel. Empty pixels have an ID of 0, and the channel supports values as high as 65534. Multiple objects can share a single Object ID. This buffer is useful for extracting mattes based on the shapes of objects in the scene.

— **MaterialID:** This option enables rendering of the Material ID channel. Each material in the 3D environment can be assigned a numeric identifier when it is created. The pixels in this floating-point image channel contain the values assigned to the materials that produced the pixel. Empty pixels have an ID of 0, and the channel supports values as high as 65534. Multiple materials can share a single Material ID. This buffer is useful for extracting mattes based on a texture; for example, a mask containing all the pixels that comprise a brick texture.

Lighting

— **Enable Lighting:** When the Enable Lighting checkbox is selected, objects are lit by any lights in the scene. If no lights are present, all objects are black.

— **Enable Shadows:** When the Enable Shadows checkbox is selected, the renderer produces shadows, at the cost of some speed.

OpenGL Controls

Render 3D Open GL controls

Output Channels

In addition to the usual Red, Green, Blue, and Alpha channels, the OpenGL render engine can also embed the following channels into the image. Enabling additional channels consumes additional memory and processing time, so these should be used only when required.
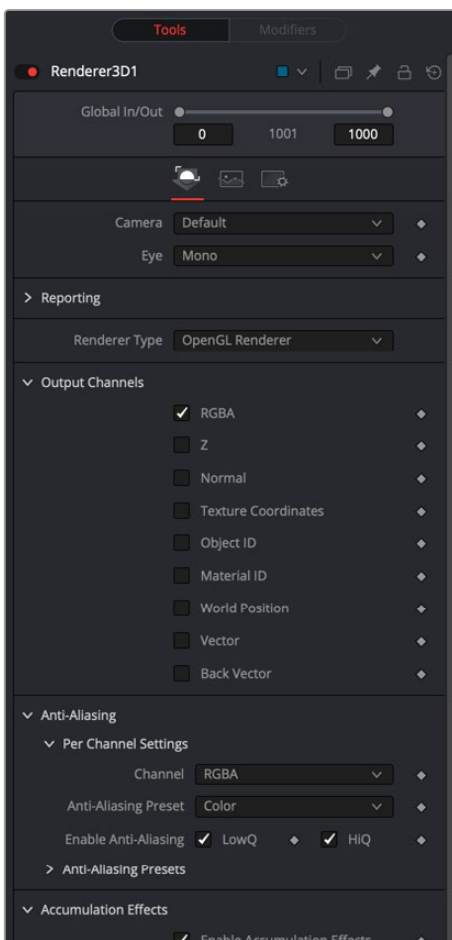
— **RGBA:** This option tells the renderer to produce the Red, Green, Blue, and Alpha color channels of the image. These channels are required, and they cannot be disabled.

— **Z:** This option enables rendering of the Z-channel. The pixels in the Z-channel contain a value that represents the distance of each pixel from the camera. Note that the Z-channel values cannot include anti-aliasing. In pixels where multiple depths overlap, the frontmost depth value is used for this pixel.

— **Normal:** This option enables rendering of the X, Y, and Z Normals channels. These three channels contain pixel values that indicate the orientation (direction) of each pixel in the 3D space. A color channel containing values in a range from [–1,1] is represented by each axis.

— **TexCoord:** This option enables rendering of the U and V mapping coordinate channels. The pixels in these channels contain the texture coordinates of the pixel. Although texture coordinates are processed internally within the 3D system as three-component UVW, Fusion images store only UV components. These components are mapped into the Red and Green color channels.

— **ObjectID:** This option enables rendering of the ObjectID channel. Each object in the 3D environment can be assigned a numeric identifier when it is created. The pixels in this floating-point image channel contain the values assigned to the objects that produced the pixel. Empty pixels have an ID of 0, and the channel supports values as high as 65534. Multiple objects can share a single Object ID. This buffer is useful for extracting mattes based on the shapes of objects in the scene.

— **MaterialID:** This option enables rendering of the Material ID channel. Each material in the 3D environment can be assigned a numeric identifier when it is created. The pixels in this floating-point image channel contain the values assigned to the materials that produced the pixel. Empty pixels have an ID of 0, and the channel supports values as high as 65534. Multiple materials can share a single Material ID. This buffer is useful for extracting mattes based on a texture—for example, a mask containing all the pixels that comprise a brick texture.

Anti–Aliasing

Anti-aliasing can be enabled for each channel through the Channel menu. It produces an output image with higher quality anti-aliasing by brute force, rendering a much larger image, and then rescaling it down to the target resolution. Rendering a larger image in the first place, and then using a Resize node to bring the image to the desired resolution can achieve the exact same results. Using the supersampling built in to the renderer offers two distinct advantages over this method.

The rendering is not restricted by memory or image size limitations. For example, consider the steps to create a float-16 1920 x 1080 image with 16x supersampling. Using the traditional Resize node would require first rendering the image with a resolution of 30720 x 17280, and then using a Resize to scale this image back down to 1920 x 1080. Simply producing the image would require nearly 4 GB of memory. When anti-aliasing is performed on the GPU, the OpenGL renderer can use tile rendering to significantly reduce memory usage.

The GL renderer can perform the rescaling of the image directly on the GPU more quickly than the CPU can manage it. Generally, the more GPU memory the graphics card has, the faster the operation is performed.

Interactively, Fusion skips the anti-aliasing stage unless the HiQ button is selected in the Time Ruler. Final quality renders always include supersampling, if it is enabled.

Because of hardware limitations, point geometry (particles) and lines (locators) are always rendered at their original size, independent of supersampling. This means that these elements are scaled down from their original sizes, and likely appear much thinner than expected.

Anti-Aliasing of Aux Channels in the OpenGL Renderer

The reason Fusion supplies separate anti-aliasing options for color and aux channels in the Anti-Aliasing preset is that supersampling of color channels is quite a bit slower than aux channels. You may find that 1 x 3 LowQ/HiQ Rate is sufficient for color, but for world position or Z, you may require 4 x 12 to get adequate results. The reasons color anti-aliasing is slower are that the shaders for RGBA can be 10x to even 100x or 1000x more complex, and color is rendered with sorting enabled, while aux channels get rendered using the much faster Z-buffer method.

> **TIP:** For some things, sometimes using an SS Z-buffer improves quality, but for other things like using the merge's PerformDepthMerge option, it may make things worse.

Do not mistake anti-aliasing with improved quality. Anti-aliasing an aux channel does not mean it's better quality. In fact, anti-aliasing an aux channel in many cases can make the results much worse. The only aux channels we recommend you enable anti-aliasing on are WorldCoord and Z.

> **TIP:** We strongly recommend disabling Anti-Aliasing on Material ID and Object ID channels, TexCoord, Normal, BackVector, and Vector channels. The issue arises when you have multiple 3D surfaces with radically different TexCoord values in one pixel. The anti-aliasing does not restrict itself to sampling the main surface but samples both surfaces. For example, if one surface has TexCoords that are approximately (u,v) = (0, 0) within that pixel, and the other surface has (0.5, 0.5), you get a blending of these two. The blended area of the texture could have colors like (0, 0) or (0.5, 0.5), resulting in an oddly colored pixel artifact being output from the 2D Texture node. The same problem can happen for normals.

Enable (LowQ/HiQ)

These two check boxes are used to enable anti aliasing of the rendered image.

Supersampling LowQ/HiQ Rate

The LowQ and HiQ rate tells the OpenGL render how large to scale the image. For example, if the rate is set to 4 and the OpenGL renderer is set to output a 1920 x 1080 image, internally a 7680 x 4320 image is rendered and then scaled back to produce the target image. Set the multiplier higher to get better edge anti-aliasing at the expense of render time. Typically 8 x 8 supersampling (64 samples per pixel) is sufficient to reduce most aliasing artifacts.

The rate doesn't exactly define the number of samples done per destination pixel; the width of the reconstruction filter used may also have an impact.

Filter Type

When downsampling the supersized image, the surrounding pixels around a given pixel are often used to give a more realistic result. There are various filters available for combining these pixels. More complex filters can give better results but are usually slower to calculate. The best filter for the job often depends on the amount of scaling and on the contents of the image itself.
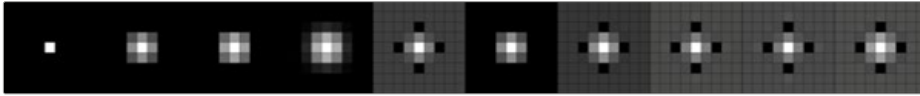
The functions of these filters are shown in the image above. From left to right these are:

Box	This is a simple interpolation scale of the image.
Bi-Linear (triangle)	This uses a simplistic filter, which produces relatively clean and fast results.
Bi-Cubic (quadratic)	This filter produces a nominal result. It offers a good compromise between speed and quality.
Bi-Spline (cubic)	This produces better results with continuous tone images but is slower than Quadratic. If the images have fine detail in them, the results may be blurrier than desired.
Catmul-Rom	This produces good results with continuous tone images which are scaled down, producing sharp results with finely detailed images.
Gaussian	This is very similar in speed and quality to Quadratic.
Mitchell	This is similar to Catmull-Rom but produces better results with finely detailed images. It is slower than Catmull-Rom.
Lanczos	This is very similar to Mitchell and Catmull-Rom but is a little cleaner and also slower.
Sinc	This is an advanced filter that produces very sharp, detailed results, however, it may produce visible `ringing' in some situations.
Bessel	This is similar to the Sinc filter but may be slightly faster.

Window Method

The Window Method menu appears only when the reconstruction filter is set to Sinc or Bessel.

Hanning	This is a simple tapered window.
Hamming	Hamming is a slightly tweaked version of Hanning.
Blackman	A window with a more sharply tapered falloff.

Accumulation Effects

Accumulation effects are used for creating depth of field effects. Enable both the Enable Accumulation Effects and Depth of Field checkboxes, and then adjust the quality and Amount sliders.

The blurrier you want the out-of-focus areas to be, the higher the quality setting you need.
A low amount setting causes more of the scene to be in focus.

The accumulation effects work in conjunction with the Focal plane setting located in the Camera 3D node. Set the Focal Plane to the same distance from the camera as the subject you want to be in focus. Animating the Focal Plane setting creates rack of focus effects.

Lighting

— **Enable Lighting:** When the Enable Lighting checkbox is selected, any lights in the scene light objects. If no lights are present, all objects are black.

— **Enable Shadows:** When the Enable Shadows checkbox is selected, the renderer produces shadows, at the cost of some speed.

Texturing

— **Texture Depth:** Lets you specify the bit depth of texture maps.

— **Warn about unsupported texture depths:** Enables a warning if texture maps are in an unsupported bit depth that Fusion can't process.

Lighting Mode

The Per-vertex lighting model calculates lighting at each vertex of the scene's geometry. This produces a fast approximation of the scene's lighting but tends to produce blocky lighting on poorly tessellated objects. The Per-pixel method uses a different approach that does not rely on the detail in the scene's geometry for lighting, so it generally produces superior results.

Although the per-pixel lighting with the OpenGL renderer produces results closer to that produced by the more accurate software renderer, it still has some disadvantages. The OpenGL renderer is less capable of dealing correctly with semi-transparency, soft shadows, and colored shadows, even with per-pixel lighting. The color depth of the rendering is limited by the capabilities of the graphics card in the system.

Transparency

The OpenGL renderer reveals this control for selecting which ordering method to use when calculating transparency.

— **Z Buffer (fast):** This mode is extremely fast and is adequate for scenes containing only opaque objects. The speed of this mode comes at the cost of accurate sorting; only the objects closest to the camera are certain to be in the correct sort order. So, semi-transparent objects may not be shown correctly, depending on their ordering within the scene.

— **Sorted (accurate):** This mode sorts all objects in the scene (at the expense of speed) before rendering, giving correct transparency.

— **Quick Mode:** This experimental mode is best suited to scenes that almost exclusively contain particles.

Shading Model

Use this menu to select a shading model to use for materials in the scene. Smooth is the shading model employed in the viewers, and Flat produces a simpler and faster shading model.

Wireframe

Renders the whole scene as wireframe. This shows the edges and polygons of the objects. The edges are still shaded by the material of the objects.

Wireframe Anti-Aliasing

Enables anti-aliasing for the Wireframe render.

OpenGL UV Renderer

The OpenGL UV renderer is a special case render engine. It is used to take a model with existing textures and render it out to produce an unwound flattened 2D version of the model. Optionally, lighting can be baked in. This is typically done so you can then paint on the texture and reapply it.
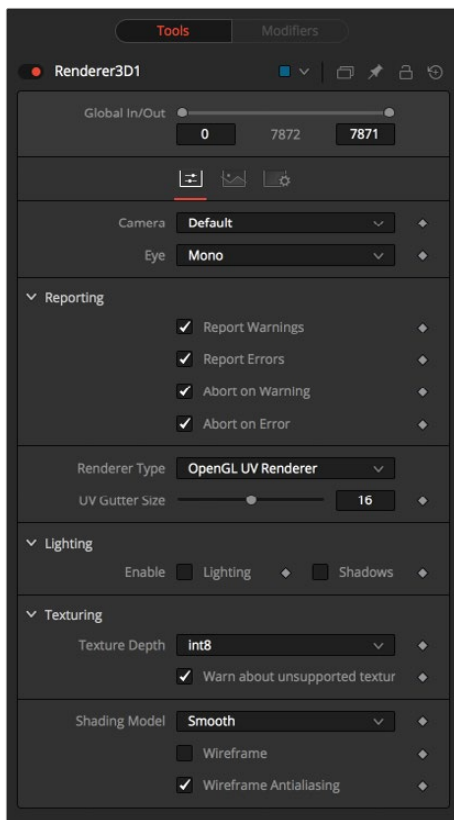
Render 3D Open GL UV controls

Below are some issues to be aware of when using the OpenGL UV renderer.

Baked-in lighting: After you have baked lighting into a model's texture, you need to be careful to turn lighting off on the object later when you render it with the baked-in lighting texture.

Single textures/multiple destinations: Beware of cases where a single area of the texture map is used on multiple areas of the model. This is often done to save texture memory and decrease modeling time. An example is the texture for a person where the artist mirrored the left side mesh/uvs/texture to produce the right side. Trying to bake in lighting in this case won't work.

Unwrapped more the one mesh: Unwrapping more than one mesh at once can cause problems. The reason is that most models are authored so they make maximum usage of (u,v) in [0,1] x [0,1], so that in general models overlap each other in UV space.

Seams: When the UV gutter size is left at 0, this produces seams when the model is retextured with the unwrapped texture.

UV Gutter Size: Increase this value to hide seams between faces.

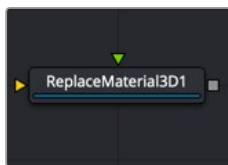**Image and Settings Tabs**

The remaining controls for the Image and Settings tabs are common to many 3D nodes.
Their descriptions can be found in "The Common Controls" section at the end of this chapter.

Replace Material 3D [3Rpl]

The Replace Material 3D node

Replace Material 3D Node Introduction

The Replace Material 3D node replaces the material applied to all the geometry in the input scene with its own material input. Any lights or cameras in the input scene are passed through unaffected.

The scope of the replacement can be limited using Object and Material identifiers in the Inspector. The scope can also be limited to individual channels, making it possible to use a completely different material on the Red channel, for example.

Since the Text 3D node does not include a material input, you can use the Replace Material to add material shaders to the text.

Inputs

The Replace Material node has two inputs: one for the 3D scene, object, or 3D text that contains the original material, and a material input for the new replacement material.

— **SceneInput:** The orange scene input accepts a 3D scene or 3D text that you want to replace the material.

— **MaterialInput:** The green material input accepts either a 2D image or a 3D material. If a 2D image is provided, it is used as a diffuse texture map for the basic material built into the node. If a 3D material is connected, then the basic material is disabled.

Basic Node Setup

The Replace Material 3D node is inserted directly after the 3D object or scene whose material you want to replace. Below, it is used to replace the default material on a Text 3D node with a chrome shader.
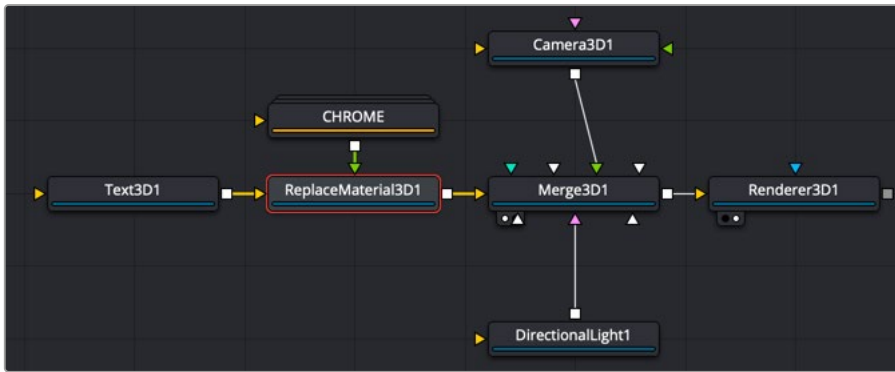
Replace Material 3D used to replace the default material of a Text 3D node

Inspector
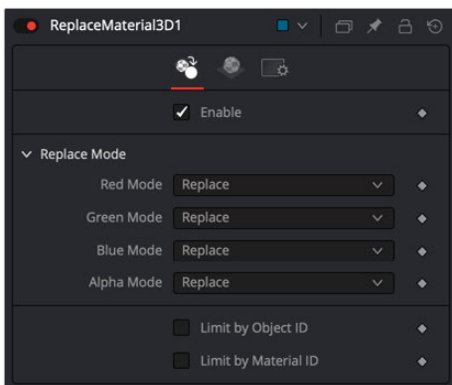
Replace Material 3D controls

Controls Tab

Enable

This checkbox enables the material replacement. This is not the same as the red switch in the upper-left corner of the Inspector. The red switch disables the tool altogether and passes the image on without any modification. The enable checkbox is limited to the effect part of the tool. Other parts, like scripts in the Settings tab, still process as normal.

Replace Mode

The Replace Mode section offers four methods of replacing each RGBA channel:

— **Keep:** Prevents the channel from being replaced by the input material.

— **Replace:** Replaces the material for the corresponding color channel.

— **Blend:** Blends the materials together.

— **Multiply:** Multiplies the channels of both inputs.

Limit by Object ID/Material ID

When enabled, a slider appears where the desired IDs can be set. Other objects keep their materials. If both options are enabled, an object must satisfy both conditions.

Common Controls

Material and Settings Tabs

The remaining controls for the Material and Settings tabs are common to many 3D nodes. Their descriptions can be found in "The Common Controls" section at the end of this chapter.

Replace Normals 3D [3RpN]
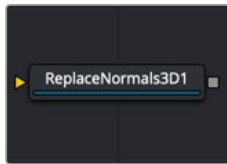
The Replace Normals 3D node

Replace Normals Node Introduction

In 3D modeling, normals are vectors used to determine the direction light reflects off surfaces. The Replace Normals node is used to replace the normals/tangents on incoming geometry, effectively adjusting the surface of an object between smooth and flat. All geometry connected to the scene input on the node is affected. Lights/Cameras/PointClouds/Locators/Materials, and other non-mesh nodes are passed through unaffected. The normals/tangents affected by this node are Per-vertex normals/tangents, not Per-face normals/tangents. The input geometry must have texture coordinates in order for tangents to be computed. Sometimes geometry does not have texture coordinates, or the texture coordinates were set to All by the FBX import because they were not present on the FBX.

Inputs

The Replace Normals node has a single input for the 3D scene or incoming geometry.

— **SceneInput:** The orange scene input accepts a 3D scene or 3D geometry that contains the normal coordinates you want to modify.

Basic Node Setup

The Replace Normals 3D node is inserted directly after the 3D object or scene whose normals you want to modify. Below, it is used to smooth the material on an imported 3D model.
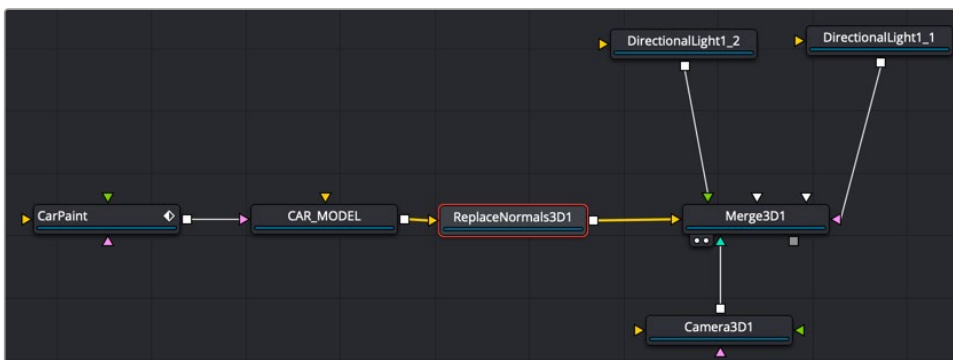
Replace Normals 3D used to smooth normals on 3D geometry

Inspector
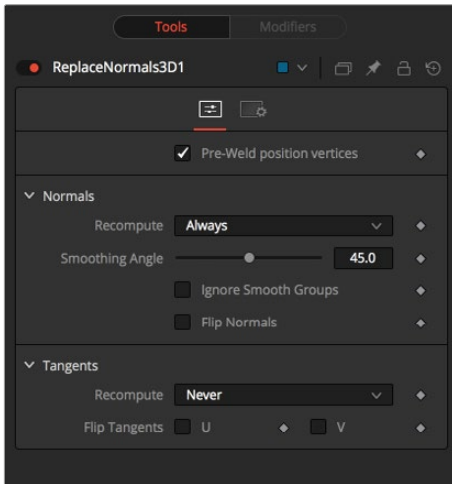
Replace Normals 3D controls

Control Tab

The options in the Control tab deal with repairing 3D geometry and then recomputing normals/tangents.

Pre-Weld Position Vertices

Sometimes position vertices are duplicated in a geometry, even though they have the same position, causing normals/tangents to be miscomputed. The results of pre-welding are thrown away; they do not affect the output geometry's position vertices.

Recompute

Controls when normals/tangents are recomputed.

— **Always:** The normals on the mesh are always recomputed.
— **If Not Present:** The normals on the mesh are recomputed only if they are not present.
— **Never:** The normals are never computed. This option is useful when animating.

Smoothing Angle

Adjacent faces with angles in degrees smaller than this value have their adjoining edges smoothed across. A typical value one might choose for the Smoothing Angle is between 20 and 60 degrees. There is special case code for 0.0f and 360.0f (f stands for floating-point value). When set to 0.0f, faceted normals are produced; this is useful for artistic effect.

Ignore Smooth Groups

If set to False, two faces that have different Smooth Groups are not smoothed across (e.g., the faces of a cube or the top surfaces of a cylinder have different Smooth Groups). If you check this On and set the smoothing angle large enough, the faces of a cube are smoothed across. There is currently no way to visualize Smooth Groups within Fusion.

Flip Normals

Flipping of tangents can sometimes be confusing. Flip has an effect if the mesh has tangent vectors. Most meshes in Fusion don't have tangent vectors until they reach a Renderer 3D, though. Also, when viewing tangent vectors in the viewers, the tangent vectors are created if they don't exist. The confusing thing is if you view a Cube 3D that has no tangent vectors and press the FlipU/FlipV button, nothing happens. This is a result of there being no tangent vectors to create, but later the GL renderer can create some (unflipped) tangent vectors.

There are five items you should be aware of when dealing with normals.

#1 The FBX importer recomputes the normals if they don't exist, but you can get a higher-quality result from the Replace Normals node.

#2 Bump maps can sometimes depend on the model's normals. Specifically, when you simplify a complex high polygon model to a low polygon model + bump map, the normals and bump map can become "linked." Recomputing the normals in this case can make the model look funny. The bump map was intended to be used with the original normals.

#3 Most primitives in Fusion are not generated with tangents; when needed, they are generated on the fly by a Renderer 3D and cached.

#4 Tangents currently are only needed for bump mapping. If a material needs bump mapping, then tangents are created. These tangents are created with some default settings (e.g., Smoothing Angle, and so on). If you don't want Fusion automatically creating tangents, you can use the Replace Normals node to create them manually.

#5 All computations are done in the local coordinates of the geometries instead of in the coordinate system of the Replace Normals 3D node. This can cause problems when there is a non-uniform scale applied to the geometry before Replace Normals 3D is applied.

Common Controls

Settings Tab

The Settings tab is common to many 3D nodes. The description of these controls can be found in "The Common Controls" section at the end of this chapter.

Replicate 3D [3Rep]
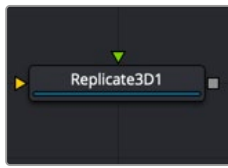
The Replicate 3D node

Replicate 3D Node Introduction

The Replicate 3D node replicates input geometry at positions of destination vertices. The vertices can be mesh vertices as well as particle positions. For each copy of the replicated input geometry, various transformations can be applied. The options in the Jitter tab allow non-uniform transformations, such as random positioning or sizes.

Inputs

There are two inputs on the Replicate 3D node: one for the destination geometry that contains the vertices, and one for the 3D geometry you want to replicate.

— **Destination:** The orange destination input accepts a 3D scene or geometry with vertex positions, either from the mesh or 3D particle animations.
— **Input[#]:** The input accepts the 3D scene or geometry for replicating. Once this input is connected, a new input for alternating 3D geometry is created.

At least one connected input is required.

Basic Node Setup

In the example below, a Replicate 3D node is inserted directly after the pRender node. A spaceship FBX node is connected to the green input representing the object that will be replicated based on the particles. Each particle cell takes on the shape of the 3D geometry connected to the input.
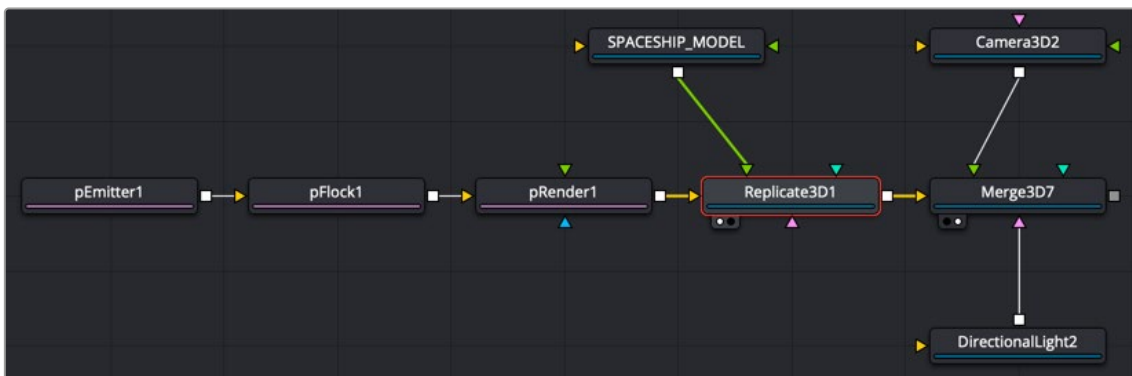
Replicate 3D used to create an armada of swarming spaceships

Inspector
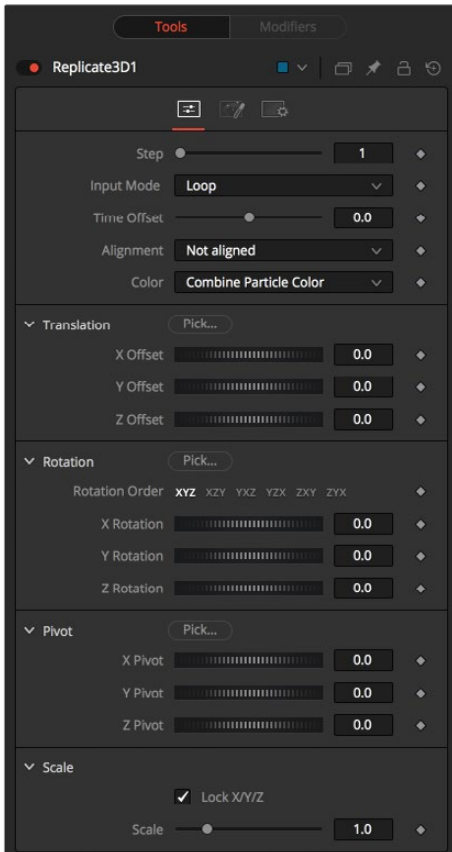
Replicated 3D Jitter controls

Controls Tab

Step

Defines how many positions are skipped. For example, a step of 3 means that only every third vertice of the destination mesh is used, while a step of 1 means that all positions are used.

The Step setting helps to keep reasonable performance for big destination meshes. On parametric geometry like a torus, it can be used to isolate certain parts of the mesh.

Point clouds are internally represented by six points once the Make Renderable option has been set. To get a single point, use a step of 6 and set an X offset of –0.5 to get to the center of the point cloud. Use –0.125 for Locator 3Ds. Once these have been scaled, the offset may differ.

Input Mode

This menu defines in which order multiple input scenes are replicated at the destination. No matter which setting you choose, if only one input scene is supplied this setting has no effect.

— When set to Loop, the inputs are used successively. The first input is at the first position, the second input at the second position, and so on. If there are more positions in the destination present than inputs, the sequence is looped.

— When set to Random, a definite but random input for each position is used based on the seed in the Jitter tab. This input mode can be used to simulate variety with few input scenes.

— The Death of Particles setting causes the input geometries' IDs to change; therefore, their copy order may change.
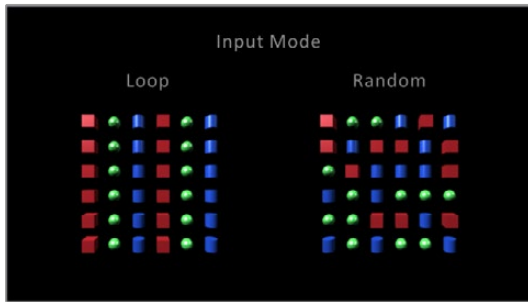
The Input Mode Loop vs. Random order

Time Offset

Use the Time Offset slider to offset any animations that are applied to the input geometry by a set amount per copy. For example, set the value to –1.0 and use a cube set to rotate on the Y-axis as the source. The first copy shows the animation from a frame earlier; the second copy shows animation from a frame before that, etc.

This can be used with great effect on textured planes—for example, where successive frames of a video clip can be shown.

Alignment

Alignment specifies how to align the copies in respect of the destination mesh normal or particle rotation.

— **Not Aligned:** Does not align the copy. It stays rotated in the same direction as its input mesh.
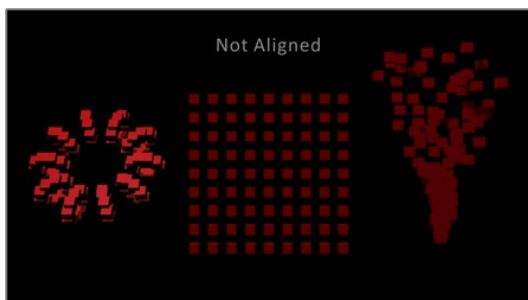
Replicate 3D Not Aligned layout

— **Aligned:** This mode uses the point's normal and tries to reconstruct an upvector. It works best with organic meshes that have unwelded vertices, like imported FBX meshes, since it has the same rotations for vertices at the same positions. On plane geometric meshes, a gradual shift in rotation is noticeable. For best results, it is recommended to use this method at the origin before any transformations.
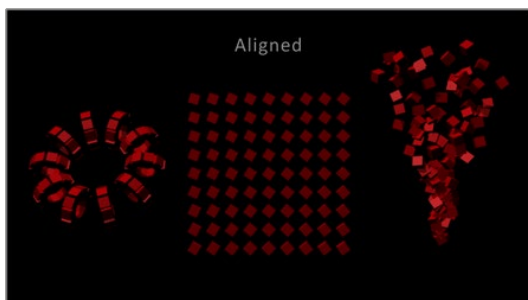
Replicate 3D Aligned layout

— **Aligned TBN:** This mode results in a more accurate and stable alignment based on the tangent, binormal, and normal of the destination point. This works best for particles and geometric shapes. On unwelded meshes, two copies of multiple unwelded points at the same position may lead to different alignments because of their individual normals.
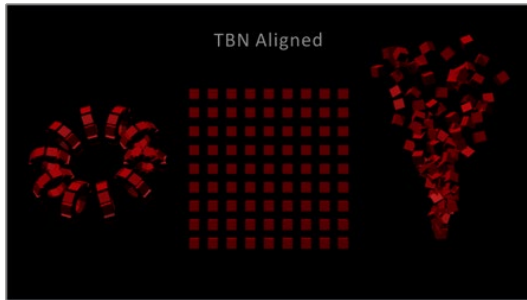
Replicate 3D Aligned TBN layout

Color

Affects the diffuse color or shader of each copy based on the input's particle color.

— **Use Object Color:** Does not use the color of the destination particle.
— **Combine Particle Color:** Uses the shader of any input mesh and modifies the diffuse color to match the color from the destination particle.
— **Use Particle Color:** Replaces the complete shader of any input mesh with a default shader. Its diffuse color is taken from the destination particle.
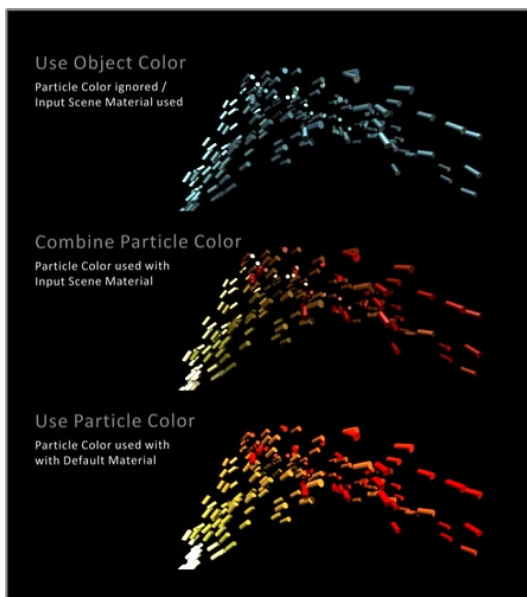
Replicate 3D Color options

Translation

These three sliders tell the node how much offset to apply to each copy. An X Offset of 1 would offset each copy one unit, one unit along the X-axis from the last copy.

Rotation Order

These buttons can be used to set the order in which rotations are applied to the geometry. Setting the rotation order to XYZ would apply the rotation on the X-axis first, followed by the Y-axis rotation, and then the Z-axis rotation.

XYZ Rotation

These three rotation sliders tell the node how much rotation to apply to each copy.

XYZ Pivot

The pivot controls determine the position of the pivot point used when rotating each copy.

Lock XYZ

When the Lock XYZ checkbox is selected, any adjustment to the scale is applied to all three axes simultaneously.

If this checkbox is disabled, the Scale slider is replaced with individual sliders for the X, Y, and Z scales.

Scale

The Scale control sets how much scaling to apply to each copy.
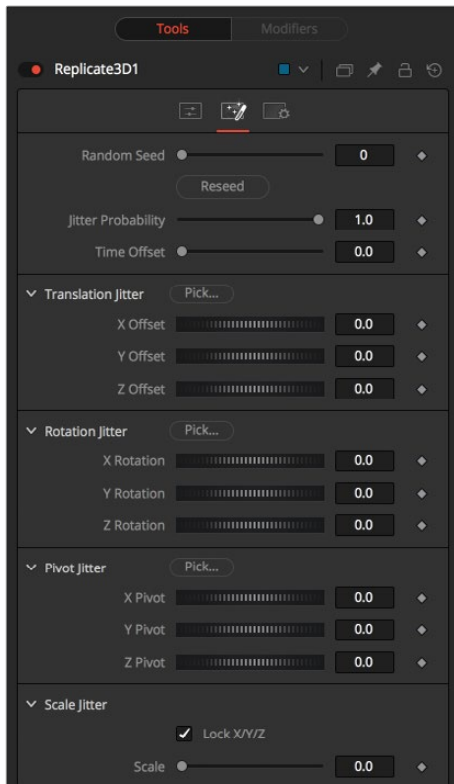
Replicated 3D Jitter controls

Jitter Tab

The Jitter tab can be used to introduce randomness to various parameters.

Random Seed/Randomize

The Random Seed is used to generate the jitter applied to the replicated objects. Two Replicate nodes with identical settings but different random seeds will produce two completely different results. Click the Randomize button to assign a Random Seed value.

Time Offset

Use the Time Offset slider to offset any animations that are applied to the source geometry. Unlike Time Offset on the Controls tab, Jitter Time Offset is random, based on the Random Seed setting.

Translation XYZ Jitter

Use these three controls to adjust the variation in the translation of the replicated objects.

Rotation XYZ Jitter

Use these three controls to adjust the variation in the rotation of the replicated objects.

Pivot XYZ Jitter

Use these three controls to adjust the variation in the rotational pivot center of the replicated objects. This affects only the additional jitter rotation, not the rotation produced by the rotation settings in the Controls tab.

Scale XYZ Jitter

Use this control to adjust the variation in the scale of the replicated objects. Uncheck the Lock XYZ checkbox to adjust the scale variation independently on all three axes.

Common Controls

Settings Tab

The Settings tab is common to many 3D nodes. The description of these controls can be found in "The Common Controls" section at the end of this chapter.

Ribbon 3D [3Ri]
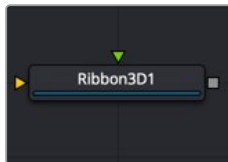
The Ribbon 3D node

Ribbon 3D Node Introduction

Ribbon 3D generates an array of subdivided line segments or a single line between two points. It is quite useful for motion graphics, especially in connection with Replicate 3D to attach other geometry to the lines, and with Displace3D for creating lightning bolt-like structures. The array of lines is, by default, assigned with texture coordinates, so they can be used with a 2D texture. As usual, UVMap 3D can be used to alter the texture coordinates. This node relies heavily on certain OpenGL features and does not produce any visible result when the Renderer 3D node is set to use the software renderer.

Furthermore, the way lines are drawn is completely up to the graphics card capabilities, so the ribbon appearance may vary based on your computer's graphics card.

Inputs

There are two inputs on the Ribbon 3D node: one for the destination geometry that contains the vertices, and one for the 3D geometry you want to replicate.

— **3D Scene:** The orange input accepts a 3D scene or geometry.
— **Material:** The input accepts the 2D texture for the ribbon.

Neither connected input is required.

Basic Node Setup

In the example below, a Ribbon 3D node is used to generate lines. A gradient background is connected to "colorize" the lines. Additional nodes are then used after the Ribbon 3D to bend and distort the lines.
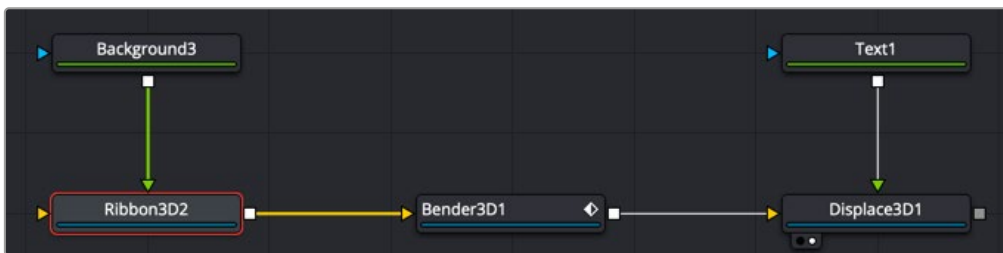
Ribbon 3D generates lines distorted by additional nodes

Inspector
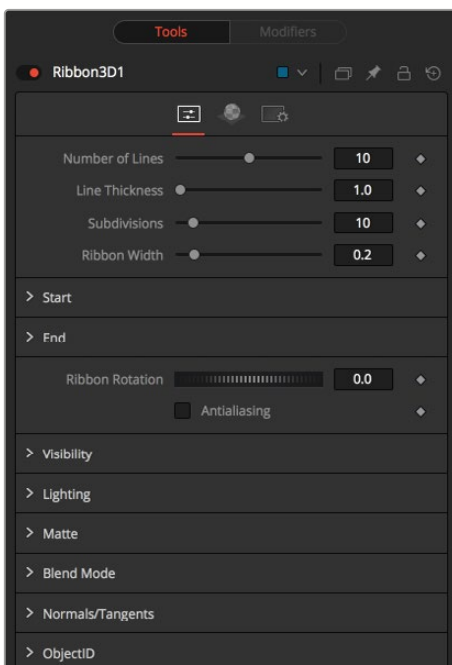
Ribbon 3D controls

Controls Tab

The Controls tab determines the number of ribbon strands, their size, length, and spacing.

Number of Lines

The number of parallel lines drawn between the start point and end point.

Line Thickness

Line thickness is allowed in the user interface to take on a floating-point value, but some graphics cards allow only integer values. Some cards may only allow lines equal to or thicker than one, or max out at a certain value.

Subdivision Level

The number of vertices on each line between start point and end points. The higher the number, the more precise and smoother 3D displacement appears.

Ribbon Width

Determines how far the lines are apart from each other.

Start

XYZ control to set the start point of the ribbon.

End

XYZ control to set the end point of the ribbon.

Ribbon Rotation

Allows rotation of the ribbon around the virtual axis defined by start point and end points.

Anti-Aliasing

Allows you to apply anti-aliasing to the rendered lines. Using anti-aliasing isn't necessarily recommended. When activated, there may be be gaps between the line segments. This is especially noticeable with high values of line thickness. Again, the way lines are drawn is completely up to the graphics card, which means that these artifacts can vary from card to card.

Common Controls

Controls, Materials, and Settings Tabs

The controls for Visibility, Lighting, Matte, Blend Mode, Normals/Tangents, and Object ID in the Controls tab are common in many 3D nodes. The Materials tab and Settings tab in the Inspector are also duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Shape 3D [3Sh]
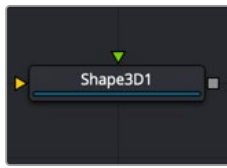
The Shape 3D node

Shape 3D Node Introduction

The Shape 3D node is used to produce several basic primitive 3D shapes, including planes, cubes, spheres, and cylinders.

Inputs

There are two optional inputs on the Shape 3D. The scene input can be used to combine additional geometry with the Shape 3D, while the material input can be used to texture map the Shape 3D object.

— **SceneInput:** Although the Shape 3D creates its own 3D geometry, you can use the orange scene input to combine an additional 3D scene or geometry.

— **MaterialInput:** The green input accepts either a 2D image or a 3D material. If a 2D image is provided, it is used as a diffuse texture map for the basic material built into the node. If a 3D material is connected, then the basic material is disabled.

Basic Node Setup

In the example below, four Shape 3D nodes are used to create the primitives of a 3D set. Two of the Shape 3D nodes are connected creating a more complex primitive shape. Those shapes can then be used with a Projector 3D to texture them with a realistic material.
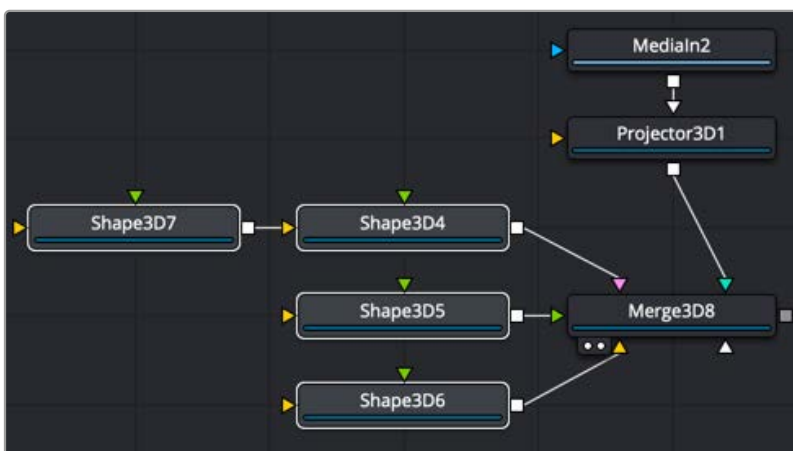
Shape 3D nodes combined with Projector 3D to create a realistic 3D set

Inspector
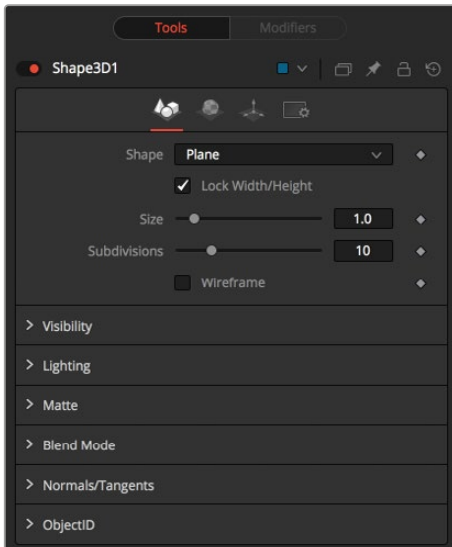
Shape 3D controls

Controls Tab

The Controls tab allows you to select a shape and modify its geometry. Different controls appear based on the specific shape that you choose to create.

Shape

This menu allows you to select the primitive geometry produced by the Shape 3D node. The remaining controls in the Inspector change to match the selected shape.

— **Lock Width/Height/Depth:** [plane, cube] If this checkbox is selected, the width, height, and depth controls are locked together as a single size slider. Otherwise, individual controls over the size of the shape along each axis are provided.

— **Size Width/Height/Depth:** [plane, cube] Used to control the size of the shape.

Cube Mapping

When Cube is selected in the shape menu, the Cube uses cube mapping to apply the Shape node's texture (a 2D image connected to the material input on the node).

Radius

When a Sphere, Cylinder, Cone, or Torus is selected in the shape menu, this control sets the radius of the selected shape.

Top Radius

When a cone is selected in the Shape menu, this control is used to define a radius for the top of a cone, making it possible to create truncated cones.

Start/End Angle

When the Sphere, Cylinder, Cone, or Torus shape is selected in the Shape menu, this range control determines how much of the shape is drawn. A start angle of 180° and end angle of 360° would only draw half of the shape.

Start/End Latitude

When a Sphere or Torus is selected in the Shape menu, this range control is used to crop or slice the object by defining a latitudinal subsection of the object.

Bottom/Top Cap

When Cylinder or Cone is selected in the Shape menu, the Bottom Cap and Top Cap checkboxes are used to determine if the end caps of these shapes are created or if the shape is left open.

Section

When the Torus is selected in the Shape menu, Section controls the thickness of the tube making up the torus.

Subdivision Level/Base/Height

The Subdivision controls are used to determine the tessellation of the mesh on all shapes. The higher the subdivision, the more vertices each shape has.

Wireframe

Enabling this checkbox causes the mesh to render only the wireframe for the object.

Common Controls

Controls, Materials, Transform and Settings Tabs

The controls for Visibility, Lighting, Matte, Blend Mode, Normals/Tangents, and Object ID in the Controls tab are common in many 3D nodes. The Materials tab, Transforms tab, and Settings tab in the Inspector are also duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Sphere Map vs. Connecting the Texture to a Sphere Directly

You can connect a LatLong (equirectangular) texture map directly to a sphere instead of piping it through the Sphere Map node first. This results in a different rendering if you set the start/end angle and latitude to less than 360°/180°. In the first case, the texture is squashed. When using the Sphere Map node, the texture is cropped. Compare:
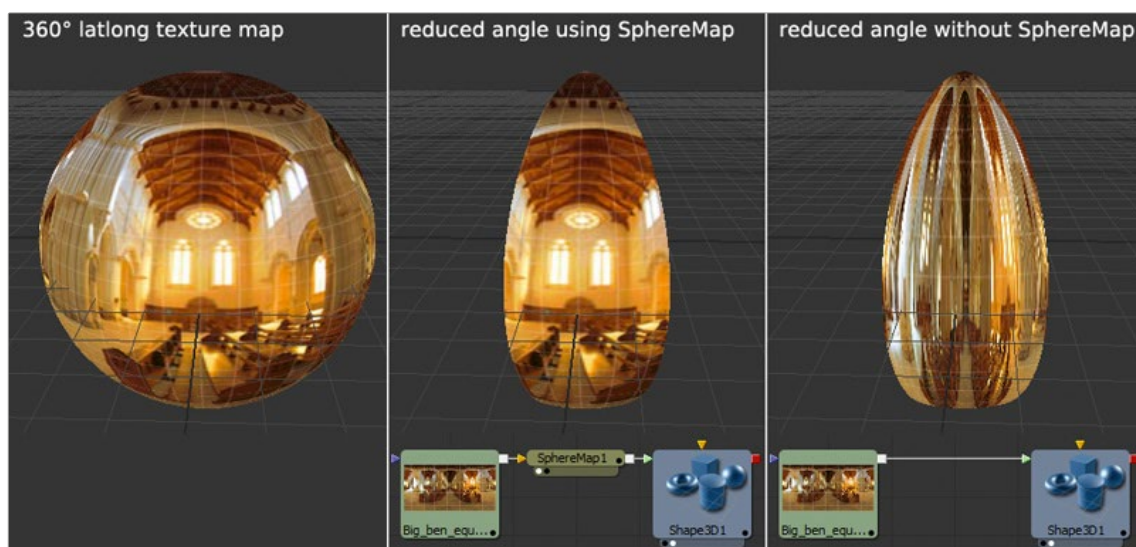
Spherical mapping differences

> **NOTE:** If you pipe the texture directly into the sphere, it is also mirrored horizontally. You can change this by using a Transform node first.

Soft Clip [3SC]
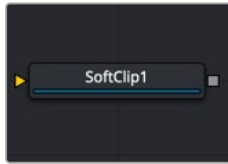
The Soft Clip node

Soft Clip Node Introduction

The Soft Clip node is used to fade out geometry and particles that get close to the camera. This helps avoid the visible "popping off" that affects many particle systems and 3D flythroughs.

This node is very similar to the Fog 3D node, in that it is dependent on the geometry's distance from the camera.

Inputs

The Soft Clip includes only a single input for a 3D scene that includes a camera connected to it.

— **SceneInput:** The orange scene input is a required connection. It accepts a 3D scene input that includes a Camera 3D node.

Basic Node Setup

The Soft Clip node is usually placed just before the Renderer 3D node to ensure that downstream adjustments to lighting and textures do not affect the result. It can be placed in any part of the 3D portion of the node tree if the soft clipping effect is only required for a portion of the scene.
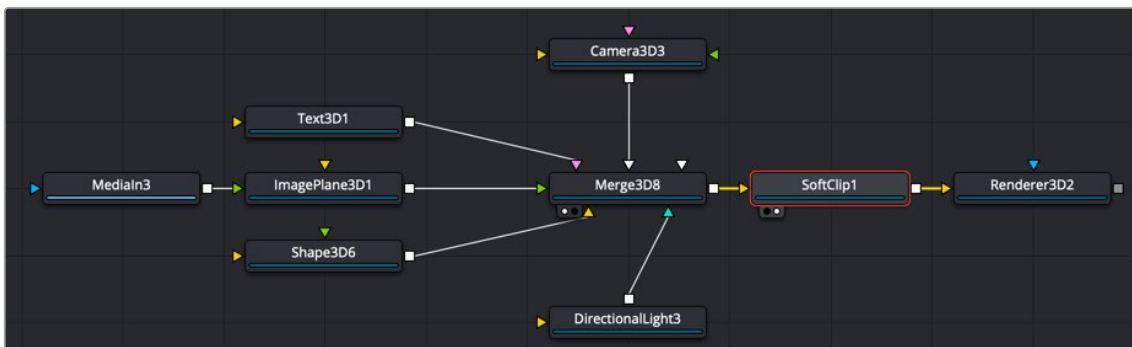
Soft Clip placed between a Merge 3D and a Renderer 3D node

Inspector
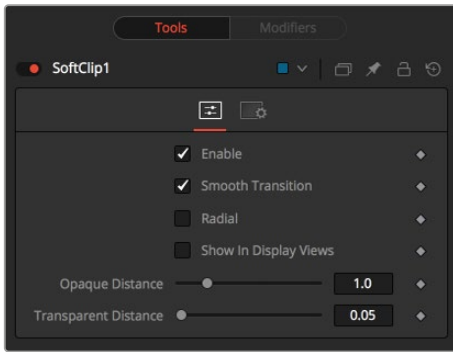
Soft Clip controls

Controls Tab

The Controls tab determines how an object transitions between opaque and transparent as it moves closer to the camera.

Enable

This checkbox can be used to enable or disable the node. This is not the same as the red switch in the upper-left corner of the Inspector. The red switch disables the tool altogether and passes the image on without any modification. The Enable checkbox is limited to the effect of the tool. Other parts, like scripts in the Settings tab, still process as normal.

Smooth Transition

By default, an object coming closer and closer to the camera slowly fades out with a linear progression. With the Smooth Transition checkbox enabled, the transition changes to a nonlinear curve, arguably a more natural-looking transition.

Radial

By default, the soft clipping is done based on the perpendicular distance to a plane (parallel with the near plane) passing through the eye point. When the Radial option is checked, the Radial distance to the eye point is used instead of the Perpendicular distance. The problem with Perpendicular distance soft clipping is that when you move the camera about, as objects on the left or right side of the frustum move into the center, they become less clipped, although they remain the same distance from the eye. Radial soft clip fixes this. Sometimes Radial soft clipping is not desirable.

For example, if you apply soft clip to an object that is close to the camera, like an image plane, the center of the image plane could be unclipped while the edges could be fully clipped because they are farther from the eye point.

Show In Display Views

Normally, the effect is only visible when the scene is viewed using a Camera node. When enabled, the soft clip becomes visible in the scene from all points of view.

Transparent/Opaque Distance

Defines the range of the soft clip. The objects begin to fade in from an opacity of 0 at the Transparent distance and are fully visible at the Opaque distance. All units are expressed as distance from the camera along the Z-axis.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Spherical Camera [3SC]
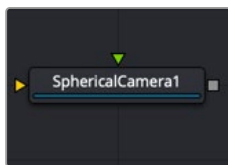
The Spherical Camera node

Spherical Camera Node Introduction

The Spherical Camera allows the 3D Renderer node to output an image covering all viewing angles, laid out in several different formats. This image may be used, for example, as a skybox texture or reflection map or viewed in a VR headset. The Image Width setting in the 3D Renderer sets the size of each square cube face, so the resulting image may be a multiple of this size horizontally and vertically.

Inputs

The Spherical camera node has two inputs.

— **Image:** This orange image input requires an image in a spherical layout, which can be any of LatLong (2:1 equirectangular), Horizontal/Vertical Cross, or Horizontal/Vertical Strip.
— **Stereo Input:** The green input for a right stereo camera if you are working in stereo VR.

Neither input is required.

Basic Node Setup

In many ways, the Spherical Camera is set up identically to the regular Camera 3D node. The output of the camera connects into a Merge 3D. Typically, the Merge 3D has an image from a LatLong or H Cross/V Cross formatted image either directly or through a Panomap node. The image is wrapped around a sphere, and the camera is placed inside the sphere.
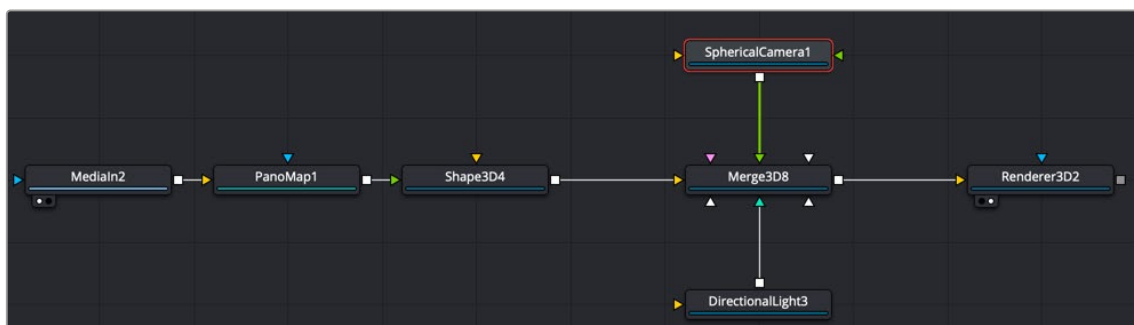
Spherical Camera placed inside a sphere

Inspector
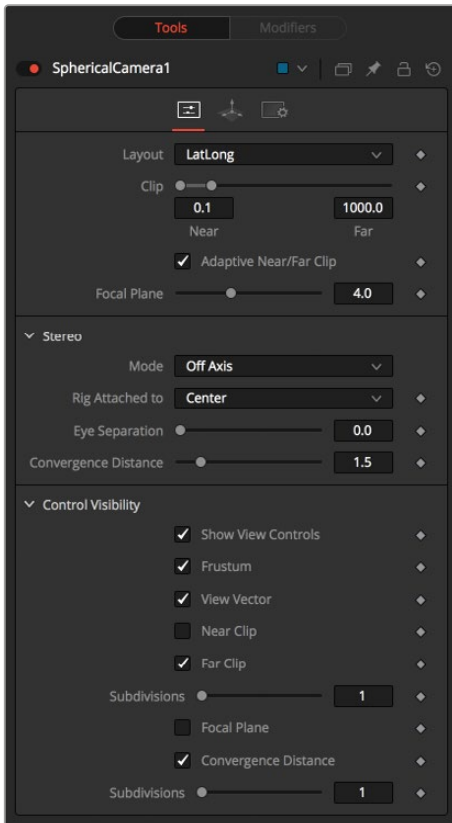
Spherical Camera controls

Controls Tab

Layout

— **VCross and HCross:** VCross and HCross are the six square faces of a cube laid out in a cross, vertical or horizontal, with the forward view in the center of the cross, in a 3:4 or 4:3 image.

— **VStrip and HStrip:** VStrip and HStrip are the six square faces of a cube laid vertically or horizontally in a line, ordered as Left, Right, Up, Down, Back, Front (+X, -X, +Y, -Y, +Z, -Z), in a 1:6 or 6:1 image.

— **LatLong:** LatLong is a single 2:1 image in equirectangular mapping.

Near/Far Clip

The clipping plane is used to limit what geometry in a scene is rendered based on the object's distance from the camera's focal point. This is useful for ensuring that objects that are extremely close to the camera are not rendered and for optimizing a render to exclude objects that are too far away to be useful in the final rendering.

The default perspective camera ignores this setting unless the Adaptively Adjust Near/Far Clip checkbox control below is disabled.

The values are expressed in units, so a far clipping plane of 20 means that any objects more than 20 units from the camera are invisible to the camera. A near clipping plane of 0.1 means that any objects closer than 0.1 units are also invisible.

Adaptively Adjust Near/Far Clip

When selected, the Renderer automatically adjusts the camera's near/far clipping plane to match the extents of the scene. This setting overrides the values of the Near and Far clip range control described above. This option is not available for orthographic cameras.

Viewing Volume Size

The Viewing Volume Size control appears only when the Projection Type is set to Orthographic. It determines the size of the box that makes up the camera's field of view. The Z distance of an orthographic camera from the objects it sees does not affect the scale of those objects, only the viewing size does.

> **NOTE:** A smaller range between the near and far clipping plane allows greater accuracy in all depth calculations. If a scene begins to render strange artifacts on distant objects, try increasing the distance for the near clip plane. Use the vertical aperture size to get the vertical angle of view and the horizontal aperture size to get the horizontal angle of view.

Plane of Focus (for Depth of Field)

This value is used by the OpenGL renderer to calculate depth of field. It defines the distance to a virtual target in front of the camera.

Stereo Method

This control allows you to adjust your stereoscopic method to your preferred working model.

Toe In

Both cameras point at a single focal point. Though the result is stereoscopic, the vertical parallax introduced by this method can cause discomfort by the audience.

Off Axis

Often regarded as the correct way to create stereo pairs, this is the default method in Fusion. Off Axis introduces no vertical parallax, thus creating less stressful stereo images.

Parallel

The cameras are shifted parallel to each other. Since this is a purely parallel shift, there is no Convergence Distance control. Parallel introduces no vertical parallax, thus creating less stressful stereo images.

Eye Separation

Defines the distance between both stereo cameras. If the Eye Separation is set to a value larger than 0, controls for each camera are shown in the viewer when this node is selected. There is no Convergence Distance control in Parallel mode.

Convergence Distance

This control sets the stereoscopic convergence distance, defined as a point located along the Z-axis of the camera that determines where both left and right eye cameras converge.

Control Visibility

Allows you to selectively activate the onscreen controls that are displayed along with the camera.

— **Frustum:** Displays the actual viewing cone of the camera.

— **View Vector:** Displays a white line inside the viewing cone, which can be used to determine the shift when in Parallel mode.

— **Near Clip:** The Near clipping plane. This plane can be subdivided for better visibility.

— **Far Clip:** The Far clipping plane. This plane can be subdivided for better visibility.

— **Plane of Focus:** The camera focal point according to the Plane of Focus slider explained above. This plane can be subdivided for better visibility.

— **Convergence Distance:** The point of convergence when using Stereo mode. This plane can be subdivided for better visibility.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Text 3D [3Txt]
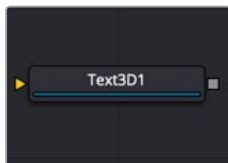
The Text 3D node

Text 3D Node Introduction

The Text3D node is a 3D version of the 2D Text+ node. The controls for this node are mostly identical to the controls for the 2D version in almost all respects, except Text 3D supports only one shading element.

The Text 3D node was based on a tool that predates the Fusion 3D environment. So, some of the controls found in the basic primitive shapes and geometry loaders, such as many of the material, lighting, and matte options, are not found in this node's controls. The Text 3D node has a built-in material, but unlike the other 3D nodes it does not have a material input. The Shading tab contains controls to adjust the diffuse and specular components. To replace this default material with a more advanced material, follow the Text Plus node with a Replace Material 3D node. The Override 3D node can be used to control the lighting, visibility, and matte options for this node.

When network rendering a comp that contains Text 3D nodes, each render machine is required to have the necessary fonts installed or the network rendering fails. Fusion does not share or copy fonts to render slaves.

Inputs

— **SceneInput:** The orange scene input accepts a 3D scene that can be combined with the 3D text created in the node.

— **ColorImage:** The green color image input accepts a 2D image and wraps it around the text as a texture. This input is visible only when Image is selected in the Material Type menu located in the Shading tab.

— **BevelTexture:** The magenta bevel texture input accepts a 2D image and wraps it around the bevel as a texture. This input is visible only when one Material is disabled in the Shader tab and Image is selected in the Bevel Type menu.

Basic Node Setup

The Text 3D node generates text, so most often this node starts a branch of your node tree. However, to apply more realistic materials, a replace Material node is often added after the Text 3D, prior to connecting into a Merge 3D node.
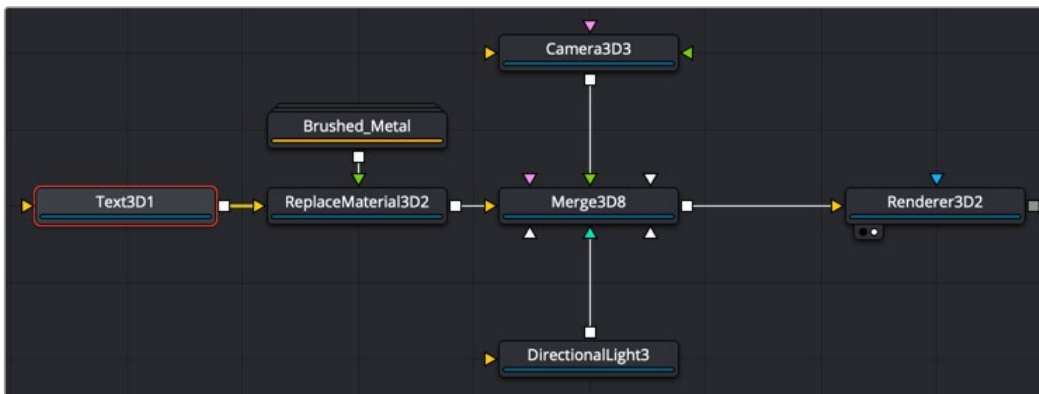
Text 3D with Replace Material node added to apply alternative material

Inspector
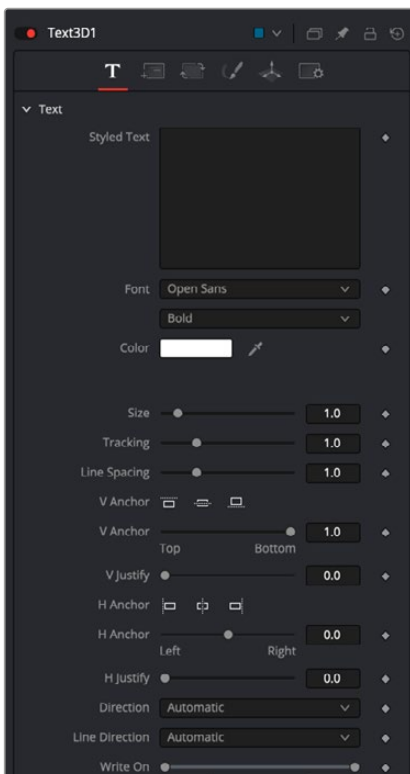
Text 3D controls

Text Tab

The Text 3D text tab in the Inspector is divided into three sections: Text, Extrusion, and Advanced Controls. The Text section includes parameters that are familiar to anyone who has used a word processor. It includes commonly used text formatting options. The Extrusion section includes controls to extrude the text and create beveled edges for the text. The Advanced controls are used for kerning options.

Styled Text

The Edit box in this tab is where the text to be created is entered. Any common character can be typed into this box. The common OS clipboard shortcuts (Command-C or Ctrl-C to copy, Command-X or Ctrl-X to cut, Command-V or Ctrl-V to paste) also work; however, right-clicking on the Edit box displays a custom contextual menu with several modifiers you can add for more animation and formatting options.

Font

Two Font menus are used to select the font family and typeface such as Regular, Bold, and Italic.

Color

This control sets the basic tint color of the text. This is the same Color control displayed in the Material type section of the Shader tab.

Size

This control is used to increase or decrease the size of the text. This is not like selecting a point size in a word processor. The size is relative to the width of the image.

Tracking

The Tracking parameter adjusts the uniform spacing between each character of text.

Line Spacing

Line Spacing adjusts the distance between each line of text. This is sometimes called leading in word-processing applications.

V Anchor

The Vertical Anchor controls consist of three buttons and a slider. The three buttons are used to align the text vertically to the top, middle, or bottom baseline of the text. The slider can be used to customize the alignment. Setting the Vertical Anchor affects how the text is rotated but also the location for line spacing adjustments. This control is most often used when the Layout type is set to Frame in the Layout tab.

V Justify

The Vertical Justify slider allows you to customize the vertical alignment of the text from the V Anchor setting to full justification so it is aligned evenly along the top and bottom edges. This control is most often used when the Layout type is set to Frame in the Layout tab.

H Anchor

The Horizontal Anchor controls consist of three buttons and a slider. The three buttons justify the text alignment to the left edge, middle, or right edge of the text. The slider can be used to customize the justification. Setting the Horizontal Anchor affects how the text is rotated but also the location for tracking (leading) spacing adjustments. This control is most often used when the Layout type is set to Frame in the Layout tab.

H Justify

The Horizontal Justify slider allows you to customize the justification of the text from the H Anchor setting to full justification so it is aligned evenly along the left and right edges. This control is most often used when the Layout type is set to Frame in the Layout tab.

Direction

This menu provides options for determining the direction in which the text is to be written.

Line Direction

These menu options are used to determine the text flow from top to bottom, bottom to top, left to right, or right to left.

Write On

This range control is used to quickly apply simple Write On and Write Off animation to the text.
To create a Write On effect, animate the End portion of the control from 1 to 0 over the length of time required. To create a Write Off effect, animate the Start portion of the range control from 0 to 1.

Extrusion Depth

An extrusion of 0 produces completely 2D text. Any value greater than 0 extrudes the text to generate text with depth.
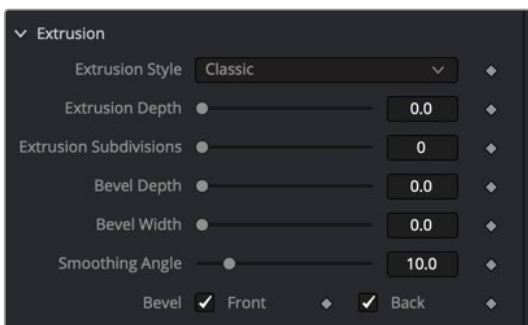
Text 3D extrusion and bevel controls

Bevel Depth

Increase the value of the Bevel Depth slider to bevel the text. The text must have extrusion before this control has any effect.

Bevel Width

Use the Bevel Width control to increase the width of the bevel.

Smoothing Angle

Use this control to adjust the smoothing angle applied to the edges of the bevel.

Front/Back Bevel

Use these checkboxes to enable beveling for the front and back faces of the text separately

Custom Extrusion

In Custom mode, the Smoothing Angle controls the smoothing of normals around the edges of a text character. The spline itself controls the smoothing along the extrusion profile. If a spline segment is

smoothed, for example by using the shortcut Shift-S, the normals are smoothed as well. If the control point is linear, the shading edge is sharp. The first and last control points on the spline define the extent of the text.

Custom Extrusion Subdivisions

Controls the number of subdivisions within the smoothed portions of the extrusion profile.

> **TIP:** Splines can also be edited from within the Spline Editor panel. It provides a larger working space for working with any spline including the Custom Extrusion.
>
> **Extrusion profile spline control:** Do not try to go to zero size at the Front/Back face. This results in Z-fighting resulting from self-intersecting faces. To avoid this problem, make sure the first and last point have their profiles set to 0.

Force Monospaced

This slider control can be used to override the kerning (spacing between characters) that is defined in the font. Setting this slider to zero (the default value) causes Fusion to rely entirely on the kerning defined with each character. A value of one causes the spacing between characters to be completely even, or monospaced.
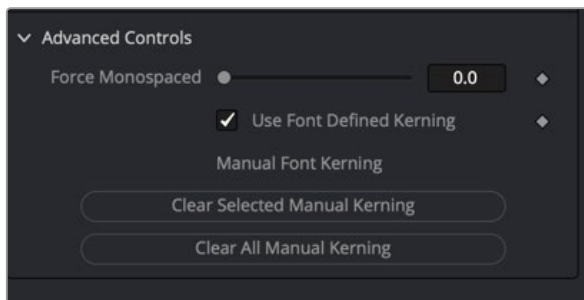
Text 3D Advanced Controls can be
used to manually kern letters.

Use Font Defined Kerning

This enables kerning as specified in the True Type font and is on by default.

Manual Font Kerning

Manual Font Kerning is only performed using the Text+ node. To perform manual kerning on Text3D, create the text using the Text+ node and kern it in that tool. Then, right-click over the tool's name in the Inspector and choose Copy. Once the settings are copied, select the Text 3D node and choose Paste Settings from the Inspector's contextual menu. Once the manual kerning is pasted in the Text 3D node, the two buttons in the Inspector clear either the selected character's kerning or all the kerning adjustment in the current text.

Layout Tab

The Layout Tab is used to position the text in one of four different layout types.
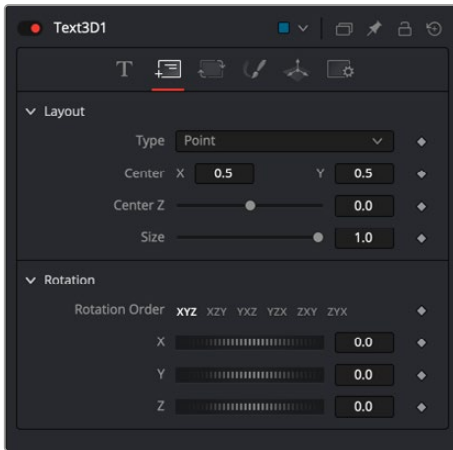
Text 3D Layout tab for changing
the layout of the text block

Layout Type

This menu selects the layout type for the text.

— **Point:** Point layout is the simplest of the layout modes. Text is arranged around an adjustable center point.

— **Frame:** Frame layout allows you to define a rectangular frame used to align the text. The alignment controls are used to justify the text vertically and horizontally within the boundaries of the frame.

— **Circle:** Circle layout places the text around the curve of a circle or oval. Control is offered over the diameter and width of the circular shape. When the layout is set to this mode, the Alignment controls determine whether the text is positioned along the inside or outside of the circle's edge, and how multiple lines of text are justified.

— **Path:** Path layout allows you to shape your text along the edges of a path. The path can be used simply to add style to the text, or it can be animated using the Position on Path control that appears when this mode is selected.

Center X, Y, and Z

These controls are used to position the center of the layout. For instance, moving the center X, Y, and Z parameters when the layout is set to Frame moves the position of the frame the text is within.

Size

This slider is used to control the scale of the layout element. For instance, increasing size when the layout is set to Frame increases the frame size the text is within.

Width and Height

The Width and Height controls are visible when the Layout mode is set to Circle or Frame. The Width and Height controls are visible only when the Layout mode is set to Frame. They are used to adjust the dimensions and aspect of the Layout element.

Rotation Order

These buttons allow you to select the order in which 3D rotations are applied to the text.

X, Y, and Z

These angle controls can be used to adjust the angle of the Layout element along any axis.

Fit Characters

This menu control is visible only when the Layout type is set to Circle. This menu is used to select how the characters are spaced to fit along the circumference.

Position on Path

The Position on Path control is used to control the position of the text along the path. Values less than zero or greater than one cause the text to move beyond, continuing in the same direction set by the last two points on the path.

Right-Click Here for Shape Animation

This label appears only when the Layout type is set to Path. It is used to provide access to a contextual menu that provides options for connecting the path to other paths in the node tree, and animating the spline points on the path over time.

Transform Tab

There are actually two Transform tabs in the Text 3D Inspector. The first Transform tab is unique to the Text 3D tool, while the second is the common Transform tab found on many 3D nodes. The Text 3D-specific Transform tab is described below since it contains some unique controls for this node.
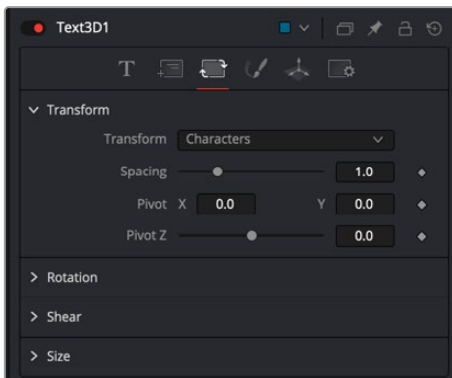
Text 3D Transform tab

Transform

This menu determines the portion of the text affected by the transformations applied in this tab. Transformations can be applied to line, word, and character levels simultaneously. This menu is only used to keep the number of visible controls to a reasonable number.

— **Characters:** Each character of text is transformed along its own center axis.
— **Words:** Each word is transformed separately on the word's center axis.
— **Lines:** Each line of the text is transformed separately on that line's center axis.

Spacing

The Spacing slider is used to adjust the amount of space between each line, word, or character. Values less than one usually cause the characters to begin overlapping.

Pivot X, Y, and Z

This provides control over the exact position of the axis. By default, the axis is positioned at the calculated center of the line, word, or character. The pivot control works as an offset, such that a value of 0.1, 0.1 in this control would cause the axis to be shifted downward and to the right for each of the text elements. Positive values in the Z-axis slider move the axis further along the axis (away from the viewer). Negative values bring the axis of rotation closer.

Rotation Order

These buttons are used to determine the order in which transforms are applied. X, Y, and Z would mean that the rotation is applied to X, then Y, and then Z.

X, Y, and Z

These controls can be used to adjust the angle of the text elements in any of the three dimensions.

Shear X and Y

Adjust these sliders to modify the slanting of the text elements along the X- and Y-axis.

Size X and Y

Adjust these sliders to modify the size of the text elements along the X- and Y-axis.

Shading

The Shading tab for the Text 3D node controls the overall appearance of the text and how lights affect its surface.
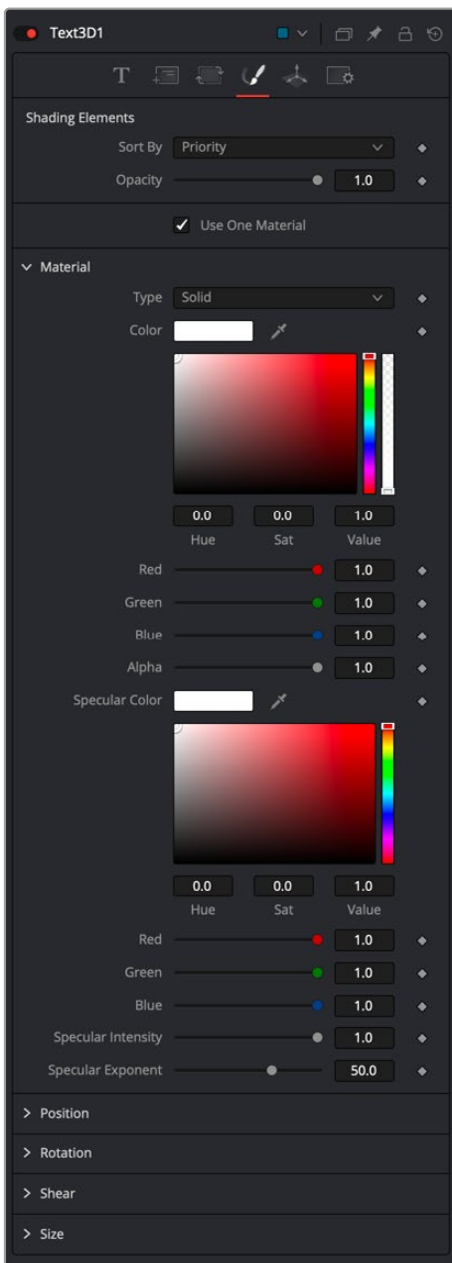
Text 3D Shading tab

Opacity

Reducing the material's opacity decreases the color and Alpha values of the specular and diffuse colors equally, making the material transparent and allowing hidden objects to be seen through the material.

Use One Material

Deselecting this option reveals a second set of Material controls for the beveled edge of the text.

Type

To use a solid color texture, select the Solid mode. Selecting the Image mode reveals a new external input on the node that can be connected to another 2D image.

Specular Color

Specular Color determines the color of light that reflects from a shiny surface. The more specular a material is, the glossier it appears. Surfaces like plastics and glass tend to have white specular highlights, whereas metallic surfaces like gold have specular highlights that tend to inherit their color from the material color. The basic shader material does not provide an input for textures to control the specularity of the object. Use nodes from the 3D Material category when more precise control is required over the specular appearance.

Specular Intensity

Specular Intensity controls the strength of the specular highlight. If the specular intensity texture port has a valid input, then this value is multiplied by the Alpha value of the input.

Specular Exponent

Specular Exponent controls the falloff of the specular highlight. The greater the value, the sharper the falloff, and the smoother and glossier the material appears. The basic shader material does not provide an input for textures to control the specular exponent of the object. Use nodes from the 3D Material category when more precise control is required over the specular exponent.

Image Source

This control determines the source of the texture applied to the material. If the option is set to Tool, then an input appears on the node that can be used to apply the output of a 2D node as the texture. Selecting Clip opens a file browser that can be used to select an image or image sequence from disk. The Brush option provides a list of clips found in the Fusion\brushes folder.

Bevel Material

This option appears only when the Use One Material checkbox control is selected. The controls under this option are an exact copy of the Material controls above but are applied only to the beveled edge of the text.

Position, Rotation, Shear, and Size

These transform controls act similarly to the transform controls in the Transform tab when a single shading element is enabled from the top of the Shading tab. However, when two or more shading elements are enabled, these transform controls are applied to the currently selected shading element. This allows you to independently control the position, rotation, shearing, and size of borders, fill colors, and shadows.

> **Uncapped 3D Text**   To hide the front face of extruded text, uncheck Use One Material on the Shading tab and reduce the first material's color to black, including its Alpha value.

Common Controls

Transform and Settings Tabs

The Transform and Settings tabs in the Inspector are duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Text 3D Modifiers

Right-clicking within the Styled Text box displays a menu with the following text modifiers. Only one modifier can be applied to a Text 3D Styled Text box. Below is a brief list of the text specific modifiers, but for more information *see Chapter 122, "Modifiers,"* in the DaVinci Resolve Reference Manual or Chapter 60 in the Fusion Reference Manual.

Animate

Use the Animate command to set to a keyframe on the entered text and animate the content over time.

Character Level Styling

The Text 3D node doesn't support Character Level Styling directly. However, you can create a Text+ node first and modify its text field with a Character Level Styling modifier. Then either connect the Text 3D's text field to the modifier that is now available or copy the Text+ node and paste its settings to the Text 3D node (right-click > Paste Settings)

Comp Name

Comp Name puts the name of the composition in the Styled Text box and is generally used as a quick way to create slates.

Follower

Follower is a text modifier that can be used to ripple animation applied to the text across each character in the text. See "Text Modifiers" at the end of this chapter.

Publish

Publish the text for connection to other text nodes.

Text Scramble

A text modifier ID is used to randomize the characters in the text. See "Text Modifiers" at the end of this chapter.

Text Timer

A text modifier is used to count down from a specified time or to output the current date and time. See "Text Modifiers" at the end of this chapter.

Time Code

A text modifier is used to output Time Code for the current frame. See "Text Modifiers" at the end of this chapter.

Connect To

Use this option to connect the text generated by this Text node to the published output of another node.

Transform 3D [3Xf]
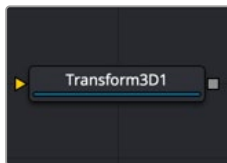
The Transform 3D node

Transform 3D Node Introduction

The Transform 3D node can be used to translate, rotate, or scale all the elements within a scene without requiring a Merge 3D node. This can be useful for hierarchical transformations or for offsetting objects that are merged into a scene multiple times. Its controls are identical to those found in other 3D nodes' Transformation tabs.

Inputs

The Transform node has a single required input for a 3D scene or 3D object.

— **Scene Input:** The orange scene input is connected to a 3D scene or 3D object to apply a second set of transformation controls.

Basic Node Setup

The Transform 3D node adds 3D position, rotation, and pivot control onto any existing transforms in the 3D node prior to it. You can combine multiple Transform 3D nodes together to build parenting or hierarchical movement.
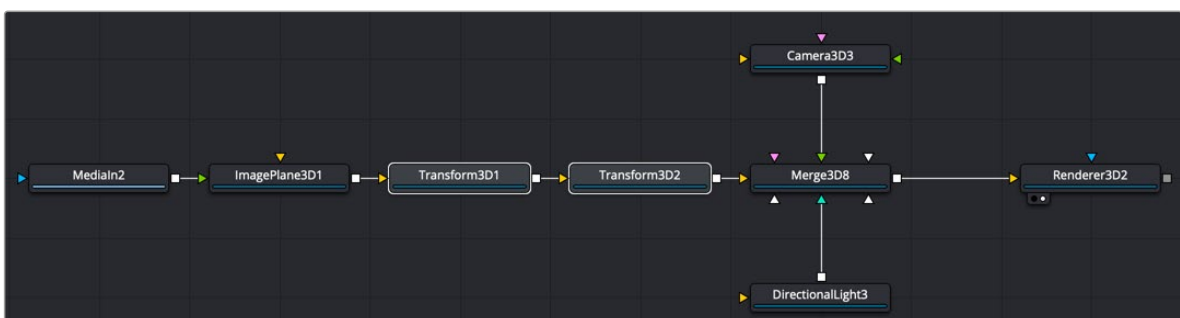
Transform 3D nodes strung together to create a parenting hierarchy

Inspector
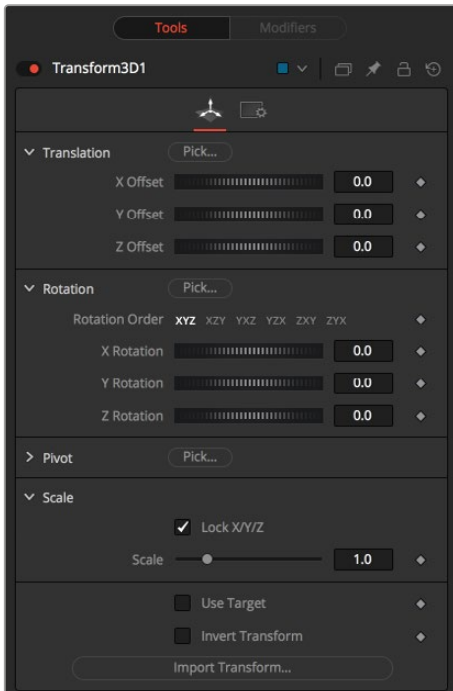
Transform 3D controls

Controls Tab

The Controls tab is the primary tab for the Transform 3D node. It includes controls to translate, rotate, or scale all elements within a scene without requiring a Merge 3D node.

Translation

— **X, Y, Z Offset:** Controls are used to position the 3D element in 3D space.

Rotation

— **Rotation Order:** Use these buttons to select the order used to apply the rotation along each axis of the object. For example, XYZ would apply the rotation to the X-axis first, followed by the Y-axis, and then the Z-axis.

— **X, Y, Z Rotation:** Use these controls to rotate the object around its pivot point. If the Use Target checkbox is selected, then the rotation is relative to the position of the target; otherwise, the global axis is used.

Pivot Controls

— **X, Y, Z Pivot:** A pivot point is the point around which an object rotates. Normally, an object rotates around its own center, which is considered to be a pivot of 0,0,0. These controls can be used to offset the pivot from the center.

Scale

— **X, Y, Z Scale:** If the Lock X/Y/Z checkbox is checked, a single scale slider is shown. This adjusts the overall size of the object. If the Lock checkbox is unchecked, individual X, Y, and Z sliders are displayed to allow scaling in any dimension.

> **NOTE:** If the Lock checkbox is checked, scaling of individual dimensions is not possible, even when dragging specific axes of the Transformation widget in Scale mode.

Use Target

Selecting the Use Target checkbox enables a set of controls for positioning an XYZ target. When Use Target is enabled, the object always rotates to face the target. The rotation of the object becomes relative to the target.

Import Transform

Opens a file browser where you can select a scene file saved or exported by your 3D application. It supports the following file types:

LightWave Scene	`.lws`
Max Scene	`.ase`
Maya Ascii Scene	`.ma`
dotXSI	`.xsi`

The Import Transform button imports only transformation data. For 3D geometry, lights and cameras, consider using the File > FBX Import option from the menus.
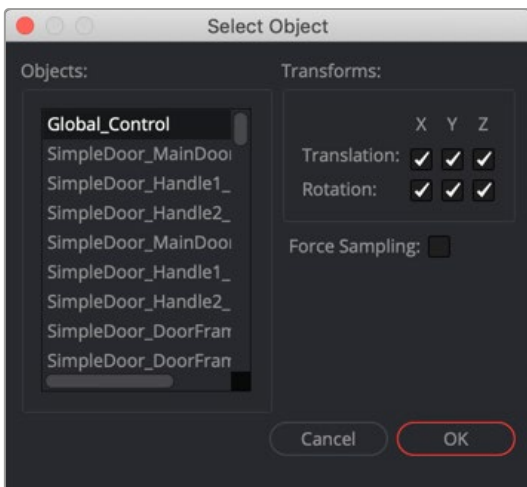
Import Transform Browser

Onscreen Transformation Controls

Onscreen Transformation controls provide an alternative way of using the controls in the Inspector. The viewer includes modes for transformation, rotation, and scaling. To change the mode of the onscreen controls, select one of the three buttons in the toolbar along the side of the viewer. The modes can also be toggled using the keyboard shortcut Q for translation, W for rotation, and E for scaling. In all three modes, an individual axis of the control may be dragged to affect just that axis, or the center of the control may be dragged to affect all three axes.

The scale sliders for most 3D nodes default to locked, which causes uniform scaling of all three axes. Unlock the Lock X/Y/Z Scale checkbox to scale an object on a single axis only.

Transform 3D onscreen transformation controls

Triangulate 3D [3Tri]

The Triangulate 3D node

Triangulate 3D Node Introduction

The Triangulate 3D node is a unique node in that it has no controls. This node turns polygon shapes into triangles. For instance, a quad that is four points becomes two triangles. It is used to convert complex polygon shapes into a mesh for easier processing.

Inputs

The Triangulate 3D node has a single required input for a 3D scene or 3D object.

— **Scene Input:** The orange scene input is connected to the 3D scene or 3D object you want to triangulate.

Basic Node Setup

The Triangulate 3D node is placed after the geometry you want to triangulate.
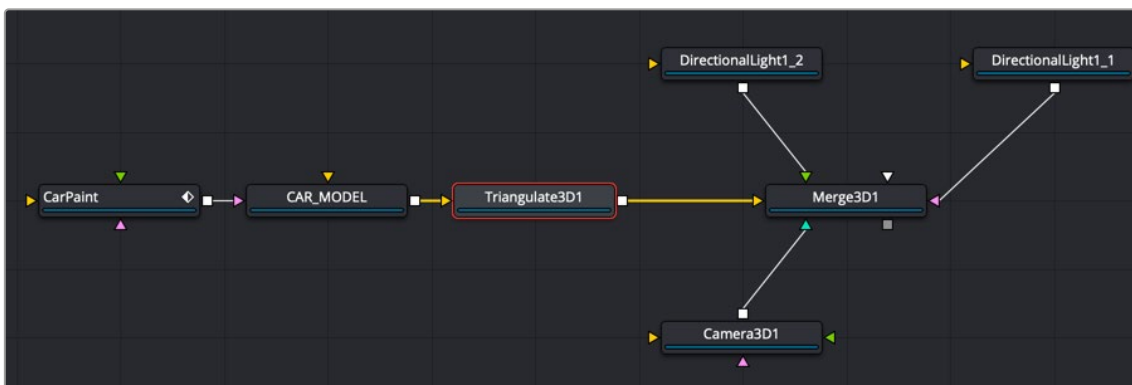
Triangulate 3D nodes connected after an imported model

Inspector
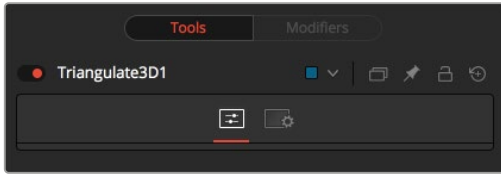
Triangulate 3D controls

Controls Tab

There are no controls for this node.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

UV Map 3D [3UV]
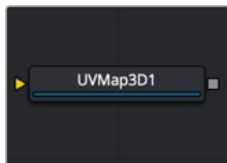
The UVMap 3D node

UV Map 3D Node Introduction

The UV Map 3D node replaces the UV texture coordinates on the geometry in the scene. These coordinates tell Fusion how to apply a texture to an object. While it is possible to adjust the global properties of the selected mapping mode, it is not possible to manipulate the UV coordinates of individual vertices directly from within Fusion. The onscreen controls drawn in the viewers are for reference only and cannot be manipulated.

Camera Projections with UV Map 3D

The Camera Mapping mode makes it possible to project texture coordinates onto geometry through a camera. Once you select Camera from the Mapping mode menu, then connect the Camera 3D node that you want to use to create the UV coordinates.

> NOTE That this does not directly project an image through the camera. The image to be projected should be connected to the diffuse texture input of whatever material is assigned to the objects. When the texture is applied, it uses the UV coordinates created by the camera. Because this is a texture projection and not light, the Alpha channel of the texture correctly sets the opacity of the geometry.

See the Camera 3D and Projector 3D nodes for alternate approaches to projection.

The projection can optionally be locked to the vertices as it appears on a selected frame.

This fails if the number of vertices in the mesh changes over time, as Fusion must be able to match up the mesh at the reference time and the current time. To be more specific, vertices may not be created or destroyed or reordered. So, projection locking does not work for many particle systems, or for primitives with animated subdivisions, or with duplicate nodes using non-zero time offsets.

> **NOTE:** The UV Map 3D node does not put a texture or material on the mesh; it only modifies the texture coordinates that the materials use. This may be confusing because the material usually sits upstream, as seen in the Basic Node Setup example below.

Inputs

The UV Map 3D node has two inputs: one for a 3D scene or 3D object and another optional input for a Camera 3D node.

— **Scene Input:** The orange scene input is connected to the 3D scene or 3D object you want to triangulate.

— **CameraInput:** This input expects the output of the Camera 3D node. It is only visible when the Camera Map mode menu is set to Camera.

Basic Node Setup

The UV Map 3D node is placed after all the geometry and set to Camera Map. Connecting a camera to the UV map allows you to line up the texture based on a centered camera position and 3D geometry.
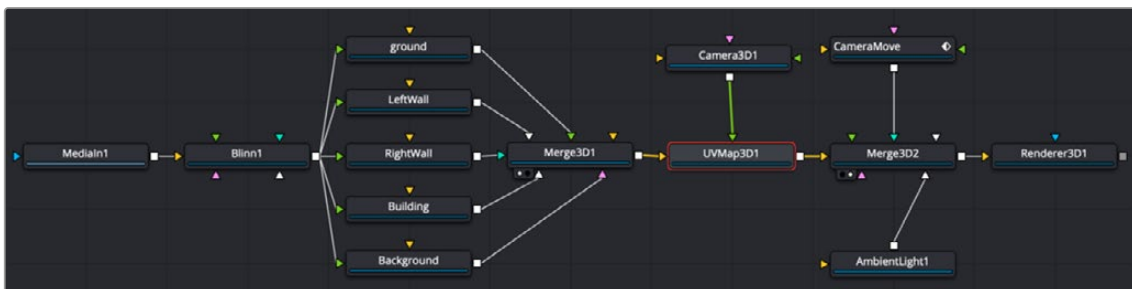
UV Map 3D is placed after the Merge 3D, with a camera connected to line up the texture

Inspector
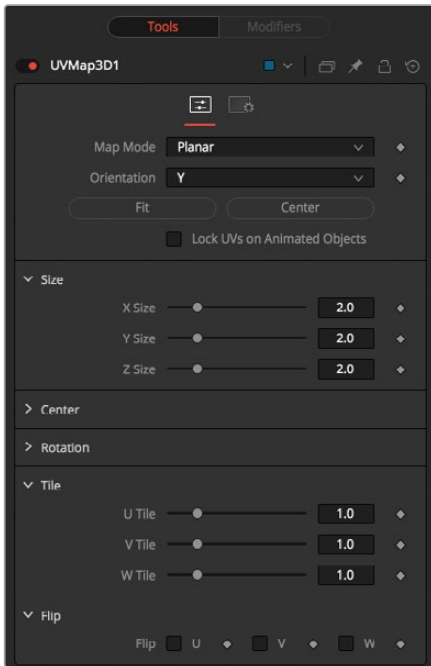
UV Map 3D controls

Controls Tab

The UV Map 3D Controls tab allows you to select Planar, Cylindrical, Spherical, XYZ, and Cubic mapping modes, which can be applied to basic Fusion primitives as well as imported geometry. The position, rotation, and scale of the texture coordinates can be adjusted to allow fine control over the texture's appearance. An option is also provided to lock the UV produced by this node to animated geometry according to a reference frame. This can be used to ensure that textures applied to animated geometry do not slide.

Map Mode

The Map mode menu is used to define how the texture coordinates are created. You can think of this menu as a way to select the virtual geometry that projects the UV space on the object.

— **Planar:** Creates the UV coordinates using a plane.
— **Cylindrical:** Creates the UV coordinates using a cylindrical-shaped object.
— **Spherical:** The UVs are created using a sphere.
— **XYZ to UVW:** The position coordinates of the vertices are converted to UVW coordinates directly. This is used for working with procedural textures.
— **CubeMap:** The UVs are created using a cube.
— **Camera:** Enables the Camera input on the node. After connecting a camera to the node, the texture coordinates are created based on camera projection.

Orientation X/Y/Z

Defines the reference axis for aligning the Map mode.

Fit

Clicking this button fits the Map mode to the bounding box of the input scene.

Center

Clicking this button moves the center of the Map mode to the bounding box center of the input scene.

Lock UVs on Animated Objects

If the object is animated, the UVs can be locked to it by enabling this option. The option also reveals the Ref Time slider, where it is possible to choose a reference frame for the UV mapping. Using this feature, it is not required to animate the UV map parameters. It is enough to set up the UV map at the reference time.

Size X/Y/Z

Defines the size of the projection object.

Center X/Y/Z

Defines the position of the projection object.

Rotation/Rotation Order

Use these buttons to select which order is used to apply the rotation along each axis of the object. For example, XYZ would apply the rotation to the X-axis first, followed by the Y-axis, and then the Z-axis.

Rotation X/Y/Z

Sets the orientation of the projection object for each axis, independent from the rotation order.

Tile U/V/W

Defines how often a texture fits into the projected UV space on the applicable axis. Note that the UVW coordinates are transformed, not a texture. This works best when used in conjunction with the Create Texture node.

Flip U/V/W

Mirrors the texture coordinates around the applicable axis.

Flip Faces (Cube Map Mode Only)

Mirrors the texture coordinates on the individual faces of the cube.

> **NOTE:** To utilize the full capabilities of the UV Map 3D node, it helps to have a basic understanding of how 2D images are mapped onto 3D geometry. When a 2D image is applied to a 3D surface, it is converted into a texture map that uses UV coordinates to determine how the image translates to the object. Each vertex on a mesh has a (U, V) texture coordinate pair that describes the appearance the object takes when it is unwrapped and flattened. Different mapping modes use different methods for working out how the vertices transform into a flat 2D texture. When using the UV Map 3D node to modify the texture coordinates on a mesh, it's best to do so using the default coordinate system of the mesh or primitive. So the typical workflow would look like Shape 3D > UV Map 3D > Transform 3D. The Transformation tab on the Shape node would be left to its default values, and the Transform 3D node following the UV Map 3D does any adjustments needed to place the node in the scene. Modifying/animating the transform of the Shape node causes the texture to slide across the shape, which is generally undesirable. The UV Map 3D node modifies texture coordinates per vertex and not per pixel. If the geometry the UV map is applied to is poorly tessellated, then undesirable artifacts may appear.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Weld 3D [3We]

The Weld 3D node

Weld 3D Node Introduction

Sometimes 3D geometry has vertices that should have been joined when the geometry was created, but for one reason or another they are not joined. This can cause artifacts, especially when the two vertices have different normals.

For example, you may find:

— The different normals produce hard shading/lighting edges where none were intended.
— If you try to Displace 3D the vertices along their normals, they crack.
— Missing pixels or doubled-up pixels in the rendered image.
— Particles pass through the tiny invisible cracks.

Instead of round tripping back to your 3D modeling application to fix the "duplicated" vertices, the Weld 3D node allows you to do this in Fusion. Weld 3D welds together vertices with the same or nearly the same positions. This can be used to fix cracking issues when vertices are displaced by welding the geometry before the Displace. There are no user controls to pick vertices. Currently, this node welds together just position vertices; it does not weld normals, texcoords, or any other vertex stream. So, although the positions of two vertices have been made the same, their normals still have their old values. This can lead to hard edges in certain situations.

Inputs

The Weld 3D node has a single input for a 3D scene or 3D object you want to repair.

— **Scene Input:** The orange scene input is connected to the 3D scene or 3D object you want to fix.

Basic Node Setup

The Weld 3D node is placed after the geometry that has duplicate vertices problems. Sometimes problems are exposed when displacing the geometry. In that case, placing the weld after the geometry but before the Displace 3D can repair the issues.
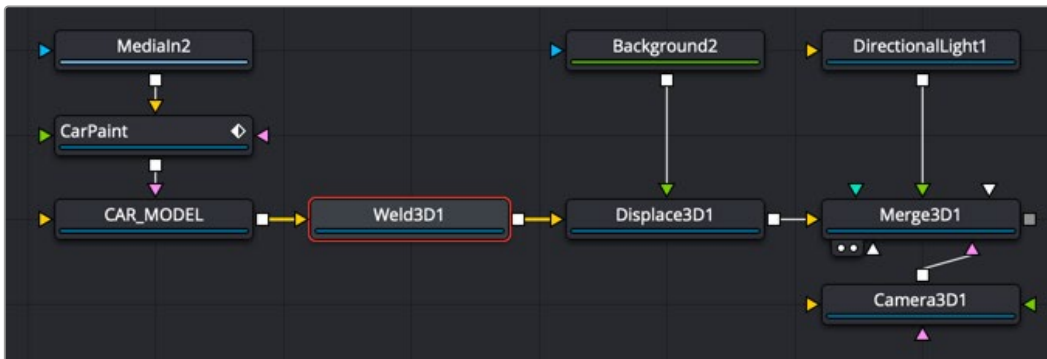
Weld 3D is placed after the 3D geometry that needs repair

Inspector
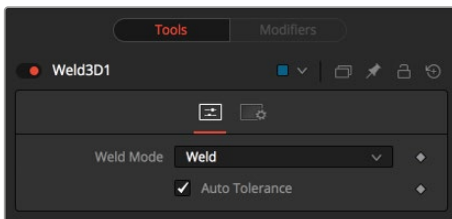
Weld 3D controls

Controls Tab

The Controls tab for the Weld 3D node includes a simple Weld Mode menu. You can choose between welding vertices or fracturing them.

Fracture

Fracturing is the opposite of welding, so all vertices are unwelded. This means that all polygon adjacency information is lost. For example, an Image Plane 3D normally consists of connected quads that share vertices. Fracturing the image plane causes it to become a bunch of unconnected quads.

Tolerance

In auto mode, the Tolerance value is automatically detected. This should work in most cases. It can also be adjusted manually if needed.

USAGE Use Weld 3D when issues occur with the geometry. Don't use it everywhere just because it's there, as it influences render time.

Weld 3D is intended to be used as a mesh robustness tool and not as a mesh editing tool to merge vertices. If you can see the gap between the vertices you want to weld in the 3D view, you are probably misusing Weld 3D. Unexpected things may happen when you do this; do so at your own peril.

LIMITATIONS Setting the tolerance too large can cause edges/faces to collapse to points.

If your model has detail distributed over several orders of scale, picking a tolerance value can be hard or impossible.

For example, suppose you have a model of the International Space Station and there are lots of big polygons and lots of really tiny polygons. If you set the tolerance too large, small polygons that shouldn't merge do; if you set the tolerance too small, some large polygons won't be merged.

Vertices that are far from the origin can fail to merge correctly. This is because bignumber + epsilon can exactly equal bignumber in float math. This is one reason it may be best to merge in local coordinates and not in world coordinates.

Sometimes Weld 3D-ing a mesh can make things worse. Take Fusion's cone, for instance. The top vertex of the cone is currently duplicated for each adjoining face, and they all have different normals. If you weld the cone, the top vertices merge and only have one normal, making the lighting look weird.

Weld 3D is not multithreaded.

WARNING Do not misuse Weld 3D to simplify (reduce the polygon count of) meshes. It is designed to efficiently weld vertices that differ by only very small values, like a 0.001 distance.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Modifier

Coordinate Transform 3D

Because of the hierarchical nature of the Fusion 3D node tree, the original position of an object in the 3D scene often fails to indicate the current position of the object. For example, an image plane might initially have a position at 1, 2, 1, but then be scaled, offset, and rotated by other nodes further downstream in the 3D scene, ending up with an absolute location of 10, 20, 5.

This can complicate connecting an object further downstream in the composition directly to the position of an upstream object. The Coordinate Transform modifier can be added to any set of XYZ coordinate controls and calculate the current position of a given object at any point in the scene hierarchy.

To add a Coordinate Transform modifier, simply right-click a number field on any node and select Modify With/CoordTransform Position from the Controls' contextual menu.

Inspector
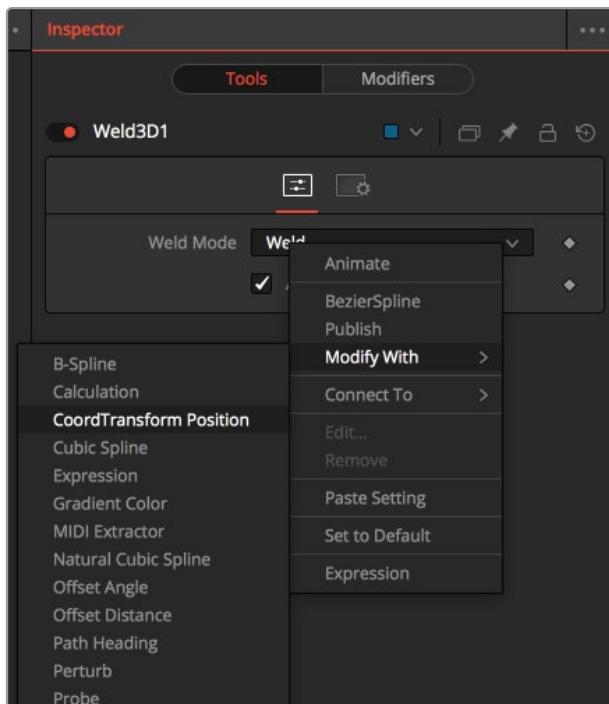
Weld 3D modifier tools

Target Object

This control should be connected to the 3D node that produces the original coordinates to be transformed. To connect a node, drag and drop a node from the node tree into the Text Edit control, or right-click the control and select the node from the contextual menu. It is also possible to type the node's name directly into the control.

Sub ID

The Sub ID slider can be used to target an individual sub-element of certain types of geometry, such as an individual character produced by a Text 3D node or a specific copy created by a Duplicate 3D node.

Scene Input

This control should be connected to the 3D node that outputs the scene containing the object at the new location. To connect a node, drag and drop a node from the node tree into the Text Edit control, or right-click the control and select an object from the Connect To submenu.

The Common Controls

Nodes that handle 3D geometry share several identical controls in the Inspector. This section describes controls that are common among 3D nodes.

Common Controls Tab
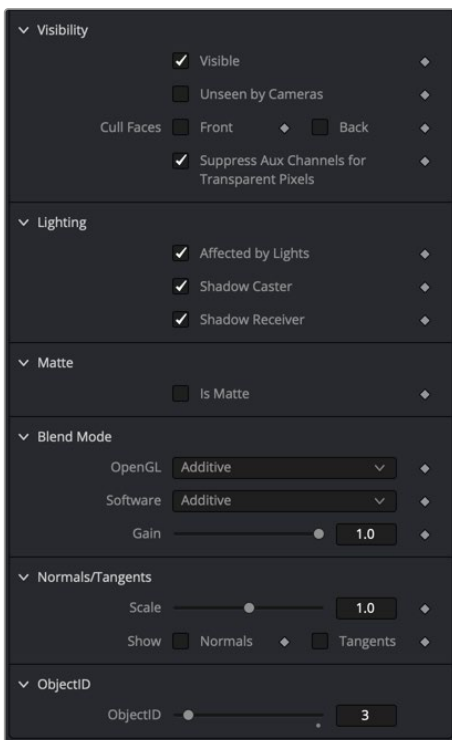
Common Controls 3D tab

These controls are often displayed in the lower half of the Controls tab. They appear in nodes that create or contain 3D geometry.

Visibility

— **Visible:** If this option is enabled, the object is visible in the viewers and in final renders. When disabled, the object is not visible in the viewers nor is it rendered into the output image by the Renderer 3D node. Also, a non-visible object does not cast shadows.

— **Unseen by Cameras:** When the Unseen by Cameras checkbox is enabled, the object is visible in the viewers (unless the Visible checkbox is disabled), except when viewed through a camera. Also, the object is not rendered into the output image by the Renderer 3D node. However, shadows cast by an unseen object are still visible when rendered by the software renderer in the Renderer 3D node, though not by the OpenGL renderer.

— **Cull Front Face/Back Face:** Use these options to eliminate rendering and display of certain polygons in the geometry. If Cull Back Face is selected, polygons facing away from the camera are not rendered and do not cast shadows. If Cull Front Face is selected, polygons facing toward the camera are not rendered and do not cast shadows. Enabling both options has the same effect as disabling the Visible checkbox.

— **Suppress Aux Channels for Transparent Pixels:** In previous versions of Fusion, transparent pixels were excluded by the software and Open GL render options in the Renderer 3D node. To be more specific, the software renderer excluded pixels with R,G,B,A set to 0, and the GL renderer excluded pixels with A set to 0. This is now optional. The reason you might want to do this is to get aux channels (e.g., Normals, Z, UVs) for the transparent areas. For example, suppose you want to replace the texture on a 3D element that is transparent in certain areas with a texture that is transparent in different areas. It would then be useful to have transparent areas set aux channels (particularly UVs). As another example, suppose you are adding depth of field. You probably do not want the Z-channel to be set on transparent areas, as this gives you a false depth. Also, keep in mind that the exclusion is based on the final pixel color including lighting, if it is on. So, if you have a specular highlight on a clear glass material, this checkbox does not affect it.

Lighting

— **Affected by Lights:** Disabling this checkbox causes lights in the scene to not affect the object. The object does not receive nor cast shadows, and it is shown at the full brightness of its color, texture, or material.

— **Shadow Caster:** Disabling this checkbox causes the object not to cast shadows on other objects in the scene.

— **Shadow Receiver:** Disabling this checkbox causes the object not to receive shadows cast by other objects in the scene.

Matte

Enabling the Is Matte option applies a special texture, causing the object to not only become invisible to the camera, but also making everything that appears directly behind the camera invisible as well. This option overrides all textures. For more information on Fog 3D and Soft Clipping, *see Chapter 85, "3D Compositing Basics,"* in the DaVinci Resolve Reference Manual, or Chapter 23 in the Fusion Reference Manual.

— **Is Matte:** When activated, objects whose pixels fall behind the matte object's pixels in Z do not get rendered. Two additional options are displayed when the Is Matte checkbox is activated.

— **Opaque Alpha:** When the Is Matte checkbox is enabled, the Opaque Alpha checkbox sets the Alpha value of the matte object to 1.

— **Infinite Z:** This option sets the value in the Z-channel to infinite. This checkbox is visible only when the Is Matte option is enabled.

Blend Mode

A Blend mode specifies which method is used by the renderer when combining this object with the rest of the scene. The blend modes are essentially identical to those listed in the section for the 2D Merge node. For a detailed explanation of each mode, see the section for that node.

The blending modes were originally designed for use with 2D images. Using them in a lit 3D environment can produce undesirable results. For best results, use the Apply modes in unlit 3D scenes using the software option in the Renderer 3D node.

— **OpenGL Blend Mode:** Use this menu to select the blending mode that is used when the geometry is processed by the OpenGL renderer in the Renderer 3D node. This is also the mode used when viewing the object in the viewers. Currently the OpenGL renderer supports a limited number of blending modes.

— **Software Blend Mode:** Use this menu to select the blending mode that is used when the geometry is processed by the software renderer. Currently, the software renderer supports all the modes described in the Merge node documentation, except for the Dissolve mode.

Normal/Tangents

Normals are imaginary lines perpendicular to each point on the surface of an object. They are used to illustrate the exact direction and orientation of every polygon on 3D geometry. Knowing the direction and orientation determines how the object gets shaded. Tangents are lines that exists along the surface's plane. These lines are tangent to a point on the surface. The tangent lines are used to describe the direction of textures you apply to the surface of 3D geometry.

— **Scale:** This slider increases or decreases the length of the vectors for both normals and tangents.

— **Show Normals:** Displays blue vectors typically extending outside the surface of the geometry. These normal vectors help indicate how different areas of the surface are illuminated based on the angle at which the light hits it.

— **Show Tangents:** Displays green vectors for Y and red vectors of X. The X and Y vectors represent the direction of the image or texture you are applying to the geometry.

Object ID

Use this slider to select which ID is used to create a mask from the object of an image. Use the Sample button in the same way as the Color Picker to grab IDs from the image displayed in the viewer. The image or sequence must have been rendered from a 3D software package with those channels included.

Common Materials Tab
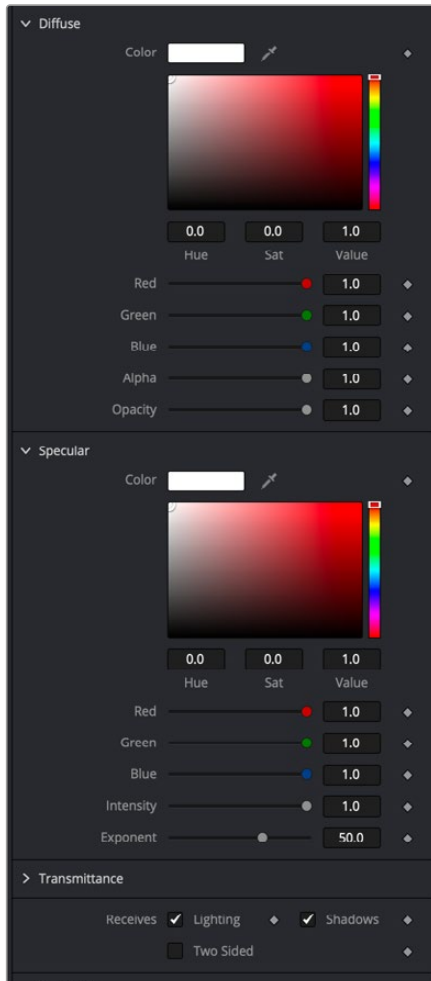
Common Materials 3D tab

The controls in the Materials tab are used to determine the appearance of the 3D object when lit. Most of these controls directly affect how the object interacts with light using a basic shader. For more advanced control over the objects appearance, you can use tools from the 3D Materials category of the Effects Library. These tools can be used to assemble a more finely detailed and precise shader.

When a shader is constructed using the 3D Material tools and connected to the 3D Object's material input, the controls in this tab are replaced by a label that indicates that an external material is currently in use.

Diffuse

Diffuse describes the base surface characteristics without any additional effects like reflections or specular highlights.

Diffuse Color

The Diffuse Color determines the basic color of an object when the surface of that object is either lit indirectly or lit by an ambient light. If a valid image is provided to the tools diffuse texture input, then the RGB values provided here are also multiplied by the color values of the pixels in the diffuse texture. The Alpha channel of the diffuse material can be used to control the transparency of the surface.

Alpha

This slider sets the material's Alpha channel value. This affects diffuse and specular colors equally, and affects the Alpha value of the material in the rendered output. If the tools diffuse texture input is used, then the Alpha value provided here is multiplied by the Alpha channel of the pixels in the image.

Opacity

Reducing the material's Opacity decreases the color and Alpha values of the specular and diffuse colors equally, making the material transparent and allowing hidden objects to be seen through the material.

Specular

The Specular section provides controls for determining the characteristics of light that reflects toward the viewer. These controls affect the appearance of the specular highlight that appears on the surface of the object.

Specular Color

Specular Color determines the color of light that reflects from a shiny surface. The more specular a material is, the glossier it appears. Surfaces like plastics and glass tend to have white specular highlights, whereas metallic surfaces like gold have specular highlights that tend to inherit their color from the material color. The basic shader material does not provide an input for textures to control the specularity of the object. Use tools from the 3D Material category when more precise control is required over the specular appearance.

Specular Intensity

Specular Intensity controls how strong the specular highlight is. If the specular intensity texture input has a valid connection, then this value is multiplied by the Alpha value of the input.

Specular Exponent

Specular Exponent controls the falloff of the specular highlight. The greater the value, the sharper the falloff, and the smoother and glossier the material appears. The basic shader material does not provide an input for textures to control the specular exponent of the object. Use tools from the 3D Material category when more precise control is required over the specular exponent.

Transmittance

Transmittance controls the way light passes through a material. For example, a solid blue sphere casts a black shadow, but one made of translucent blue plastic would cast a much lower density blue shadow.

There is a separate opacity option. Opacity determines how transparent the actual surface is when it is rendered. Fusion allows adjusting both opacity and transmittance separately. This might be a bit counter-intuitive to artists who are unfamiliar with 3D software at first. It is possible to have a surface that is fully opaque but transmits 100% of the light arriving upon it, effectively making it a luminous/emissive surface.

Attenuation

Attenuation determines how much color is transmitted through the object. For an object to have transmissive shadows, set the attenuation to (1, 1, 1), which means 100% of green, blue, red light passes through the object. Setting this color to RGB (1, 0, 0) means that the material transmits 100% of the red arriving at the surface but none of the green or blue light. This allows "stained glass" shadows.

Alpha Detail

When the Alpha Detail slider is set to 0, the Alpha channel of the object is ignored and the entire object casts a shadow. If it is set to 1, the Alpha channel determines what portions of the object cast a shadow.

Color Detail

The Color Detail slider modulates light passing through the surface by the diffuse color + texture colors. Use this to throw a shadow that contains color details of the texture applied to the object. Increasing the slider from 0 to 1 brings in more of diffuse color + texture color into the shadow. Note that the Alpha and opacity of the object are ignored when transmitting color, allowing an object with a solid Alpha to still transmit its color to the shadow.

Saturation

The Saturation slider controls the saturation of the color component transmitted to the shadow. Setting this to 0.0 results in monochrome shadows.

Receives Lighting/Shadows

These checkboxes control whether the material is affected by lighting and shadows in the scene. If turned off, the object is always fully lit and/or unshadowed.

Two-Sided Lighting

This makes the surface effectively two-sided by adding a second set of normals facing the opposite direction on the back side of the surface. This is normally off, to increase rendering speed, but can be turned on for 2D surfaces or for objects that are not fully enclosed, to allow the reverse or interior surfaces to be visible as well.

Normally, in a 3D application, only the front face of a surface is visible and the back face is culled, so that if a camera were to revolve around a plane in a 3D application, when it reached the backside, the plane would become invisible. Making a plane two sided in a 3D application is equivalent to adding another plane on top of the first but rotated by 180 degrees so the normals are facing the opposite direction on the backside. Thus, when you revolve around the back, you see the second image plane that has its normals facing the opposite way.

Fusion does exactly the same thing as 3D applications when you make a surface two sided. The confusion about what two-sided lighting does arises because Fusion does not cull backfacing polygons by default. If you revolve around a one-sided plane in Fusion, you still see it from the backside (but you are seeing the frontside bits duplicated through to the backside as if it were transparent). Making the plane two sided effectively adds a second set of normals to the backside of the plane.

Note that this can become rather confusing once you make the surface transparent, as the same rules still apply and produce a result that is counterintuitive. If you view from the frontside a transparent two-sided surface illuminated from the backside, it looks unlit.

Material ID

This control is used to set the numeric identifier assigned to this material. The Material ID is an integer number that is rendered into the MatID auxiliary channel of the rendered image when the Material ID option is enabled in the Renderer 3D tool. For more information, *see Chapter 85, "3D Compositing Basics,"* in the DaVinci Resolve Reference Manual, or Chapter 23 in the Fusion Reference Manual.

Common Transform Tab
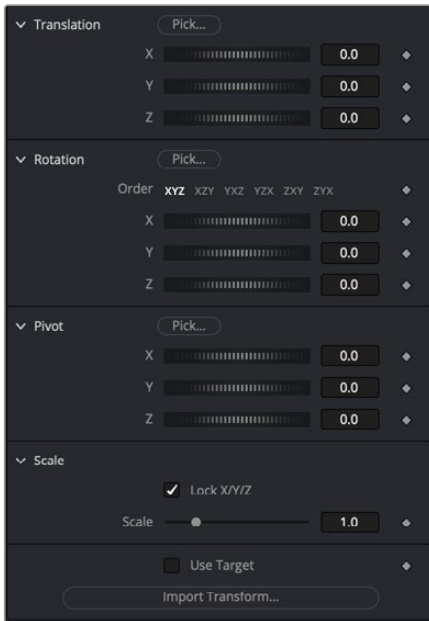
Common Transform 3D tab

Many tools in the 3D category include a Transform tab used to position, rotate, and scale the object in 3D space.

Translation

X, Y, Z Offset

These controls can be used to position the 3D element.

Rotation

Rotation Order

Use these buttons to select which order is used to apply rotation along each axis of the object.
For example, XYZ would apply the rotation to the X axis first, followed by the Y axis and then finally the Z axis.

X, Y, Z Rotation

Use these controls to rotate the object around its pivot point. If the Use Target checkbox is selected, then the rotation is relative to the position of the target; otherwise, the global axis is used.

Pivot

X, Y, Z Pivot

A Pivot point is the point around which an object rotates. Normally, an object rotates around its own center, which is considered to be a pivot of 0,0,0. These controls can be used to offset the pivot from the center.

Scale

X, Y, Z Scale

If the Lock X/Y/Z checkbox is checked, a single Scale slider is shown. This adjusts the overall size of the object. If the Lock checkbox is unchecked, individual X, Y, and Z sliders are displayed to allow individual

scaling in each dimension. Note: If the Lock checkbox is checked, scaling of individual dimensions is not possible, even when dragging specific axes of the Transformation Widget in scale mode.

Use Target

Selecting the Use Target checkbox enables a set of controls for positioning an XYZ target. When target is enabled, the object always rotates to face the target. The rotation of the object becomes relative to the target.

Import Transform

Opens a file browser where you can select a scene file saved or exported by your 3D application. It supports the following file types:

LightWave Scene	`.lws`
Max Scene	`.ase`
Maya Ascii Scene	`.ma`
dotXSI	`.xsi`

The Import Transform button imports only transformation data. For 3D geometry, lights, and cameras, consider using the File > FBX Import option.

Onscreen Transformation Controls

Viewer Transform controls

Most of the controls in the Transform tab are represented in the viewer with onscreen controls for transformation, rotation, and scaling. To change the mode of the onscreen controls, select one of the three buttons in the toolbar in the upper left of the viewer. The modes can also be toggled using the keyboard shortcut Q for translation, W for rotation, and E for scaling. In all three modes, individual axes of the control may be dragged to affect just that axis, or the center of the control may be dragged to affect all three axes.

The scale sliders for most 3D tools default to locked, which causes uniform scaling of all three axes. Unlock the Lock X/Y/Z Scale checkbox to scale an object on a single axis only.

Settings Tab

Common Settings 3D controls

The Common Settings tab can be found on most tools in Fusion. The following controls are specific settings for 3D nodes.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node are displayed in the Inspector. Dragging a connected node from the node tree into the field hides that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line reappears.

Comment Tab

The Comment tab contains a single text control that is used to add comments and notes to the tool. When a note is added to a tool, a small red dot icon appears next to the setting's tab icon and a text bubble appears on the node. To see the note in the Node Editor, hold the mouse pointer over the node for a moment. The contents of the Comments tab can be animated over time, if required.

Scripting Tab

The Scripting tab is present on every tool in Fusion. It contains several edit boxes used to add scripts that process when the tool is rendering. For more details on the contents of this tab, please consult the scripting documentation.

3D Light Nodes

This chapter details the 3D Light nodes available when creating 3D composites in Fusion. The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Ambient Light [3AL]

The Ambient Light node

Ambient Light Node Introduction

An Ambient Light is a directionless light that globally illuminates a scene. It has no real position or rotation, although an onscreen control appears in the viewer to indicate that a light is present in the scene. Position controls for the viewer are provided to move the widget out of the way of other geometry, if necessary.

Similar to a Camera 3D, you connect lights into a Merge 3D and view them in the scene by viewing the Merge 3D node. Selecting a light node and loading it into the viewer does not show anything.

Inputs

The Ambient Light node includes a single optional orange input for a 3D scene or 3D geometry.

— **SceneInput:** The orange input is an optional input that accepts a 3D scene. If a scene is provided, the Transform controls in this node apply to the entire scene provided.

Basic Node Setup

The Ambient Light node is designed to be part of a larger 3D scene. You connect the light directly into a Merge 3D. Separating lights into different Merge 3D nodes allows you to control which lights affect which objects.
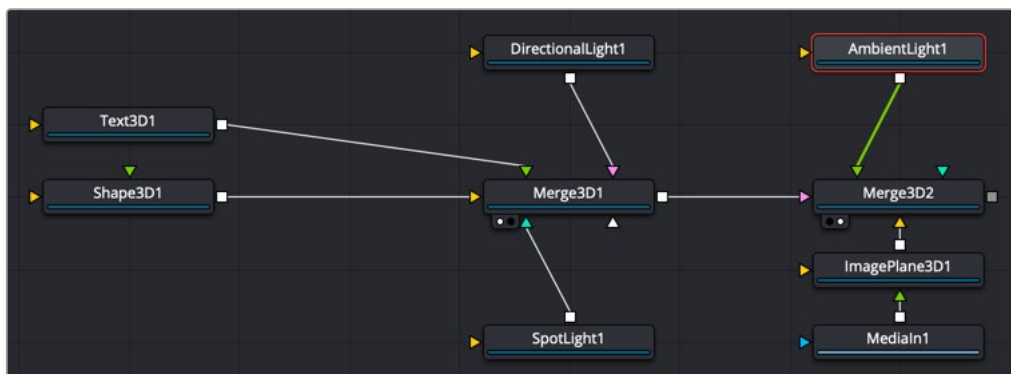
Ambient Light node structure

Inspector

Ambient Light controls

Controls Tab

The Controls tab is used to set the color and brightness of the ambient light.

Enabled

When the Enabled checkbox is turned on, the ambient light affects the scene. When the checkbox to turned off, the light is turned off. This checkbox performs the same function as the red switch to the left of the node's name in the Inspector.

Color

Use this standard Color control to set the color of the light.

Intensity

Use this slider to set the Intensity of the ambient light. A value of 0.2 indicates 20% percent light. A perfectly white texture lit only with a 0.2 ambient light would render at 20% gray (.2, .2, .2).

Common Controls

Transform and Settings Tabs

The options presented in the Transform and Settings tabs are commonly found in other lighting nodes. For more detail on the controls found in these tabs, see "The Common Controls" section at the end of this chapter.

Directional Light [3DL]

The Directional Light node

Directional Light Node Introduction

A directional light is a light with a clear direction but without a clear source or distance, similar to sunlight. This light shows an onscreen control, but the position of the control has no meaning. The rotation of the control is used to determine from where in the scene the light appears to be coming.

Similar to a Camera 3D, you connect lights into a Merge 3D and view them in the scene by viewing the Merge 3D node. Selecting a light node and loading it into the viewer does not show anything.

Inputs

The Directional Light node includes a single optional orange input for a 3D scene or 3D geometry.

— **SceneInput:** The orange input is an optional input that accepts a 3D scene. If a scene is provided, the Transform controls in this node apply to the entire scene provided.

Basic Node Setup

The Directional Light node is designed to be part of a larger 3D scene. You connect the light directly into a Merge 3D. Separating lights into different Merge 3D nodes allows you to control which lights affect which objects.
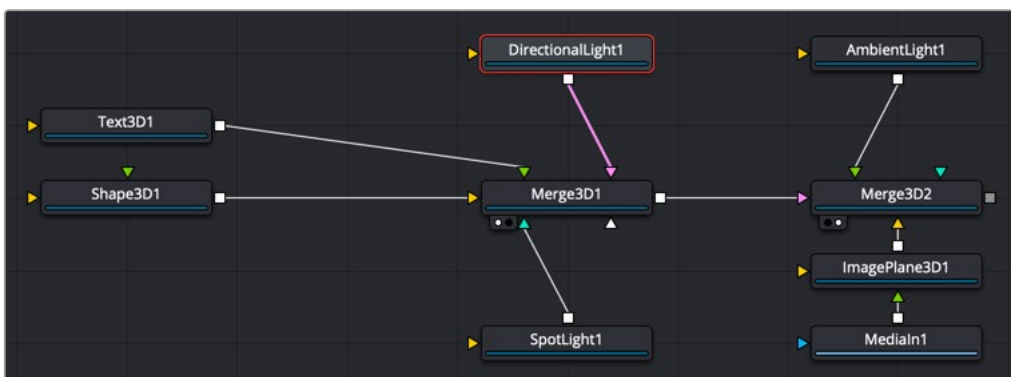
Directional Light node structure

Inspector

Directional Light controls

Controls Tab

The Controls tab is used to set the color and brightness of the directional light. The direction of the light source is controlled by the rotation controls in the Transform tab.

Enabled

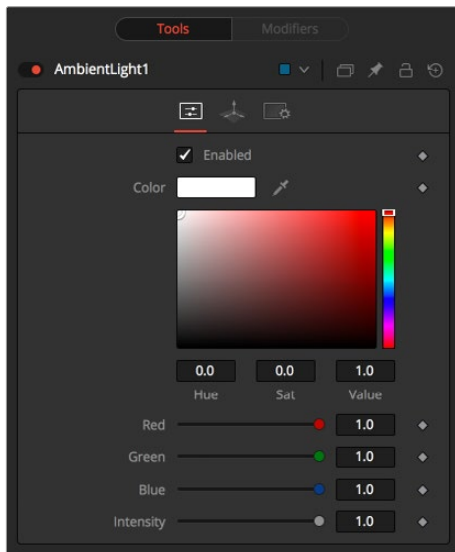When the Enabled checkbox is turned on, the directional light affects the scene. When the checkbox is turned off, the light is turned off. This checkbox performs the same function as the red switch to the left of the node's name in the Inspector.

Color

Use this standard Color control to set the color of the light.

Intensity

Use this slider to set the Intensity of the directional light. A value of 0.2 indicates 20% percent light.

Common Controls

Transform and Settings Tabs

The options presented in the Transform and Settings tabs are commonly found in other lighting nodes. For more detail on the controls found in these tabs, see "The Common Controls" section at the end of this chapter.

Point Light [3PL]

The Point Light node

Point Light Node Introduction

A point light is a light source with a clear position in space that emits light in all directions. A light bulb is a good example of a point light.

This light shows an onscreen control, although only the position and distance of the control affect the light. Since the light is a 360-degree source, rotation has no meaning. Additionally, a point light may fall off with distance, unlike an ambient or directional light.

Similar to a Camera 3D, you connect lights into a Merge 3D and view them in the scene by viewing the Merge 3D node. Selecting a light node and loading it into the viewer does not show anything.

Inputs

The Point Light node includes a single optional orange input for a 3D scene or 3D geometry.

— **SceneInput:** The orange input is an optional input that accepts a 3D scene. If a scene is provided, the Transform controls in this node apply to the entire scene provided.

Basic Node Setup

The Point Light node is designed to be part of a larger 3D scene. You connect the light directly into a Merge 3D. Separating lights into different Merge 3D nodes allows you to control which lights affect which objects.
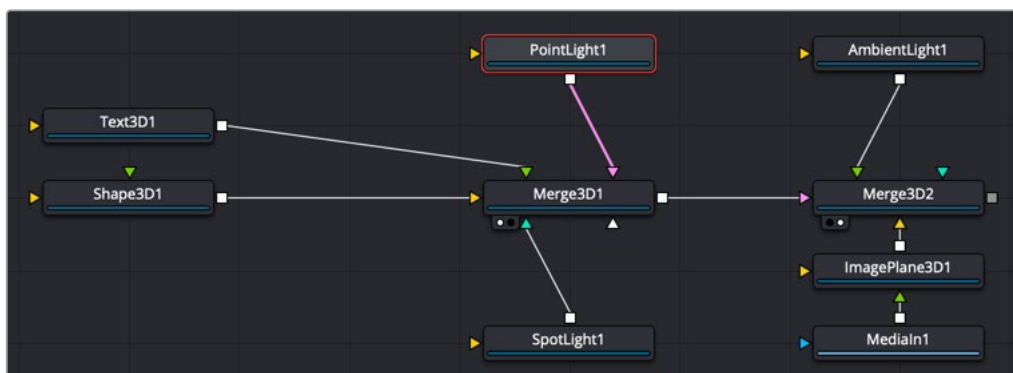
Point Light node structure

Inspector

Point Light controls

Controls Tab

The Controls tab is used to set the color and brightness of the point light. The position and distance of the light source are controlled in the Transform tab.

Enabled

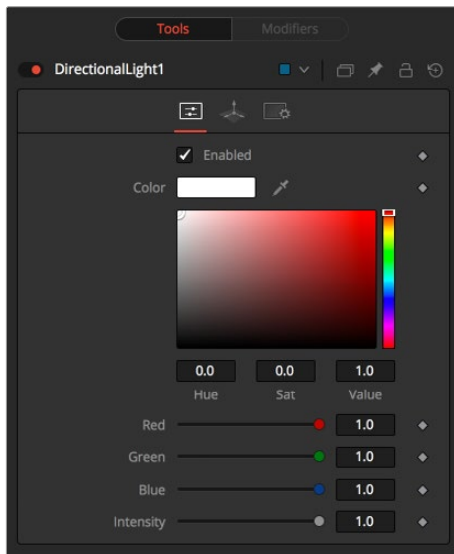When the Enabled checkbox is turned on, the point light affects the scene. When the checkbox is turned off the light is turned off. This checkbox performs the same function as the red switch to the left of the node's name in the Inspector.

Color

Use this standard Color control to set the color of the light.

Intensity

Use this slider to set the Intensity of the point light. A value of 0.2 indicates 20% percent light.

Decay Type

A point light defaults to No Decay, meaning that its light has equal intensity at all points in the scene. To cause the intensity to fall off with distance, set the Decay Type to either Linear or Quadratic modes.

Common Controls

Transform and Settings Tabs

The options presented in the Transform and Settings tabs are commonly found in other lighting nodes. For more detail on the controls found in these tabs, see "The Common Controls" section at the end of this chapter.

Spot Light [3SL]
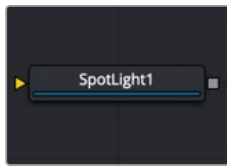
The Spot Light node

Spot Light Node Introduction

A spotlight is a light that comes from a specific point and that has a clearly defined cone, with falloff of the light to the edges. Experienced stage and theatre lighting technicians may recognize the spotlight as being very similar to practical lights used in live productions. This is the only type of light capable of casting shadows.

Similar to a Camera 3D, you connect lights into a Merge 3D and view them in the scene by viewing the Merge 3D node. Selecting a light node and loading it into the viewer does not show anything.

Inputs

The Spot Light node includes a single optional orange input for a 3D scene or 3D geometry.

— **SceneInput:** The orange input is an optional input that accepts a 3D scene. If a scene is provided, the Transform controls in this node apply to the entire scene provided.

Basic Node Setup

The Spot Light node is designed to be part of a larger 3D scene. You connect the light directly into a Merge 3D. Separating lights into different Merge 3D nodes allows you to control which lights affect which objects.
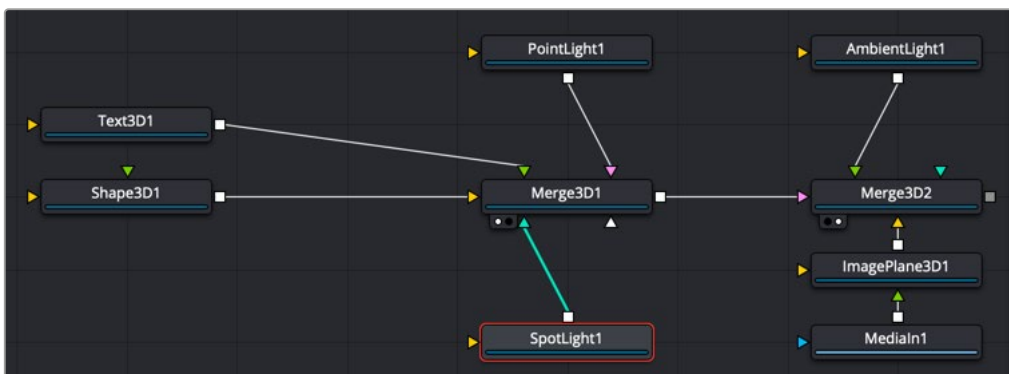
Spot Light node structure

Inspector
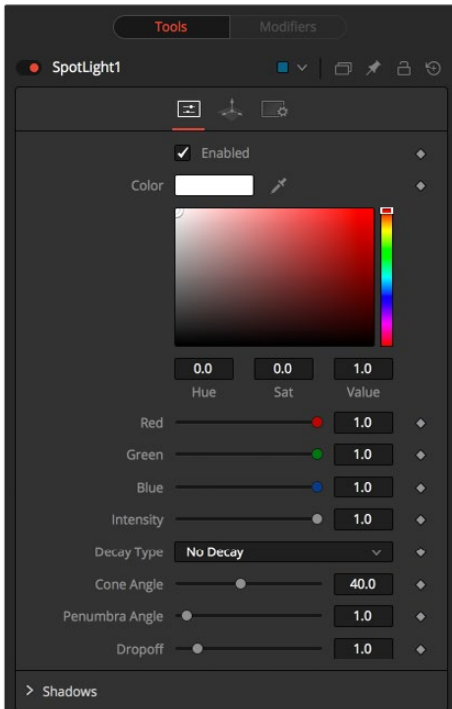
Spot Light control

Controls Tab

The Controls tab is used to set the color and brightness of the spotlight. The position, rotation, and distance of the light source are controlled in the Transform tab.

Enabled

When the Enabled checkbox is turned on, the spotlight affects the scene. When the checkbox is turned off the light is turned off. This checkbox performs the same function as the red switch to the left of the node's name in the Inspector.
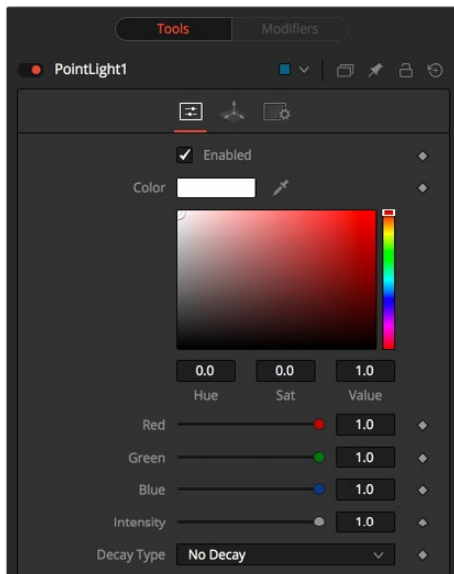
Color

Use this standard Color control to set the color of the light.

Intensity

Use this slider to set the Intensity of the spot light. A value of 0.2 indicates 20% percent light.

Decay Type

A spotlight defaults to No Falloff, meaning that its light has equal intensity on geometry despite the distance from the light to the geometry. To cause the intensity to fall off with distance, set the Decay type to either Linear or Quadratic modes.

Cone Angle

The Cone Angle of the light refers to the width of the cone where the light emits its full intensity. The larger the angle, the wider the cone angle, up to a limit of 90 degrees.

Penumbra Angle

The Penumbra Angle determines the area beyond the cone angle where the light's intensity falls off toward 0. A larger penumbra angle defines a larger falloff, while a value of 0 generates a hard-edged light.

Dropoff

The Dropoff controls how quickly the penumbra angle falls off from full intensity to 0.

Shadows

This section provides several controls used to define the shadow map used when this spotlight creates shadows. For more information, *see Chapter 85, "3D Compositing Basics,"* in the DaVinci Resolve Reference Manual, or Chapter 23 in the Fusion Reference Manual.

Enable Shadows

The Enable Shadows checkbox should be selected if the light is to produce shadows. This defaults to selected.

Shadow Color

Use this standard Color control to set the color of the shadow. This defaults to black (0, 0, 0).

Density

The shadow density determines the transparency of the shadow. A density of 1.0 produces a completely opaque shadow, whereas lower values make the shadow more transparent.

Shadow Map Size

The Shadow Map Size control determines the size of the bitmap used to create the shadow map. Larger values produce more detailed shadow maps at the expense of memory and performance.

Shadow Map Proxy

Shadow Map Proxy determines the size of the shadow map used when the Proxy or Auto Proxy modes are enabled. A value of 0.5 would produce a shadow map at half the resolution defined in the Shadow Map Size.

Multiplicative/Additive Bias

Shadows are essentially textures applied to objects in the scene, so there is occasionally Z-fighting, where the portions of the object that should be receiving the shadows render over the top of the shadow. Biasing works by adding a small depth offset to move the shadow away from the surface it is shadowing, eliminating the Z-fighting. Too little bias and the objects can self-shadow themselves. Too much bias and the shadow can become separated from the surface. Adjust the Multiplicative Bias first, and then fine tune the result using the Additive Bias control.

For more information, see the Multiplicative and Additive Bias section of Chapter 85, "3D Compositing Basics," in the DaVinci Resolve Reference Manual, or Chapter 25 in the Fusion Reference Manual.

Force All Materials Non-Transmissive

Normally, an RGBAZ shadow map is used when rendering shadows. By enabling this option, you are forcing the renderer to use a Z-only shadow map. This can lead to significantly faster shadow rendering while using a fifth as much memory. The disadvantage is that you can no longer cast "stained glass"-like shadows.

Shadow Map Sampling

Sets the quality for sampling of the shadow map.

Softness

Soft edges in shadows are produced by filtering the shadow map when it is sampled. Fusion provides two separate filtering methods for rendering shadows, which produce different effects.

> **NOTE:** Shadows have a hard edge. No filtering of the shadow map is done at all. The advantage of this method is that you only have to sample one pixel in the shadow map, so it is fast.

— **Constant:** Shadows edges have a constant softness. A filter with a constant width is used when sampling the shadow map. Adjusting the Constant Softness slider controls the size of the filter. Note that the larger you make the filter, the longer it takes to render the shadows.

— **Variable:** The shadow edge softness grows the further the shadow receiver is positioned from the shadow caster. The variable softness is achieved by changing the size of the filter based on the distance between the receiver and caster. When this option is selected, the Softness Falloff, Min Softness, and Max Softness sliders appear.

Constant Softness

If the Softness is set to Constant, then this slider appears. It can be used to set the overall softness of the shadow.

Softness Falloff

The Softness Falloff slider appears when the Softness is set to variable. This slider controls how fast the softness of shadow edges grows with distance. More precisely, it controls how fast the shadow map filter size grows based upon the distance between the shadow caster and receiver. Its effect is mediated by the values of the Min and Max Softness sliders.

Min Softness

The Min Softness slider appears when the Softness is set to Variable. This slider controls the Minimum Softness of the shadow. The closer the shadow is to the object casting the shadow, the sharper it is, up to the limit set by this slider.

Max Softness

The Max Softness slider appears when the Softness is set to Variable. This slider controls the Maximum Softness of the shadow. The further the shadow is from the object casting the shadow, the softer it is, up to the limit set by this slider.

Common Controls

Transform and Settings Tabs

The options presented in the Transform and Settings tabs are commonly found in other lighting nodes. For more detailed information on the controls found in these tabs, see "The Common Controls" section at the end of this chapter.

The Common Controls

Nodes that handle 3D lighting share several identical controls in the Inspector. This section describes controls that are common among 3D lighting nodes.

Common Transform Tab
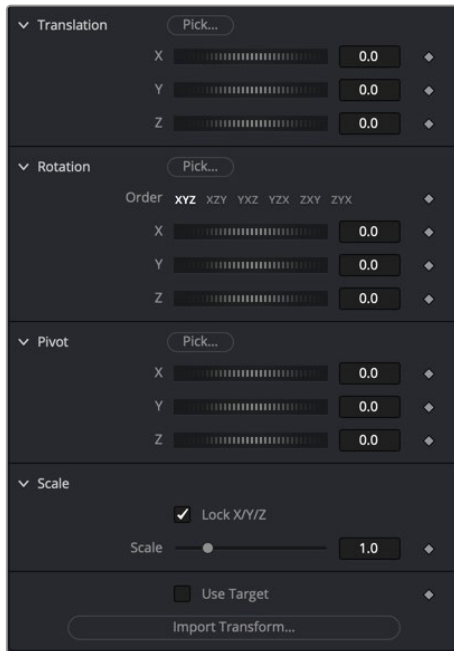
Common Transform 3D tab

Many tools in the 3D category include a Transform tab used to position, rotate, and scale the object in 3D space.

Translation

X, Y, Z Offset

These controls can be used to position the 3D element.

Rotation

Rotation Order

Use these buttons to select which order is used to apply Rotation along each axis of the object. For example, XYZ would apply the rotation to the X axis first, followed by the Y axis, and finally the Z axis.

X, Y, Z Rotation

Use these control to rotate the object around its pivot point. If the Use Target checkbox is selected, then the rotation is relative to the position of the target; otherwise, the global axis is used.

Pivot

X, Y, Z Pivot

A pivot point is the point around which an object rotates. Normally, an object rotates around its own center, which is considered to be a pivot of 0,0,0. These controls can be used to offset the pivot from the center.

Scale

X, Y, Z Scale

If the Lock X/Y/Z checkbox is checked, a single Scale slider is shown. This adjusts the overall size of the object. If the Lock checkbox is unchecked, individual X, Y, and Z sliders are displayed to allow individual scaling in each dimension. Note: If the Lock checkbox is checked, scaling of individual dimensions is not possible, even when dragging specific axes of the Transformation Widget in scale mode.

Use Target

Selecting the Use Target checkbox enables a set of controls for positioning an XYZ target. When Target is enabled, the object always rotates to face the target. The rotation of the object becomes relative to the target.

Import Transform

Opens a file browser where you can select a scene file saved or exported by your 3D application. It supports the following file types:

LightWave Scene	`.lws`
Max Scene	`.ase`
Maya Ascii Scene	`.ma`
dotXSI	`.xsi`

The Import Transform button imports only transformation data. For 3D geometry, lights, and cameras, consider using the File > FBX Import option.

Onscreen Transformation Controls

Viewer Transform buttons

Viewer Transform buttons

Most of the controls in the Transform tab are represented in the viewer with onscreen controls for transformation, rotation, and scaling. To change the mode of the onscreen controls, select one of the three buttons in the toolbar in the upper left of the viewer. The modes can also be toggled using the keyboard shortcut Q for translation, W for rotation, and E for scaling. In all three modes, individual axes of the control may be dragged to affect just that axis, or the center of the control may be dragged to affect all three axes.

The Scale sliders for most 3D tools default to locked, which causes uniform scaling of all three axes. Unlock the Lock X/Y/Z Scale checkbox to scale an object on a single axis only.

Settings Tab

Common Settings 3D controls

The Common Settings tab can be found on almost every tool found in Fusion. The following controls are specific settings for 3D nodes.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, fields for each input on a node are displayed. Dragging a connected node from the node tree into the field hides that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line reappears.

Comment Tab

The Comment tab contains a single text control that is used to add comments and notes to the tool. When a note is added to a tool, a small red dot icon appears next to the setting's tab icon, and a text bubble appears on the node. To see the note in the Node Editor, hold the mouse pointer over the node for a moment. The contents of the Comments tab can be animated over time, if required.

Scripting Tab

The Scripting tab is present on every tool in Fusion. It contains several edit boxes used to add scripts that process when the tool is rendering. For more details on the contents of this tab, please consult the scripting documentation.

3D Material Nodes

This chapter details the 3D Material nodes available when creating 3D composites in Fusion. The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Blinn [3Bl]

The Blinn node

Blinn Node Introduction

The Blinn node is a basic illumination material that can be applied to geometry in the 3D scene. It describes how the object responds to light and provides multiple texture map inputs to allow fine control over the diffuse, specular, and bump map components of the material.

The standard basic material provided in the Materials tab of most geometry nodes is a simplified version of the Blinn node. The primary difference is that the Blinn node provides additional texture map inputs beyond just diffuse.

The Blinn node outputs a 3D Material that can be connected to the material inputs on any 3D geometry node.

The Blinn model in Fusion calculates the highlight as the dot product of the surface normal and the half angle vector between light source and viewer (dot(N, H)). This may not always match the Blinn model illumination model used by other 3D applications.

Inputs

There are five inputs on the Blinn node that accept 2D images or 3D materials. These inputs control the overall color and image used for the 3D object as well as the color and texture used in the specular highlight. Each of these inputs multiplies the pixels in the texture map by the equivalently named parameters in the node itself. This provides an effective method for scaling parts of the material.

— **Diffuse Texture:** The orange Diffuse Texture input accepts a 2D image or a 3D material to be used as a main object texture map.

— **Specular Color Material:** The green Specular Color material input accepts a 2D image or a 3D material to be used as the color texture map for specula highlight areas.

— **Specular Intensity Materials:** The magenta Specular Intensity material input accepts a 2D image or a 3D material to be used to alter the intensity of specular highlights. When the input is a 2D image, the Alpha channel is used to create the map, while the color channels are discarded.

— **Specular Exponent Material:** The teal Specular Exponent material input accepts a 2D image or a 3D material that is used as a falloff map for the material's specular highlights. When the input is a 2D image, the Alpha channel is used to create the map, while the color channels are discarded.

— **Bump Map Material:** The white Bump Map material input accepts only a 3D material. Typically, you connect the texture into a Bump Map node, and then connect the Bump Map node to this input. This input uses the RGB information as texture-space normals.

When nodes have as many inputs as this one does, it is often difficult to make connections with any precision. Hold down the Option (macOS) or Alt (Windows) key while dragging the output from another node over the node tile, and keep holding Option or Alt when releasing the left mouse button. A small drop-down menu listing all the inputs provided by the node appears. Click on the desired input to complete the connection. Alternatively, you can drag the output from a node with the right mouse button to activate the same menu.

Basic Node Setup

The output of a Blinn node output is connected to the material input on a 3D scene or 3D geometry node to which you want the shader applied. The Blinn inputs can use images as the diffuse color material (orange) and specular color material (green). This can lead to a smooth, shiny material.

A Blinn shader with diffuse and specular color materials connected

Inspector
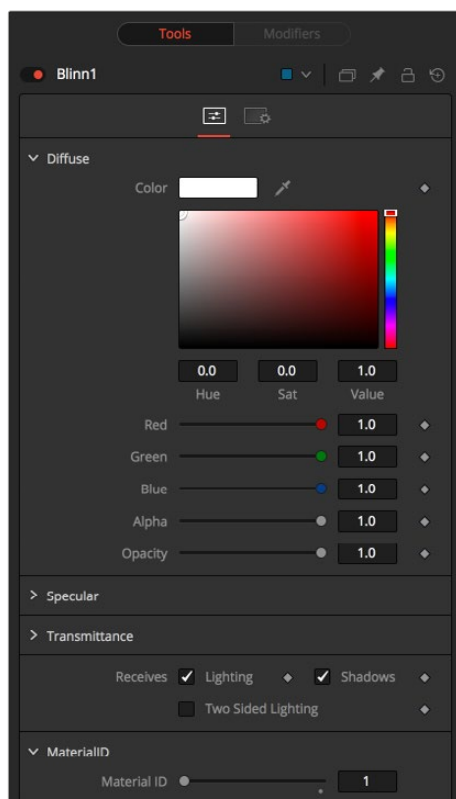
Blinn controls

Controls Tab

The Controls tab is the primary tab for the Blinn node. It controls the color and shininess applied to the surface of the 3D geometry.

Diffuse

Diffuse describes the base surface characteristics without any additional effects like reflections or specular highlights. Besides defining the base color of an object, the diffuse color also defines the transparency of the object. The Alpha in a diffuse texture map can be used to make portions of the surface transparent.

Diffuse Color

A material's Diffuse Color describes the base color presented by the material when it is lit indirectly or by ambient light. If a diffuse texture map is provided, then the color value provided here is multiplied by the color values in the texture.

Alpha

This slider sets the material's Alpha channel value. This affects diffuse and specular colors equally and affects the Alpha value of the material in the rendered output. If a diffuse texture map is provided, then the Alpha value set here is multiplied by the Alpha values in the texture map.

Opacity

Reducing the material's opacity decreases the color and Alpha values of the specular and diffuse colors equally, making the material transparent.

Specular

The parameters in the Specular section describe the look of the specular highlight of the surface. These values are evaluated in a different way for each illumination model.

Specular Color

Specular Color determines the color of light that reflects from a shiny surface. The more specular a material is, the glossier it appears. Surfaces like plastics and glass tend to have white specular highlights, whereas metallic surfaces like gold have specular highlights that inherit their color from the material color. If a specular texture map is provided, then the value provided here is multiplied by the color values from the texture.

Specular Intensity

Specular Intensity controls how strong the specular highlight is. If the specular intensity texture is provided, then this value is multiplied by the Alpha value of the texture.

Specular Exponent

Specular Exponent controls the falloff of the specular highlight. The greater the value, the sharper the falloff, and the smoother and glossier the material appears. If the specular exponent texture is provided, then this value is multiplied by the Alpha value of the texture map.

Transmittance

Transmittance controls the way light passes through a material. For example, a solid blue sphere casts a black shadow, but one made of translucent blue plastic would cast a much lower density blue shadow.

There is a separate Opacity option. Opacity determines how transparent the actual surface is when it is rendered. Fusion allows adjusting both opacity and transmittance separately. At first, this might be a bit counterintuitive to those who are unfamiliar with 3D software. It is possible to have a surface that is fully opaque but transmits 100% of the light arriving upon it, effectively making it a luminous/emissive surface.

Attenuation

Attenuation determines how much color is passed through the object. For an object to have transmissive shadows, set the attenuation to (1, 1, 1), which means 100% of green, blue, and red light passes through the object. Setting this color to RGB (1, 0, 0) means that the material transmits 100% of the red arriving at the surface but none of the green or blue light. This can be used for "stained glass"-styled shadows.

Alpha Detail

When the Alpha Detail slider is set to 0, the Alpha channel of the object is ignored and the entire object casts a shadow. If it is set to 1, the Alpha channel determines what portions of the object cast a shadow.

Color Detail

The Color Detail slider modulates light passing through the surface by the diffuse color + texture colors. Use this to throw a shadow that contains color details of the texture applied to the object. Increasing the slider from 0 to 1 brings in more diffuse color + texture color into the shadow. Note that the Alpha and opacity of the object are ignored when transmitting color, allowing an object with a solid Alpha to still transmit its color to the shadow.

Saturation

The Saturation slider controls the saturation of the color component transmitted to the shadow. Setting this to 0.0 results in monochrome shadows.

Receives Lighting/Shadows

These checkboxes control whether the material is affected by lighting and shadows in the scene. If turned off, the object is always fully lit and/or unshadowed.

Two-Sided Lighting

This effectively makes the surface two sided by adding a second set of normals facing the opposite direction on the backside of the surface. This is normally off to increase rendering speed, but it can be turned on for 2D surfaces or for objects that are not fully enclosed, to allow the reverse or interior surfaces to be visible as well.

Normally, in a 3D application, only the front face of a surface is visible and the back face is culled, so that if a camera were to revolve around a plane in a 3D application, when it reached the backside, the plane would become invisible. Making a plane two sided in a 3D application is equivalent to adding another plane on top of the first but rotated by 180 degrees so the normals are facing the opposite direction on the backside. Thus, when you revolve around the back, you see the second image plane, which has its normals facing the opposite way.

Fusion does exactly the same thing as 3D applications when you make a surface two sided. The confusion about what two-sided lighting does arises because Fusion does not cull back-facing polygons by default. If you revolve around a one-sided plane in Fusion, you still see it from the backside (but you are seeing the frontside duplicated through to the backside as if it were transparent). Making the plane two sided effectively adds a second set of normals to the backside of the plane.

> **NOTE:** This can become rather confusing once you make the surface transparent, as the same rules still apply and produce a result that is counterintuitive. If you view from the frontside a transparent two-sided surface illuminated from the backside, it looks unlit.

Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Channel Boolean [3Bol]

The Channel Boolean node

Channel Boolean Node Introduction

The Channel Boolean (not to be confused with the 2D Channel Booleans) can be used to remap and modify channels of 3D materials using mathematical operations. For example, if you want to use the red channel of a material to control a scalar input of an illumination model that uses the Alpha channel (e.g., Blinn. SpecularExponent), you can remap the channels here. Furthermore, it allows the use of geometry-specific information like texture space coordinates and normals.

Inputs

There are two inputs on the Channel Boolean Node: one for the foreground material, and one for the background material. Both inputs accept either a 2D image or a 3D material like Blinn, Cook-Torrence, or Phong node.

— **BackgroundMaterial:** The orange background material input accepts a 2D image or a 3D material.

— **ForegroundMaterial:** The green foreground input also accepts a 2D image or a 3D material.

Basic Node Setup

There are many uses for the material 3D Channel Boolean. Most often it is used to combine material looks or manipulate UV texture coordinates

In the below example, the Channel Boolean node combines the Cook Torrance and Blinn materials. It uses the math operands in the Channel Boolean to switch, invert, and mix the two inputs, creating a neon flickering effect.
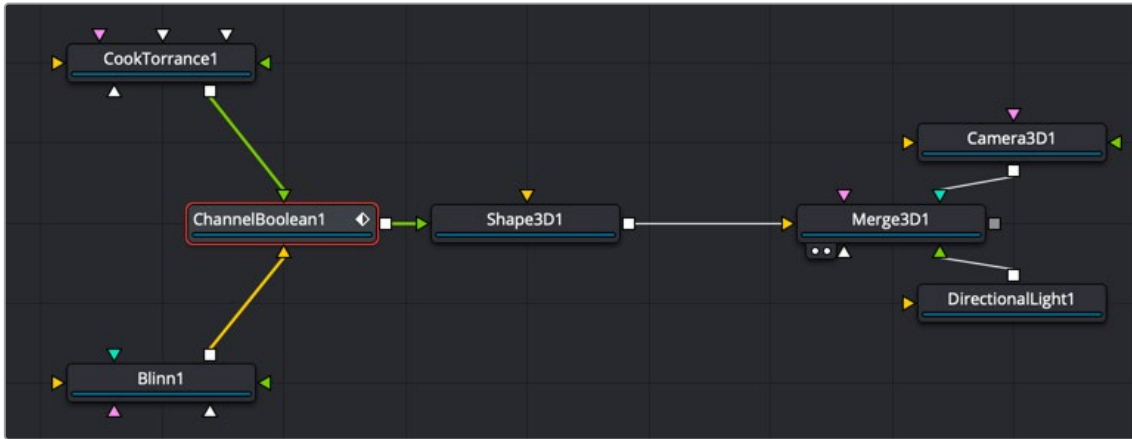
A Channel Boolean used to combine and operate on Cook Torrance and Blinn nodes

Inspector
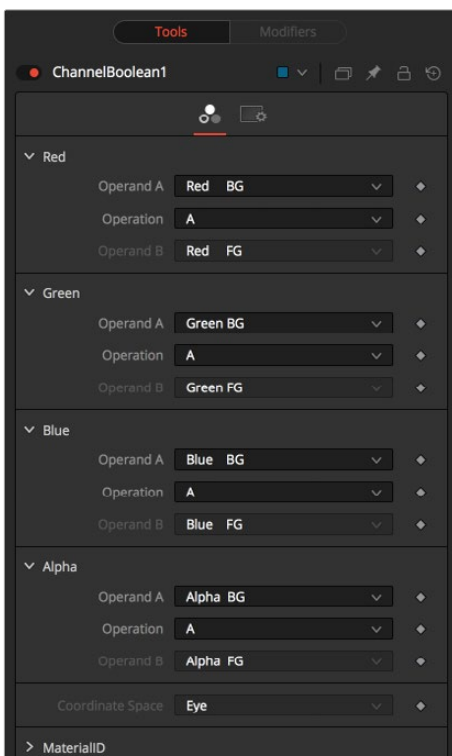
Channel Boolean controls

Controls Tab

The Controls tab includes a section for each RGBA channel. Within each channel are two input menus called Operand A and Operand B. The function performed on these two inputs is selected in the Operation menu.

Operand A/B

The Operand menus, one for each output RGBA channel, allow you to set the desired input information for the corresponding channel.

— **Red/Green/Blue/Alpha FG**

Reads the color information of the foreground material.

— **Red/Green/Blue/Alpha BG**

Reads the color information of the background material.

— **Black/White/Mid Gray**

Sets the value of the channel to 0, 0.5, or 1.

— **Hue/Lightness/Saturation FG**

Reads the color information of the foreground material, converts it into the HLS color space, and puts the selected information into the corresponding channel.

— **Hue/Lightness/Saturation BG**

Reads the color information of the background material, converts it into the HLS color space, and puts the selected information into the corresponding channel.

— **Luminance FG**

Reads the color information of the foreground material and calculates the luminance value for the channel.

— **Luminance BG**

Reads the color information of the background material and calculates the luminance value for the channel.

— **X/Y/Z Position FG**

Sets the value of the channel to the position of the pixel in 3D space. The vector information is returned in eye space.

— **U/V/W Texture FG**

Applies the texture space coordinates of the foreground material to the channels.

— **U/V/W EnvCoords FG**

Applies the environment texture space coordinates to the channels. Use it upstream of nodes modifying the environment texture coordinates like the Reflect 3D node.

— **X/Y/Z Normal**

Sets the value of the channel to the selected axis of the normal vector. The vector is returned in eye space.

Operation

Determines the Operation of how the operands are combined.

— **A:** Uses Operand A only for the output channel.

— **B:** Uses Operand B only for the output channel.

— **1-A:** Subtracts the value of Operand A from 1.

— **1-B:** Subtracts the value of Operand B from 1.

— **A+B:** Adds the value of Operand A and B.

— **A-B:** Subtracts the value of Operand B from A.

— **A*B:** Multiplies the value of both Operands.

— **A/B:** Divides the value of Operand B from A.

— **min(A,B):** Compares the values of Operands A and B and returns the smaller one.

— **max(A,B):** Compares the values of Operands A and B and returns the bigger one.

— **avg(A,B):** Returns the average value of both Operands.

Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Cook Torrance [3CT]

The Cook Torrance node

Cook Torrance Node Introduction

The Cook Torrance node is a basic illumination material that can be applied to geometry in the 3D scene. The diffuse calculation for this node is similar to that used in the basic material and the Blinn node, but the specular highlights are evaluated using an optimized Fresnel/Beckmann equation. This illumination model is primarily used for shading metal or other shiny and highly reflective surfaces.

The Cook Torrance node outputs a 3D Material that can be connected to the material inputs on any 3D geometry node.

Inputs

There are six inputs on the Cook Torrance node that accept 2D images or 3D materials. These inputs control the overall color and image used for the 3D object as well as controlling the color and texture used in the specular highlight. Each of these inputs multiplies the pixels in the texture map by the equivalently named parameters in the node itself. This provides an effective method for scaling parts of the material.

— **Diffuse Color Material:** The orange Diffuse Color material input accepts a 2D image or a 3D material to be used as overall color and texture of the object.

— **Specular Color Material:** The green Specular Color material input accepts a 2D image or a 3D material to be used as the color and texture of the specular highlight.

— **Specular Intensity Material:** The magenta Specular Intensity material input accepts a 2D image or a 3D material to alter the intensity of the specular highlight. When the input is a 2D image, the Alpha channel is used to create the map, while the color channels are discarded.

— **Specular Roughness Material:** The white Specular Roughness material input accepts a 2D image or a 3D material to be used as a map for modifying the roughness of the specular highlight. The Alpha of the texture map is multiplied by the value of the roughness control.

— **Specular Refractive Index Material:** The white Specular Refractive Index material input accepts a 2D image or a 3D material, using the RGB channels as the refraction texture.

— **Bump Map Material:** The white Bump Map material input accepts only a 3D material. Typically, you connect the texture into a Bump Map node, and then connect the Bump Map node to this input. This input uses the RGB information as texture-space normals.

Each of these inputs multiplies the pixels in the texture map by the equivalently named parameters in the node itself. This provides an effective method for scaling parts of the material.

When nodes have as many inputs as this one does, it is often difficult to make connections with any precision. Hold down the Option (macOS) or Alt (Windows) key while dragging the output from another node over the node tile, and keep holding Option or Alt when releasing the left mouse button. A small drop-down menu listing all the inputs provided by the node appears. Click on the desired input to complete the connection.

Basic Node Setup

The output of a Cook Torrance node output is connected to the material input on a 3D scene or 3D geometry node to which you want the shader applied. The Cook Torrance inputs can use images as the diffuse color material (yellow) and specular color material (green). This can result in a smooth, shiny material.

A Cook Torrance shader with diffuse and specular color materials connected

Inspector
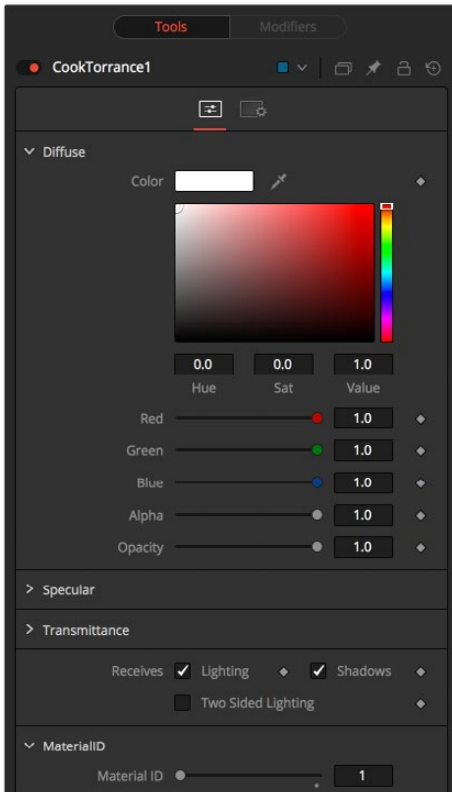
Cook Torrance controls

Controls Tab

The Controls tab contains parameters for adjusting the main color, highlight, and lighting properties of the Cook Torrance shader node.

Diffuse

Diffuse describes the base surface characteristics without any additional effects like reflections or specular highlights. Besides defining the base color of an object, the diffuse color also defines the transparency of the object. The Alpha in a diffuse texture map can be used to make portions of the surface transparent.

Diffuse Color

A material's Diffuse Color describes the base color presented by the material when it is lit indirectly or by ambient light. If a diffuse texture map is provided, then the color value provided here is multiplied by the color values in the texture.

Alpha

This slider sets the material's Alpha channel value. This affects diffuse and specular colors equally, and affects the Alpha value of the material in the rendered output. If a diffuse texture map is provided, then the Alpha value set here is multiplied by the Alpha values in the texture map.

Opacity

Reducing the material's Opacity decreases the color and Alpha values of the specular and diffuse colors equally, making the material transparent.

Specular

The parameters in the Specular section describe the look of the specular highlight of the surface. These values are evaluated in a different way for each illumination model.

Specular Color

Specular Color determines the color of light that reflects from a shiny surface. The more specular a material is, the glossier it appears. Surfaces like plastics and glass tend to have white specular highlights, whereas metallic surfaces like gold have specular highlights that inherit their color from the material color. If a specular texture map is provided, then the value provided here is multiplied by the color values from the texture.

Specular Intensity

Specular Intensity controls how strong the specular highlight is. If the specular intensity texture is provided, then this value is multiplied by the Alpha value of the texture.

Roughness

The Roughness of the specular highlight describes diffusion of the specular highlight over the surface. The greater the value, the wider the falloff, and the more brushed and metallic the surface appears. If the roughness texture map is provided, then this value is multiplied by the Alpha value from the texture.

Do Fresnel

Selecting this checkbox adds Fresnel calculations to the materials illumination model. This provides more realistic-looking metal surfaces by taking into account the refractiveness of the material.

Refractive Index

This slider appears when the Do Fresnel checkbox is selected. The Refractive Index applies only to the calculations for the highlight; it does not perform actual refraction of light through transparent surfaces. If the refractive index texture map is provided, then this value is multiplied by the Alpha value of the input.

Transmittance

Transmittance controls the way light passes through a material. For example, a solid blue sphere casts a black shadow, but one made of translucent blue plastic would cast a much lower density blue shadow.

There is a separate Opacity option. Opacity determines how transparent the actual surface is when it is rendered. Fusion allows adjusting both opacity and transmittance separately. At first, this might be a bit counterintuitive to those who are unfamiliar with 3D software. It is possible to have a surface that is fully opaque but transmits 100% of the light arriving upon it, effectively making it a luminous/emissive surface.

Attenuation

Attenuation determines how much color is passed through the object. For an object to have transmissive shadows, set the attenuation to (1, 1, 1), which means 100% of green, blue, and red light passes through the object. Setting this color to RGB (1, 0, 0) means that the material transmits 100% of the red arriving at the surface but none of the green or blue light. This can be used to create "stained glass"-styled shadows.

Alpha Detail

When the Alpha Detail slider is set to 0, the Alpha channel of the object is ignored and the entire object casts a shadow. If it is set to 1, the Alpha channel determines what portions of the object cast a shadow.

Color Detail

The Color Detail slider modulates light passing through the surface by the diffuse color + texture colors. Use this to throw a shadow that contains color details of the texture applied to the object. Increasing the slider from 0 to 1 brings in more diffuse color + texture color into the shadow. Note that the Alpha and opacity of the object are ignored when transmitting color, allowing an object with a solid Alpha to still transmit its color to the shadow.

Saturation

The Saturation slider controls the saturation of the color component transmitted to the shadow. Setting this to 0.0 results in monochrome shadows.

Receives Lighting/Shadows

These checkboxes control whether the material is affected by lighting and shadows in the scene. If turned off, the object is always fully lit and/or unshadowed.

Two-Sided Lighting

This effectively makes the surface two sided by adding a second set of normals facing the opposite direction on the backside of the surface. This is normally off to increase rendering speed, but it can be turned on for 2D surfaces or for objects that are not fully enclosed, to allow the reverse or interior surfaces to be visible as well.

Normally, in a 3D application, only the front face of a surface is visible and the back face is culled, so that if a camera were to revolve around a plane in a 3D application, when it reached the backside, the plane would become invisible. Making a plane two sided in a 3D application is equivalent to adding another plane on top of the first but rotated by 180 degrees so the normals are facing the opposite direction on the backside. Thus, when you revolve around the back, you see the second image plane, which has its normals facing the opposite way.

> **NOTE:** This can become rather confusing once you make the surface transparent, as the same rules still apply and produce a result that is counterintuitive. If you view from the frontside a transparent two-sided surface illuminated from the backside, it looks unlit.

Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Material Merge 3D [3MM]

The Material Merge node

Material Merge Node Introduction

The Material Merge node can be used to combine two separate materials together. This node can be used to composite Material nodes, combining multiple illumination materials (Blinn, Cook Torrance) with texture nodes (Bump Map, Reflection) to create complex shader networks.

The node also provides a mechanism for assigning a new material identifier to the combined material.

Inputs

The Material Merge node includes two inputs for the two materials you want to combine.

— **Background Material:** The orange Background material input accepts a 2D image or a 3D material to be used as the background material.
— **Foreground Material:** The green Foreground material input accepts a 2D image or a 3D material to be used as the foreground material. A 2D image is treated as a diffuse texture map in the basic shading model.

Basic Node Setup

The output of a Material Merge node is connected to the material input on a 3D scene or 3D geometry node. The Material Merge node below is taking in a background base layer from the Blinn shader and combining it with a more textured bump map layer.
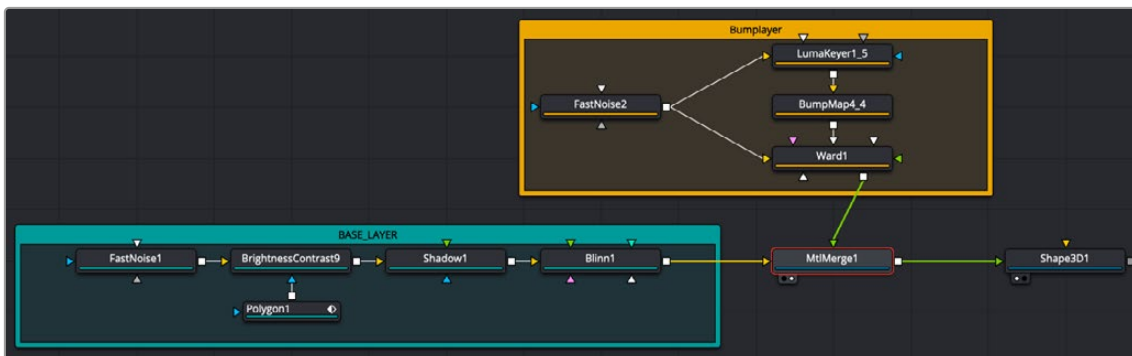
A Material Merge node combining a Blinn-based shader (teal underlay) and a Ward-based shader (orange underlay)

Inspector
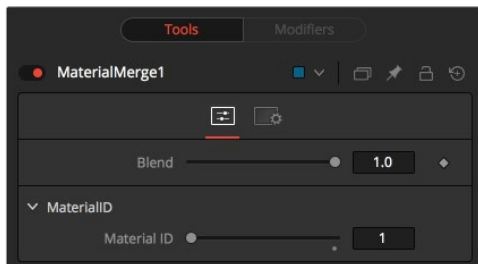
Material Merge controls

Controls Tab

The Controls tab includes a single slider for blending the two materials together.

Blend

The Blend behavior of the Material Merge is similar to the Dissolve (DX) node for images. The two materials/textures are mixed using the value of the slider to determine the percentage each input contributes. While the background and foreground inputs can be a 2D image instead of a material, the output of this node is always a material.

Unlike the 2D Dissolve node, both foreground and background inputs are required.

Material ID

This slider sets the numeric identifier assigned to the resulting material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Phong [3Ph]

The Phong node

Phong Node Introduction

The Phong node is a basic illumination material that can be applied to geometry in the 3D scene. It describes how the object responds to light and provides multiple texture map inputs to allow fine control over the diffuse, specular, and bump map components of the material.

While producing a highlight similar to that produced by the Blinn model, it is more commonly used for shiny/polished plastic surfaces.

Inputs

There are five inputs on the Phong node that accept 2D images or 3D materials. These inputs control the overall color and image used for the 3D object as well as controlling the color and texture used in the specular highlight. Each of these inputs multiplies the pixels in the texture map by the equivalently named parameters in the node itself. This provides an effective method for scaling parts of the material.

— **Diffuse Material:** The orange Diffuse material input accepts a 2D image or a 3D material to be used as a main color and texture of the object.

— **Specular Color Material:** The green Specular Color material input accepts a 2D image or a 3D material to be used as a highlight color and texture of the object.

— **Specular Intensity Material:** The magenta Specular Intensity material input accepts a 2D image or a 3D material to be used as an intensity map for the material's highlights. When the input is a 2D image, the Alpha channel is used to create the map, while the color channels are discarded.

— **Specular Exponent Material:** The teal Specular Exponent material input accepts a 2D image or a 3D material to be used as a falloff map for the material's specular highlights. When the input is a 2D image, the Alpha channel is used to create the map, while the color channels are discarded.

— **Bump Map Material:** The white Bump Map texture input accepts only a 3D material. Typically, you connect the texture into a Bump Map node, and then connect the Bump Map node to this input. This input uses the RGB information as texture-space normals.

When nodes have as many inputs as this one does, it is often difficult to make connections with any precision. Hold down the Option or Alt key while dragging the output from another node over the node tile, and keep holding Option or Alt when releasing the left mouse button. A small drop-down menu listing all the inputs provided by the node appears. Click on the desired input to complete the connection.

Basic Node Setup

The output of a Phong node is connected to the material input on a 3D scene or 3D geometry node. The Phong node below is taking in a base Color Diffuse input from the Fast Noise node and a bump map texture also generated from a Fast Noise node.
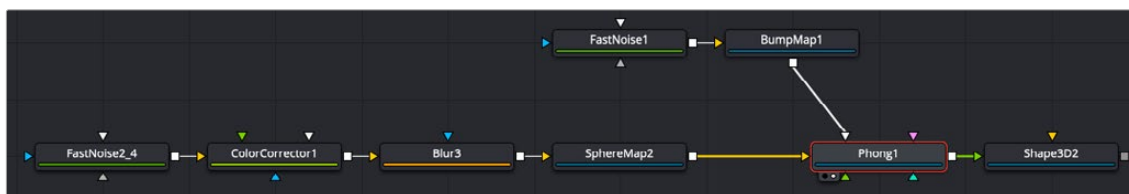
A Phong node with a diffuse color and Bump Map input

Inspector
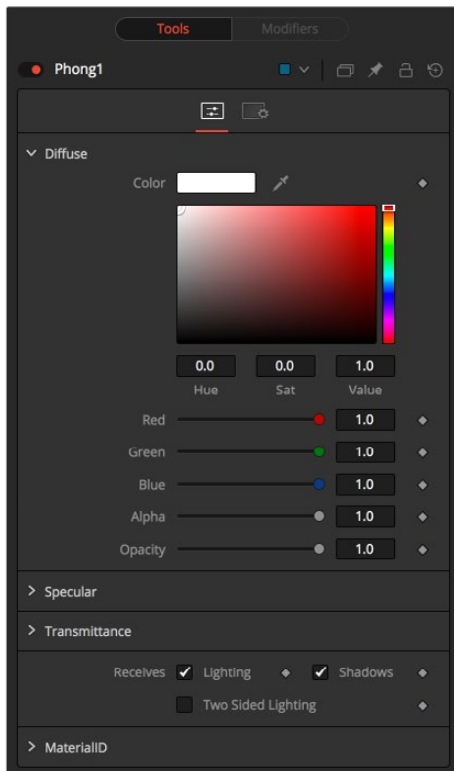
Phong controls

Controls Tab

The Controls tab contains parameters for adjusting the main color, highlight, and lighting properties of the Phong shader node.

Diffuse

Diffuse describes the base surface characteristics without any additional effects like reflections or specular highlights. Besides defining the base color of an object, the diffuse color also defines the transparency of the object.

The Alpha in a diffuse texture map can be used to make portions of the surface transparent.

Diffuse Color

A material's Diffuse Color describes the base color presented by the material when it is lit indirectly or by ambient light. If a diffuse texture map is provided, then the color value provided here is multiplied by the color values in the texture.

Alpha

This slider sets the material's Alpha channel value. This affects diffuse and specular colors equally and affects the Alpha value of the material in the rendered output. If a diffuse texture map is provided, then the Alpha value set here is multiplied by the Alpha values in the texture map.

Opacity

Reducing the material's Opacity decreases the color and Alpha values of the specular and diffuse colors equally, making the material transparent.

Specular

The parameters in the Specular section describe the look of the specular highlight of the surface. These values are evaluated in a different way for each illumination model.

Specular Color

Specular Color determines the color of light that reflects from a shiny surface. The more specular a material is, the glossier it appears. Surfaces like plastics and glass tend to have white specular highlights, whereas metallic surfaces like gold have specular highlights that inherit their color from the material color. If a specular texture map is provided, then the value provided here is multiplied by the color values from the texture.

Specular Intensity

Specular Intensity controls how strong the specular highlight is. If the specular intensity texture is provided, then this value is multiplied by the Alpha value of the texture.

Specular Exponent

Specular Exponent controls the falloff of the specular highlight. The greater the value, the sharper the falloff, and the smoother and glossier the material appears. If the specular exponent texture is provided, then this value is multiplied by the Alpha value of the texture map.

Transmittance

Transmittance controls the way light passes through a material. For example, a solid blue sphere casts a black shadow, but one made of translucent blue plastic would cast a much lower density blue shadow.

There is a separate Opacity option. Opacity determines how transparent the actual surface is when it is rendered. Fusion allows adjusting both opacity and transmittance separately. At first, this might be a bit counterintuitive to those who are unfamiliar with 3D software. It is possible to have a surface that is fully opaque but transmits 100% of the light arriving upon it, effectively making it a luminous/emissive surface.

Attenuation

Attenuation determines how much color is passed through the object. For an object to have transmissive shadows, set the attenuation to (1, 1, 1), which means 100% of green, blue, and red light passes through the object. Setting this color to RGB (1, 0, 0) means that the material transmits 100% of the red arriving at the surface but none of the green or blue light. This can be used to create "stained glass"-styled shadows.

Alpha Detail

When the Alpha Detail slider is set to 0, the Alpha channel of the object is ignored and the entire object casts a shadow. If it is set to 1, the Alpha channel determines what portions of the object cast a shadow.

Color Detail

The Color Detail slider modulates light passing through the surface by the diffuse color + texture colors. Use this to throw a shadow that contains color details of the texture applied to the object. Increasing the slider from 0 to 1 brings in more diffuse color + texture color into the shadow. Note that the Alpha and opacity of the object are ignored when transmitting color, allowing an object with a solid Alpha to still transmit its color to the shadow.

Saturation

The Saturation slider controls the saturation of the color component transmitted to the shadow. Setting this to 0.0 results in monochrome shadows.

Receives Lighting/Shadows

These checkboxes control whether the material is affected by lighting and shadows in the scene. If turned off, the object is always fully lit and/or unshadowed.

Two-Sided Lighting

This effectively makes the surface two sided by adding a second set of normals facing the opposite direction on the backside of the surface. This is normally off to increase rendering speed, but it can be turned on for 2D surfaces or for objects that are not fully enclosed, to allow the reverse or interior surfaces to be visible as well.

Normally, in a 3D application, only the front face of a surface is visible and the back face is culled, so that if a camera were to revolve around a plane in a 3D application, when it reached the backside, the plane would become invisible. Making a plane two sided in a 3D application is equivalent to adding another plane on top of the first but rotated by 180 degrees so the normals are facing the opposite direction on the backside. Thus, when you revolve around the back, you see the second image plane, which has its normals facing the opposite way.

Fusion does exactly the same thing as 3D applications when you make a surface two sided. The confusion about what two-sided lighting does arises because Fusion does not cull back-facing polygons by default. If you revolve around a one-sided plane in Fusion, you still see it from the backside (but you are seeing the frontside duplicated through to the backside as if it were transparent). Making the plane two sided effectively adds a second set of normals to the backside of the plane.

> **NOTE:** This can become rather confusing once you make the surface transparent, as the same rules still apply and produce a result that is counterintuitive. If you view from the frontside a transparent two-sided surface illuminated from the backside, it looks unlit.

Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Reflect [3Rr]

The Reflect node

Reflect Node Introduction

The Reflect node is used to add environment map reflections and refractions to materials.

Control is offered over the face on and glancing strength, falloff, per channel refraction indexes, and tinting. Several texture map inputs can modify the behavior of each parameter.

Environment mapping is an approximation that assumes an object's environment is infinitely distant from the object. It's best to picture this as a cube or sphere with the object at the center. Specifically, this infinite distance assumption means that objects cannot interact with themselves (e.g., the reflections on the handle of a teapot do not show the body of the teapot but rather the infinite environment map). It also means that if you use the same cube map on multiple objects in the scene, those objects do not inter-reflect each other (e.g., two neighboring objects would not reflect each other). If you want objects to reflect each other, you need to render a cube map for each object.

For more information, *see Chapter 85, "3D Compositing Basics,"* in the DaVinci Resolve Reference Manual, or Chapter 23 in the Fusion Reference Manual.

Inputs

There are five inputs on the Reflect node that accept 2D images or 3D materials. These inputs control the overall color and image used for the 3D object as well as controlling the color and texture used in the reflective highlights.

— **Background Material:** The orange Background material input accepts a 2D image or a 3D material. If a 2D image is provided, the node treats it as a diffuse texture map applied to a basic material.

— **Reflection Color Material:** The white Reflection Color material input accepts a 2D image or a 3D material. The RGB channels are used as the reflection texture, and the Alpha is ignored.

— **Reflection Intensity Material:** The white Reflection Intensity material input accepts a 2D image or a 3D material. The Alpha channel of the texture is multiplied by the intensity of the reflection.

— **Refraction Tint Material:** The white Refraction Tint material input accepts a 2D image or a 3D material. The RGB channels are used as the refraction texture.

— **Bump Map Texture:** The white Bump Map texture input accepts only a 3D material. Typically, you connect the texture into a Bump Map node, and then connect the Bump Map node to this input. This input uses the RGB information as texture-space normals.

When nodes have as many inputs and some using the same color as this one does, it is often difficult to make connections with any precision. Hold down the Option or Alt key while dragging the output from another node over the node tile, and keep holding Option or Alt when releasing the left mouse

button. A small drop-down menu listing all the inputs provided by the node appears. Click on the desired input to complete the connection.

Basic Node Setup

The Reflection node can be the main shader for an object as it is in the example below, or it can be used to feed the diffuse material input of a Ward, Blinn, Phong, or other material node. Usually, a Sphere Map node is used as the source of the Reflect node's reflection color input.
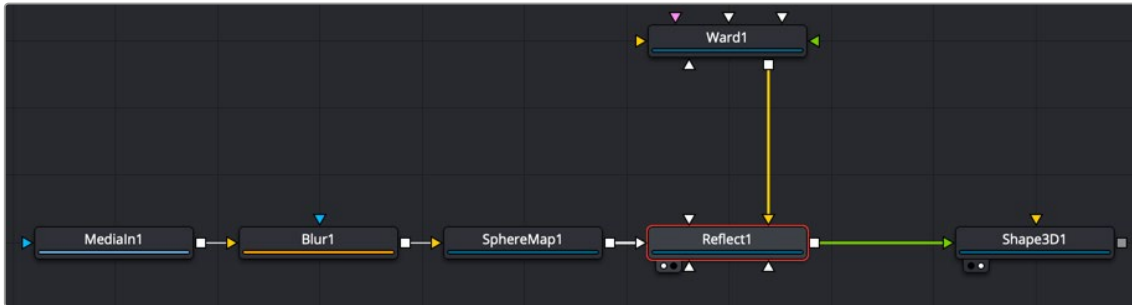
A Reflect node used to create a highly reflective surface of a Shape 3D node

Inspector
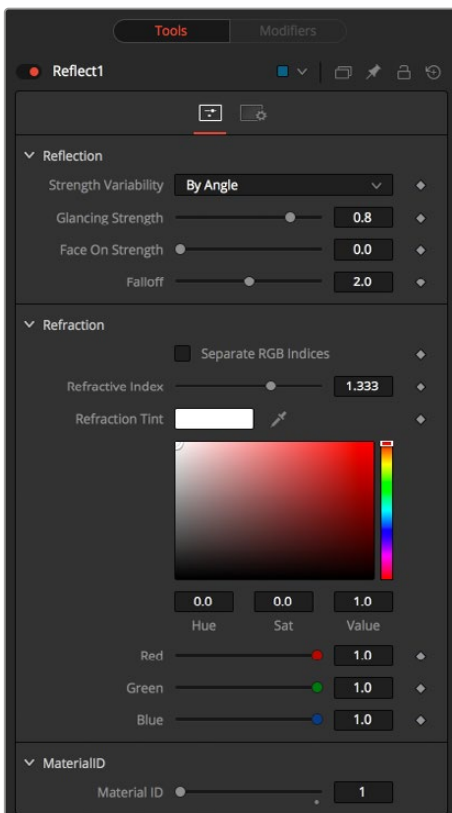
Reflect controls

Controls Tab

The Controls tab contains parameters for adjusting the reflective strength based on the orientation of the object, as well as the tint color of the Reflect shader node.

Reflection

Reflection Strength Variability

This multi-button control can be set to Constant or By Angle for varying the reflection intensity, corresponding to the relative surface orientation to the viewer. The following three controls are visible only when this control is set to By Angle.

Glancing Strength

[By Angle] Glancing Strength controls the intensity of the reflection for those areas of the geometry where the reflection faces away from the camera.

Face On Strength

[By Angle] Face On Strength controls the intensity of the reflection for those parts of the geometry that reflect directly back to the camera.

Falloff

[By Angle] Falloff controls the sharpness of the transition between the Glancing and Face On Strength regions. It can be considered similar to applying gamma correction to a gradient between the Face On and Glancing values.

Constant Strength

[Constant Angle] This control is visible only when the reflection strength variability is set to Constant. In this case, the intensity of the reflection is constant despite the incidence angle of the reflection.

Refraction

If the incoming background material has a lower opacity than 1, then it is possible to use an environment map as refraction texture, and it is possible to simulate refraction effects in transparent objects.

Separate RGB Refraction Indices

When this checkbox is enabled, the Refraction Index slider is hidden, and three sliders for adjusting the refraction index of the Red, Green, and Blue channels appear in its place. This allows simulation of the spectral refraction effects commonly seen in thick imperfect glass, for example.

Refraction Index

This slider controls how strongly the environment map is deformed when viewed through a surface. The overall deformation is based on the incidence angle. Since this is an approximation and not a simulation, the results are not intended to model real refractions accurately.

Refraction Tint

The refraction texture is multiplied by the tint color for simulating color-filtered refractions. It can be used to simulate the type of coloring found in tinted glass, as seen in many brands of beer bottles, for example.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Stereo Mix [3SMM]

The Stereo Mix node

Stereo Mix Node Overview

This node is used to swap the left and right material inputs. It is often used to output to the left and right eye of the 3D Render.

Inputs

This node has two inputs that are both required for this node to work. Both inputs accept either a 2D image or a 3D material.

— **LeftMaterial:** The orange left material input accepts a 2D image or a 3D material to be used as the material for the left eye rendering. If a 2D image is used, it is converted to a diffuse texture map using the basic material type.

— **RightMaterial:** The green right material input accepts a 2D image or a 3D material to be used as the material for the right eye rendering. If a 2D image is used, it is converted to a diffuse texture map using the basic material type.

While the inputs can be either 2D images or 3D materials, the output is always a material.

Basic Node Setup

The Stereo Mix node can be used with either stereo images or materials. The example below shows two images combined in the Stereo Mix node causing the output to be a stereo anaglyph material.

A Stereo Mix node used to combine left and right images into a single stereo material

Inspector
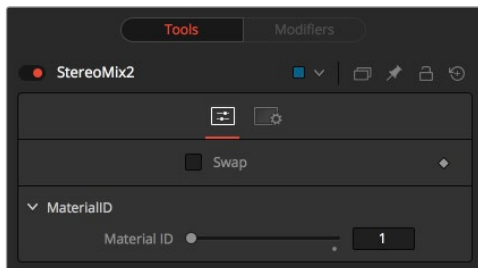
Stereo Mix controls

Controls Tab

The Controls tab contains a single switch that swaps the left and right material inputs.

Swap

This option swaps both inputs of the node.

Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Ward [3Wd]
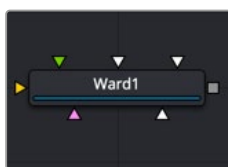
The Ward node

Ward Node Introduction

The Ward node is a basic illumination material that can be applied to geometry in the 3D scene. It describes how the object responds to light and provides multiple texture map inputs to allow fine control over the diffuse, specular, and bump map components of the material.

Specifically, the Ward node is ideal for simulating brushed metal surfaces, as the highlight can be elongated along the U or V directions of the mapping coordinates. This is known as an anisotropic highlight.

The Ward node outputs a 3D Material that can be connected to the material inputs on any 3D geometry node.

Inputs

There are six inputs on the Ward node that accept 2D images or 3D materials. These inputs control the overall color and image used for the 3D object as well as controlling the color and texture used in the specular highlight. Each of these inputs multiplies the pixels in the texture map by the equivalently named parameters in the node itself. This provides an effective method for scaling parts of the material.

— **Diffuse Material:** The orange Diffuse material input accepts a 2D image or a 3D material to be used as a main color and texture of the object.

— **Specular Color Material:** The green Specular Color material input accepts a 2D image or a 3D material to be used as a highlight color and texture of the object.

— **Specular Intensity Material:** The magenta Specular Intensity material input accepts a 2D image or a 3D material to be used as an intensity map for the material's highlights. When the input is a 2D image, the Alpha channel is used to create the map, while the color channels are discarded.

— **Spread U Material:** The white Spread U material input accepts a 2D image or a 3D material. The value of the Spread U option in the node's controls is multiplied against the pixel values in the material's Alpha channel.

— **Spread V Material:** The white Spread V material input accepts a 2D image or a 3D material. The value of the Spread V option in the node's controls is multiplied against the pixel values in the material's Alpha channel.

— **Bump Map Material:** The white Bump Map material input accepts only a 3D material. Typically, you connect the texture into a Bump Map node, and then connect the Bump Map node to this input. This input uses the RGB information as texture-space normals.

When nodes have as many inputs and some using the same color as this one does, it is often difficult to make connections with any precision. Hold down the Option or Alt key while dragging the output from another node over the node tile, and keep holding Option or Alt when releasing the left mouse button. A small drop-down menu listing all the inputs provided by the node appears. Click on the desired input to complete the connection.

Basic Node Setup

The Ward node is used to make a shiny glass surface and replace the 3D text material in the example below. A diffuse color material comes from Reflect node, and the specular color is altered by a gradient color Fast Noise node.
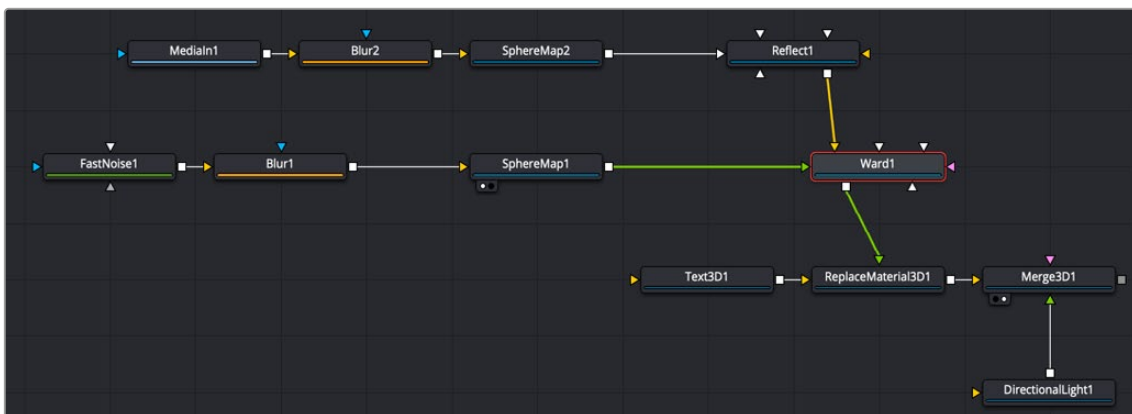
A Ward node used with a diffuse connection and specular color connection

Inspector
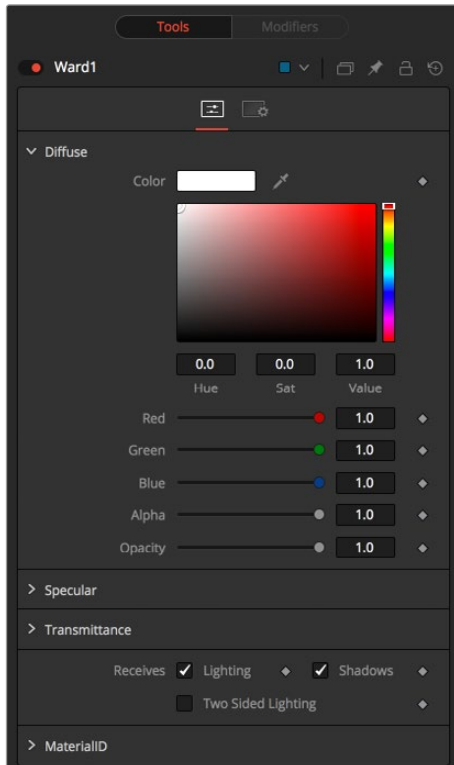
Ward controls

Controls Tab

The Controls tab contains parameters for adjusting the main color, highlight, and lighting properties of the Ward shader node.

Diffuse

Diffuse describes the base surface characteristics without any additional effects like reflections or specular highlights. Besides defining the base color of an object, the diffuse color also defines the transparency of the object. The Alpha in a diffuse texture map can be used to make portions of the surface transparent.

Diffuse Color

A material's Diffuse Color describes the base color presented by the material when it is lit indirectly or by ambient light. If a diffuse texture map is provided, then the color value provided here is multiplied by the color values in the texture.

Alpha

This slider sets the material's Alpha channel value. This affects diffuse and specular colors equally and affects the Alpha value of the material in the rendered output. If a diffuse texture map is provided, then the Alpha value set here is multiplied by the Alpha values in the texture map.

Opacity

Reducing the material's Opacity decreases the color and Alpha values of the specular and diffuse colors equally, making the material transparent.

Specular

The parameters in the Specular section describe the look of the specular highlight of the surface. These values are evaluated in a different way for each illumination model.

Specular Color

Specular Color determines the color of light that reflects from a shiny surface. The more specular a material is, the glossier it appears. Surfaces like plastics and glass tend to have white specular highlights, whereas metallic surfaces like gold have specular highlights that inherit their color from the material color. If a specular texture map is provided, then the value provided here is multiplied by the color values from the texture.

Specular Intensity

Specular Intensity controls how strong the specular highlight is. If the specular intensity texture is provided, then this value is multiplied by the Alpha value of the texture.

Spread U

Spread U controls the falloff of the specular highlight along the U-axis in the UV map of the object. The smaller the value, the sharper the falloff, and the smoother and glossier the material appears in this direction. If the Spread U texture is provided, then this value is multiplied by the Alpha value of the texture.

Spread V

Spread V controls the falloff of the specular highlight along the V-axis in the UV map of the object. The smaller the value, the sharper the falloff, and the smoother and glossier the material appear in this direction. If the Spread V texture is provided, then this value is multiplied by the Alpha value of the texture.

Transmittance

Transmittance controls the way light passes through a material. For example, a solid blue sphere casts a black shadow, but one made of translucent blue plastic would cast a much lower density blue shadow.

There is a separate Opacity option. Opacity determines how transparent the actual surface is when it is rendered. Fusion allows adjusting both opacity and transmittance separately. At first, this might be a bit counterintuitive to those who are unfamiliar with 3D software. It is possible to have a surface that is fully opaque but transmits 100% of the light arriving upon it, effectively making it a luminous/emissive surface.

Attenuation

Attenuation determines how much color is passed through the object. For an object to have transmissive shadows, set the attenuation to (1, 1, 1), which means 100% of green, blue, and red light passes through the object. Setting this color to RGB (1, 0, 0) means that the material transmits 100% of the red arriving at the surface but none of the green or blue light. This can be used to create "stained glass"-styled shadows.

Alpha Detail

When the Alpha Detail slider is set to 0, the Alpha channel of the object is ignored, and the entire object casts a shadow. If it is set to 1, the Alpha channel determines what portions of the object cast a shadow.

Color Detail

The Color Detail slider modulates light passing through the surface by the diffuse color + texture colors. Use this to throw a shadow that contains color details of the texture applied to the object. Increasing the slider from 0 to 1 brings in more diffuse color + texture color into the shadow. Note that the Alpha and opacity of the object are ignored when transmitting color, allowing an object with a solid Alpha to still transmit its color to the shadow.

Saturation

The Saturation slider controls the saturation of the color component transmitted to the shadow. Setting this to 0.0 results in monochrome shadows.

Receives Lighting/Shadows

These checkboxes control whether the material is affected by lighting and shadows in the scene. If turned off, the object is always fully lit and/or unshadowed.

Two-Sided Lighting

This effectively makes the surface two sided by adding a second set of normals facing the opposite direction on the backside of the surface. This is normally off to increase rendering speed, but it can be turned on for 2D surfaces or for objects that are not fully enclosed, to allow the reverse or interior surfaces to be visible as well.

Normally, in a 3D application, only the front face of a surface is visible and the back face is culled, so that if a camera were to revolve around a plane in a 3D application, when it reached the backside, the plane would become invisible. Making a plane two sided in a 3D application is equivalent to adding another plane on top of the first but rotated by 180 degrees so the normals are facing the opposite direction on the backside. Thus, when you revolve around the back, you see the second image plane, which has its normals facing the opposite way.

Fusion does exactly the same thing as 3D applications when you make a surface two sided. The confusion about what two-sided lighting does arises because Fusion does not cull back-facing polygons by default. If you revolve around a one-sided plane in Fusion you still see it from the backside (but you are seeing the frontside duplicated through to the backside as if it were transparent). Making the plane two sided effectively adds a second set of normals to the backside of the plane.

> **NOTE:** This can become rather confusing once you make the surface transparent, as the same rules still apply and produce a result that is counterintuitive. If you view from the frontside a transparent two-sided surface illuminated from the backside, it looks unlit.

Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail in the following "The Common Controls" section.

The Common Controls

Nodes that handle 3D geometry share a number of identical controls in the Inspector. This section describes controls that are common among 3D Material nodes.

Settings Tab
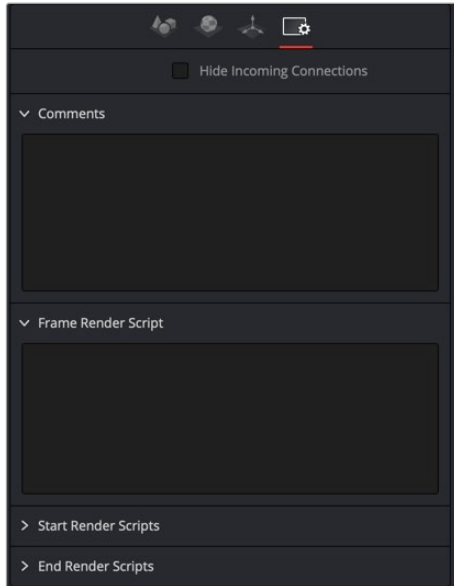
Common Settings 3D controls

Common Settings tab can be found on most tools in Fusion The following controls are specific settings for 3D nodes

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, fields for each input on a node are displayed. Dragging a connected node from the node tree into the field hide that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line reappears.

Comment Tab

The Comment tab contains a single text control that is used to add comments and notes to the tool. When a note is added to a tool, a small red dot icon appears next to the setting's tab icon, and a text bubble appears on the node. To see the note in the Node Editor, hold the mouse pointer over the node for a moment. The contents of the Comments tab can be animated over time, if required.

Scripting Tab

The Scripting tab is present on every tool in Fusion. It contains several edit boxes used to add scripts that process when the tool is rendering. For more details on the contents of this tab, please consult the scripting documentation.

3D Texture Nodes

This chapter details the 3D Texture nodes available when creating 3D composites in Fusion. The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Bump Map [3Bu]

The Bump Map node

Bump Map Node Overview

The Bump Map node is used to convert a grayscale (height map) image into a bump map and takes an input directly from a bump map created by the Create Bump Map node. The node outputs a material.

Inputs

The Bump Map node includes a single orange input for connecting a 2D image you want to use as the bump map texture, or it can accept the output of the Create Bump Map node.

— **ImageInput:** The orange Image input is used to connect a 2D RGBA image for the bump calculation or an existing bump map from the Create Bump map node.

Basic Node Setup

The Bump Map node is connected to the Bump Map material input on any one of the material shader nodes. Below, the example uses a Fast Noise node to generate an image that connects to the Bump Map node. The output of the Bump Map node connects to the Bump Map material input on a Ward node.

A Bump Map is connected to the Bump Map material input on a material node.

Inspector
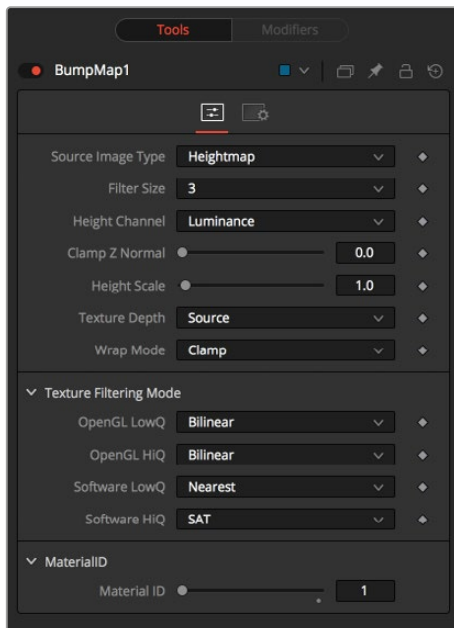
Bump Map controls

Controls Tab

The Controls tab contains all parameters for modifying the input source and the appearance of the bump map.

Source Image Type

Toggle between Height Map, which creates a bump map similar to the Create Bump Map node, and Bump Map, which expects a bump map created by the Create Bump Map node.

Filter Size

A custom filter generates the bump information. The drop-down menu sets the filter size.

Height Channel

Sets the channel from which to extract the grayscale information.

Clamp Z Normal

Clips the lower values of the blue channel in the resulting bump texture.

Height Scale

Changes the contrast of the resulting values in the bump map. Increasing this value yields a more visible bump map.

Texture Depth

Optionally converts the resulting bump map texture into the desired bit depth.

Wrap Mode

Wraps the image at the borders, so the filter produces correct result when using seamless tile textures.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Notes on Bump Maps

There is some confusion of terminology with bump mapping, depending on where you get your information. Here are Fusion conventions:

Height Map
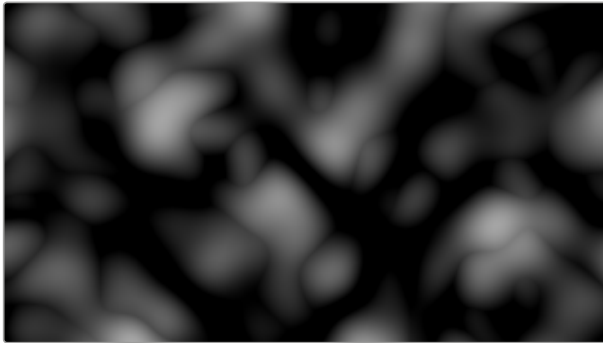
A grayscale image containing a height value per pixel
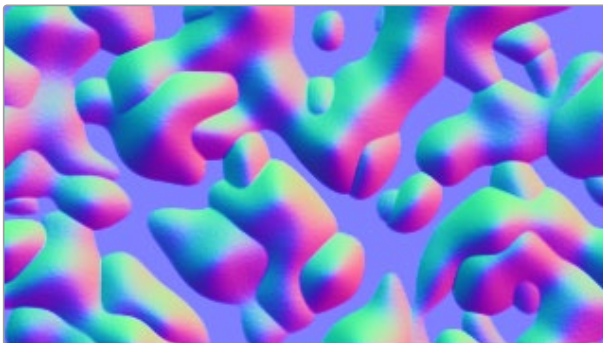
Bump Map

An image containing normals stored in the RGB channels used for modifying the existing normals (usually given in tangent space)

Normals Map

An image containing normals stored in the RGB channels used for replacing the existing normals (usually given in tangent or object space)

Catcher [3Ca]

The Catcher node

Catcher Node Overview

The Catcher material is used to "catch" texture-mode projections cast from Projector 3D and Camera 3D nodes. The intercepted projections are converted into a texture map and applied by the Catcher material to the geometry to which it is connected.

To understand the Catcher node, it helps to understand the difference between light-based projections and texture-based projections. Choosing Light from the projection mode menu on the Projector 3D or Camera 3D nodes simply adds the values of the RGB channels in the projected image to the diffuse texture of any geometry that lies within the projection cone. This makes it impossible to clip away geometry based on the Alpha channel of an image when using light mode projections.

Imagine a scenario where you want to project an image of a building onto an image plane as part of a set extension shot. You first rotoscope the image to mask out the windows. This makes it possible to see the geometry of the rooms behind the wall in the final composite. When this image is projected as light, the Alpha channel is ignored, so the masked windows remain opaque.

By connecting the Catcher to the diffuse texture map of the material applied to the image plane, and then switching the projection mode menu in the Projector 3D or Camera 3D node from Light or Ambient Light mode to Texture mode, the projected image is applied as a texture map. When using this technique for the example above, the windows would become transparent, and it would be possible to see the geometry behind the window.

The main advantages of this approach over light projection are that the Catcher can be used to project Alpha onto an object, and it doesn't require lighting to be enabled. Another advantage is that the Catcher is not restricted to the diffuse input of a material, making it possible to project specular intensity maps, or even reflection and refraction maps.

> **NOTE:** The Catcher material requires a Projector 3D or Camera 3D node in the scene, set to project an image in Texture mode on the object to which the Catcher is connected. Without a projection, or if the projection is not set to Texture mode, the Catcher simply makes the object transparent and invisible.

Inputs

The Catcher node has no inputs. The output of the node is connected to the diffuse color material input of the Blinn, Cook Torrance, or other material node applied to the 3D geometry.

Basic Node Setup

The output of a Catcher node should be connected to the material input of your 3D geometry node. A camera is set up as a proctor with an image connected to the camera's image input. When the camera is set to texture projection mode, the Catcher node is used to determine which geometry receives the texture.
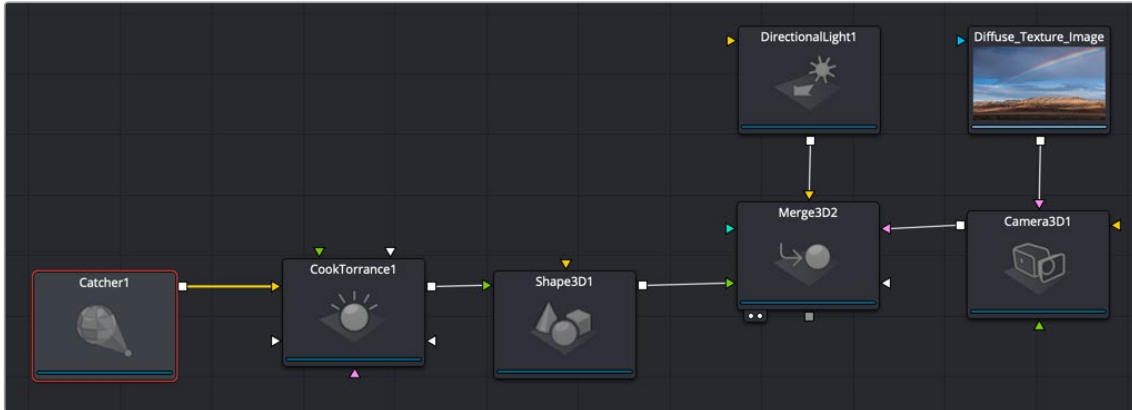
A Catcher node output is connected to the input of the geometry node that receives the texture projection

Inspector
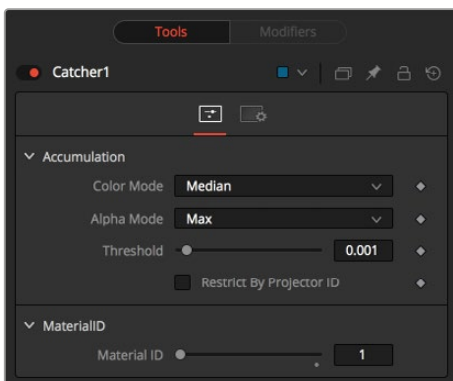
Catcher controls

Controls Tab

The Options in the Controls tab determine how the Catcher handles the accumulation of multiple projections.

Enable

Use this checkbox to enable or disable the node. This is not the same as the red switch in the upper-left corner of the Inspector. The red switch disables the tool altogether and passes the image on without any modification. The Enable checkbox is limited to the effect part of the tool. Other parts, like scripts in the Settings tab, still process as normal.

Color Mode

The Color mode menu is used to control how the Catcher combines the light from multiple projectors. It has no effect on the results when only one projector is in the scene. This control is designed to work with the software renderer in the Renderer 3D node and has no effect when using the OpenGL renderer.

Alpha Mode

The Alpha mode is used to control how the Catcher combines the Alpha channels from multiple projectors. It has no effect on the results when only one projector is in the scene. This control is designed to work with the software renderer in the Renderer 3D node and has no effect when using the OpenGL renderer.

Threshold

The Threshold can be used to exclude certain low values from the accumulation calculation. For example, when using the Median Accumulation mode, a threshold of 0.01 would exclude any pixel with a value of less than 0.01 from the median calculation.

Restrict by Projector ID

When active, the Catcher only receives light from projectors with a matching ID. Projectors with a different ID are ignored.

Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the Renderer 3D node.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

CubeMap [3Cu]
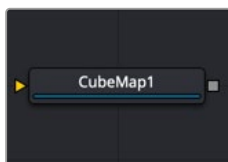
The Cube Map node

Cube Map Node Overview

The Cube Map node creates texture maps using separate images for each face of the cube. It can also extract the individual faces of the cube from a single image containing an unfolded cube in the Vertical or Horizontal Cross layouts.

A cube map is produced by mounting six cameras at 90 degrees angle of views to point up, down, left, right, front, and back.

The node provides options to set the reference coordinate system and rotation for the resulting texture map. The Cube Map node is typically used to produce environment maps for distant areas (such as skies or horizons) or reflection and refraction maps.
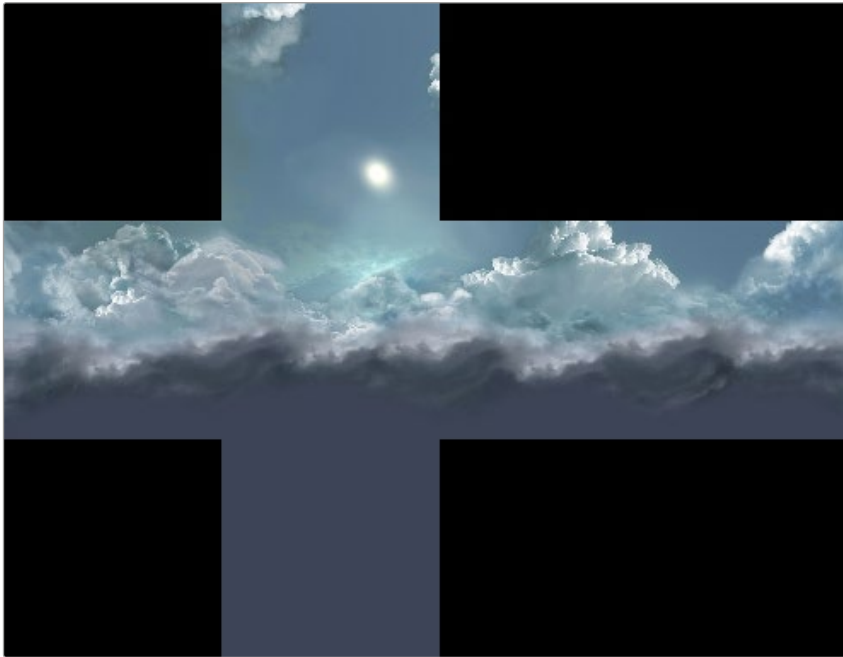
Sample cube map

Inputs

The Inputs on this node change based on the settings of the Layout menu in the Inspector. The single input uses a 2D image for the entire cube, while six inputs can handle a different 2D image for each side of a cube.

— **CrossImage:** The orange Cross Image input is visible by default or when the Layout menu in the Inspector is set to either Vertical Cross or Horizontal Cross. The input accepts a 2D image.
— **CubeMap.[DIRECTION]:** These six multi-colored inputs are visible only when the Layout menu in the Inspector is set to Separate Images. Each input accepts an image aligned to match the left, right, top, bottom, front, and back faces.

Basic Node Setup

The Cube Map node uses a vertical or horizontal cross image represented by MediaIn2 node connected into the orange cross image input. The Cube Map node is used similarly to the Sphere Map node. It creates an environment that surrounds the geometry connected to a Shader node.
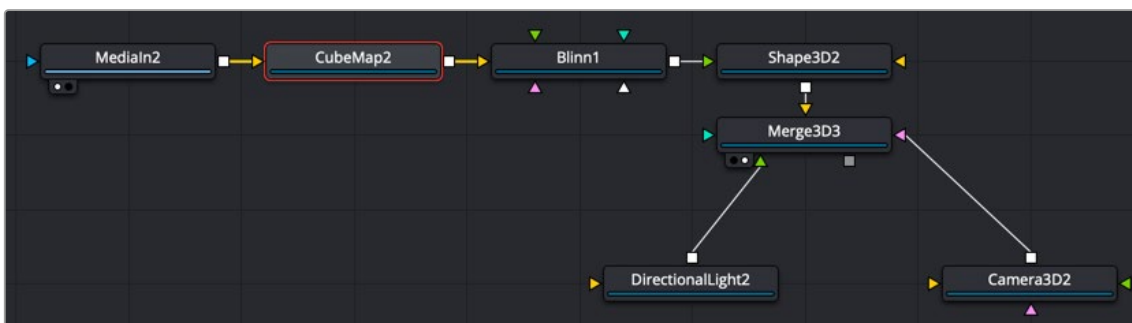
A Cube Map node receives a cross image input, creating an environment for the Shape 3D

Inspector
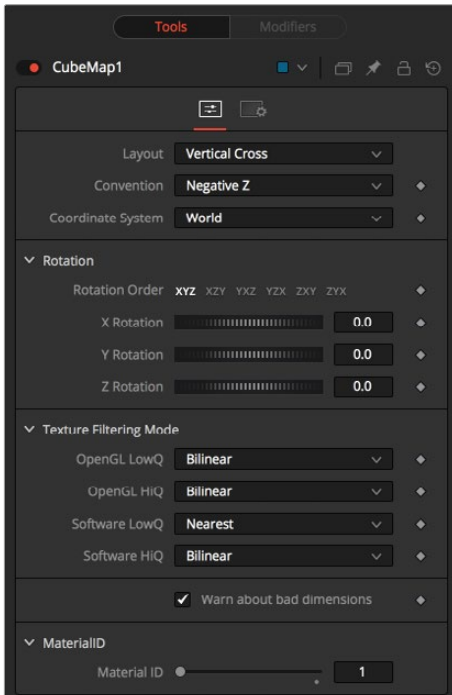
Cube Map controls

Controls Tab

Layout

The Layout menu determines the type and number of inputs for the cube map texture.
Valid options are:

— **Separate Images:** This option exposes six inputs on the node, one for each face of the cube. If the separate images are not square or not of the same size, they are rescaled into the largest 1:1 image that can contain all of them.

— **Vertical Cross:** This option exposes a single input on the node. The image should be an unwrapped texture of a cube containing all the faces organized into a Vertical Cross formation, where the height is larger than the width. If the image aspect of the cross image is not 3:4, the CubeMap node crops it down so it matches the applicable aspect ratio.

— **Horizontal Cross:** This option exposes a single input on the node. The image should be an unwrapped texture of a cube containing all the faces organized into a Horizontal Cross formation, where the width is larger than the height. If the image aspect of the cross image is not 4:3, the CubeMap node crops it down so that matches the applicable aspect ratio.

Coordinate System

The coordinate system menu sets the position values used when converting the image into a texture.

— **Model:** This option orients the texture along the object local coordinate system.

— **World:** This option orients the resulting texture using the global or world coordinate system.

— **Eye:** This option aligns the texture map to the coordinate system of the camera or viewer.

Rotation

The rotation controls are divided into buttons that select the order of rotation along each axis of the texture. For example, XYZ would apply the rotation to the X axis first, followed by the Y axis, and finally the Z axis. The other half of the rotation controls are dials that rotate the texture around its pivot point.

Warn About Bad Dimensions

Selecting this checkbox displays a warning message on the console if the dimensions of the image provided did not meet the requirements of the selected orientation mode.

Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Falloff [3Fa]

The Falloff node

Falloff Node Overview

The Falloff node blends two materials or textures together based on the incidence angle between the object to which the material is applied and the camera. This is useful when you wish to use one material for portions of the geometry that would reflect light directly back to the camera and a different material for parts that reflect light back into the scene.
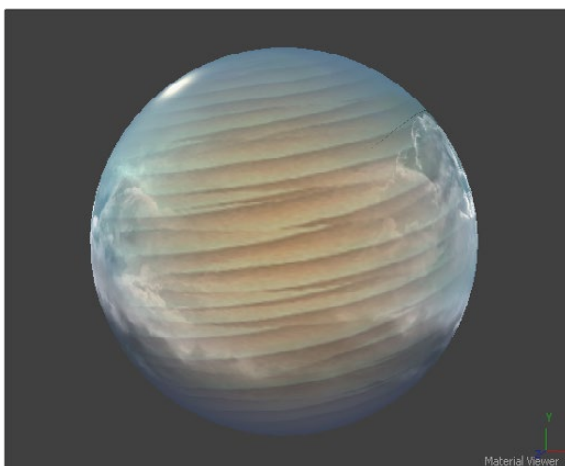
Falloff example

Inputs

The two Inputs on the Falloff node are used to connect two images or materials. One is used to reflect back at the camera, while the other reflects away from the camera and into the scene.

— **Face On Material:** The orange Face On material input accepts a 2D image or a 3D material. If a 2D image is provided, it is turned into a diffuse texture map using the basic material shader. This input is used for the material that is reflecting directly back to the camera

— **Glancing Material:** The green Glancing material input accepts a 2D image or a 3D material. If a 2D image is provided, it is turned into a diffuse texture map using the basic material shader. This input is used for the material that is reflecting away from the camera and into the scene.

While the inputs for this node can be images, the output is always a material.

Basic Node Setup

The Falloff node below is used to control the strength of the Blinn material and the Reflect material. You connect the Face On input of the Falloff node to the material you want shown for the sides of the object that face the camera and connect the Glance input to the material you want shown for the sides not directly facing the camera.
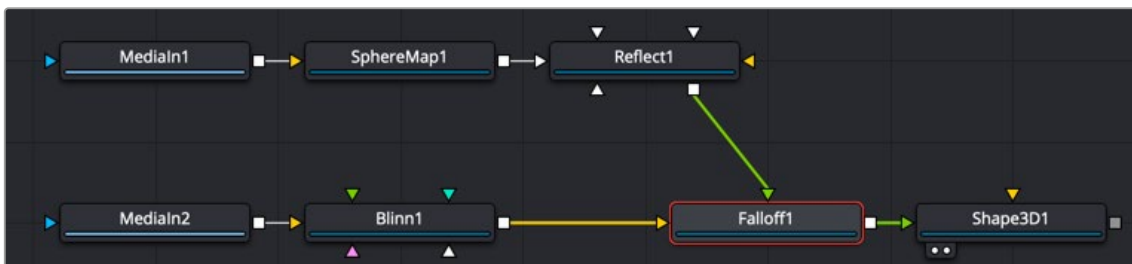
The Falloff node uses one input for the material facing the camera and one for the material not directly facing the camera.

Inspector
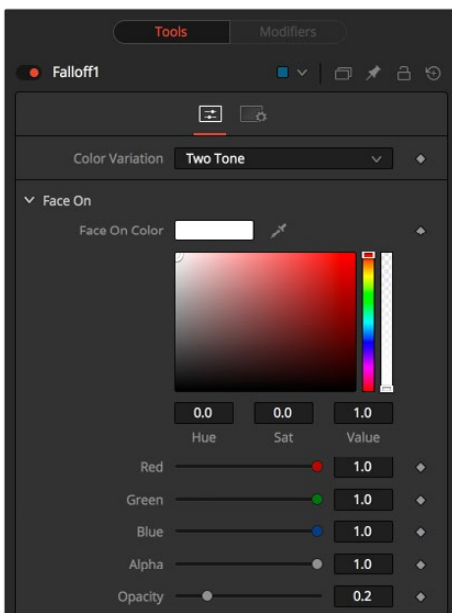
Falloff controls

Controls Tab

The parameters in the Controls tabs modify the tint and opacity of the Face On material and the Glancing material. A Falloff slider controls the blending between the two.

Color Variation

— **Two Tone:** Two regular Color controls define the colors for Glancing and Face On.

— **Gradient:** A Gradient control defines the colors for Glancing and Face On. This can be used for a multitude of effects, like creating Toon Shaders, for example.

Face On Color

The Face On Color defines the color of surface parts facing the camera. If the Face On texture map is provided, then the color value provided here is multiplied by the color values in the texture.

Reducing the material's opacity decreases the color and Alpha values of the Face On material, making the material transparent.

Glancing Color

The Glancing Color defines the color of surface parts more perpendicular to the camera. If the Glancing material port has a valid input, then this input is multiplied by this color.

Reducing the material's opacity decreases the color and Alpha values of the Glancing material, making the material transparent.

Falloff

This value controls the transition between Glancing and Face On strength. It is very similar to a gamma operation applied to a gradient, blending one value into another.
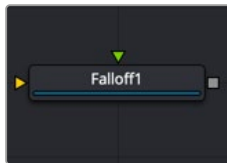
Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Fast Noise Texture [3FN]

The Fast Noise Texture node

Fast Noise Texture Node Overview

The Fast Noise Texture node is the procedural resolution-independent version of the 2D Fast Noise node. It creates a noise texture directly as a material for usage with 3D nodes. It offers a 3D volumetric mode for creating seamless textures in conjunction with nodes providing UVW texture coordinates (Similar to the UV Map 3D node set to XYZ-to-UVW or Camera).

Inputs

The Fast Noise Texture node includes an optional input that can be used to connect a 2D image or material.

— **SourceMaterial:** The Source Materials input accepts a 2D image or a 3D material. The image is then altered by the noise pattern.

Basic Node Setup

The Fast Noise Texture node below is used to generate a resolution-independent 3D texture for an FBX imported model.
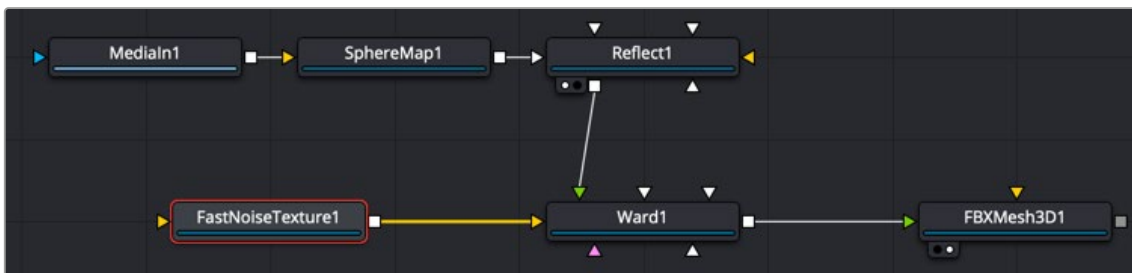
A Fast Noise Texture node generates a seamless texture, taking advantage of UVW coordinates.

Inspector
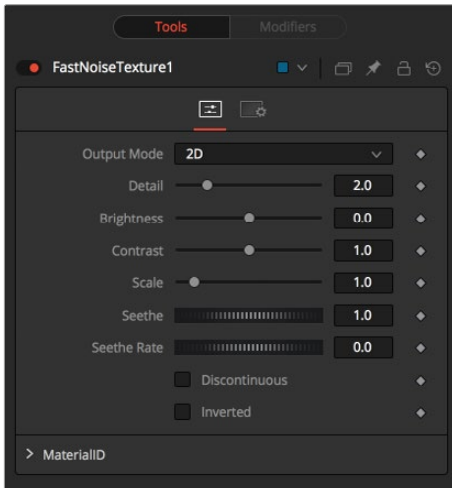
Fast Noise Texture controls

Controls Tab

The parameters of the Fast Noise Texture node control the appearance and, for 2D, the animation of the noise.

Output Mode

— **2D:** Calculates the noise texture based on 2D texture coordinates (UV). This setting allows smoothly varying the noise pattern with animation.

— **3D:** Calculates the noise texture based on 3D texture coordinates (UVW). Nodes like Shape 3D automatically provide a third texture coordinate; otherwise, a 3D texture space can be created using the UV Map node. The 3D setting does not support animation of the noise pattern.

Detail

Increase the value of this slider to produce a greater level of detail in the noise result. Larger values add more layers of increasingly detailed noise without affecting the overall pattern. High values take longer to render but can produce a more natural result (not all graphics cards support higher detail levels in hardware).

Brightness

This control adjusts the overall Brightness of the noise map.

Contrast

This control increases or decreases the overall Contrast of the noise map. It can exaggerate the effect of the noise.

Scale

The scale of the noise map can be adjusted using the Scale slider, changing it from gentle variations over the entire image to a tighter overall texture effect. This value represents the scale along the UV axis.

Scale Z

(3D only) The Scale Z value scales the noise texture along the W-axis in texture space. W represents a direction perpendicular to the UV plane for a 3D texture map.

Seethe

(2D only) The Seethe control smoothly varies the 2D noise pattern.

Seethe Rate

(2D only) As with the Seethe control above, the Seethe Rate also causes the noise map to evolve and change. The Seethe Rate defines the rate at which the noise changes each frame, causing an animated drift in the noise automatically, without the need for spline animation.

Discontinuous

Normally, the noise function interpolates between values to create a smooth continuous gradient of results. You can enable the Discontinuous checkbox to create hard discontinuity lines along some of the noise contours. The result is a dramatically different effect.

Invert

Enable the Invert checkbox to invert the noise, creating a negative image of the original pattern. This is most effective when Discontinuous is also enabled.

Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Gradient 3D [3Gd]
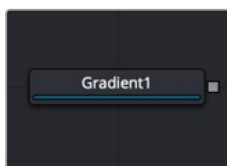
The Gradient node

Gradient Node Overview

The Gradient 3D node is used to texture objects with a variety of gradient types. It offers many of the same controls as the Background node. While it is not possible to transform the gradient directly in 3D space, it is orientable using the following nodes:

— **Texture Transform Node:** The Texture Transform node can be used to adjust the mapping per pixel.

— **UV Map Node:** The UV Map node can be used to adjust the mapping per vertex (use the XYZtoUVW mode). This has onscreen controls, so you can see what the gradient is doing. Using this node is recommended because it is faster to evaluate.

The gradient defaults to a linear gradient that goes from -1 to +1 along the Z-axis. All primitives in the Shape 3D node can output a third texture coordinate for UVW mapping.

Inputs

The Gradient node has no Inputs. The output of the node is connected to a material input on 3D geometry.

Basic Node Setup

The Gradient 3D node below is used to generate a resolution-independent 3D texture for an FBX imported model. Positioning in UVW space is easiest to do using a UV Map tool placed after the geometry.

A Gradient 3D node generates a resolution-independent
gradient texture positioned by the UV Map tool

Inspector

Gradient 3D controls

Controls Tab

The Controls tab for the Gradient node control the pattern and colors used for the gradient texture.

Gradient Type

Determines the type or pattern used for the gradient.

— **Linear:** A simple linear gradient.

— **Reflect:** Based on the Linear mode, this gradient is mirrored at the middle of the textured range.

— **Square:** The gradient is applied using a square pattern.

— **Cross:** Similar to the Reflect mode, but Cross uses two axes to apply the gradient.

— **Radial:** The Radial mode uses a circular pattern to apply the gradient.
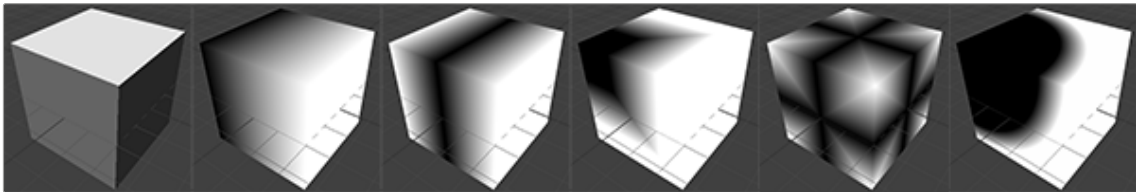
Gradient 3D modes

Gradient Bar

The Gradient control consists of a bar where it is possible to add, modify, and remove color stops of the gradient. Each triangular color stop on the Gradient bar represents a color in the gradient. It is possible to animate the color as well as the position of the point. Furthermore, a From Image modifier can be applied to the gradient to evaluate it from an image.

Interpolation Space

The gradient is linearly interpolated from point to point in RGB color space by default. This can sometimes lead to unwanted colors. Choosing another color space may provide a better result.

Scale

Allows sizing of the gradient.

Offset

Allows panning through the gradient.

Repeat

Defines how the left and right borders of the gradient are treated.

Gradients set to Once, Repeat, and Ping Pong from top to bottom, respectively, and shifting the gradient to the left

— **Once:** When using the Gradient Offset control to shift the gradient, the border colors keep their values. Shifting the default gradient to the left results in a white border on the left, while shifting it to the right results in a black border on the right.

— **Repeat:** When using the Gradient Offset control to shift the gradient, the border colors wrap around. Shifting the default gradient to the left results in a sharp jump from white to black, while shifting it to the right results in a sharp jump from black to white.

— **Ping Pong:** When using the Gradient Offset control to shift the gradient, the border colors ping-pong back and forth. Shifting the default gradient to the left results in the edge fading from white back to black, while shifting it to the right results in the edge fading from black back to white.

Sub Pixel

Determines the accuracy with which the gradient is created.

Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Sphere Map [3SpM]

The SphereMap node

Sphere Map Node Overview

The Sphere Map node can be used to create simulated environment mapping, also called reflection mapping. Ray trace rendering a reflective scene can be very time consuming, but sphere map-based reflection mapping can generate 360-degree reflections faster with little loss of accuracy. For example, when creating a reflective environment, a sphere map is created, large enough to surround the 3D object in your scene. The sphere is mapped with the environment you want reflected and connected to the Reflection Color input on a Reflect node.

Inputs

The single image input on the Sphere Map node accepts a 2D image texture in an equirectangular format (where the X-axis represents 0–360 degrees longitude, and the Y-axis represents –90 to +90 degrees latitude.)

— **ImageInput:** The orange Image input accepts a 2D RGBA image. Preferably, this is an equirectangular image that shows the entire vertical and horizontal angle of view up to 360 degrees.

Basic Node Setup

The Sphere Map node below is mapped with a spherical image to generate the environment reflected on the Shape 3D. It is connected to the Reflection Color input on a Reflect node.

A Sphere Map node generates a reflective environment when connected to a Reflect node Reflection Color input.

Inspector
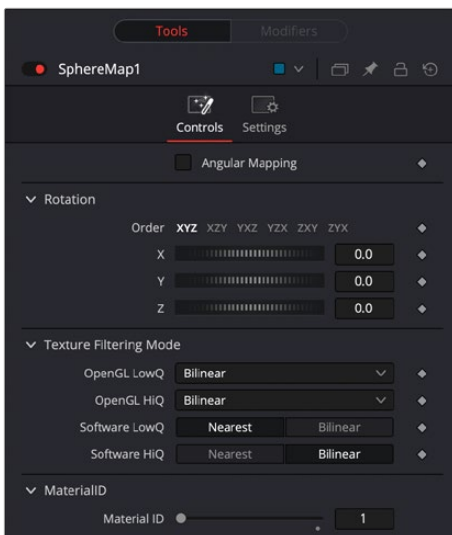
Sphere Map controls

Controls Tab

The Controls tab in the Inspector modifies the mapping of the image input to the sphere map.

Angular Mapping

Adjusts the texture coordinate mapping so the poles are less squashed and areas in the texture get mapped to equal areas on the sphere. It turns the mapping of the latitude lines from a hemispherical fisheye to an angular fisheye. This mapping attempts to preserve area and makes it easier to paint on or modify a sphere map since the image is not as compressed at the poles.

Rotation

Offers controls to rotate the texture map.

Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

The node expects an image with an aspect ratio of 2:1. Otherwise, the image is clamped according to the following rules:

— **2 * width > height:** The width is fitted onto the sphere, and the poles display clamped edges.

— **2 * width < height:** The height is fitted onto the sphere, and there is clamping about the 0-degree longitude line.

Common Controls

Settings tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Sphere Map vs. Connecting the Texture to a Sphere Directly

You can connect an equirectangular texture map directly to a sphere instead of piping it through the Sphere Map node first. This results in a different rendering if you set the start/end angle and latitude to less than 360°/180°. In the first case, the texture is squashed. When using the Sphere Map node, the texture is cropped. Compare:
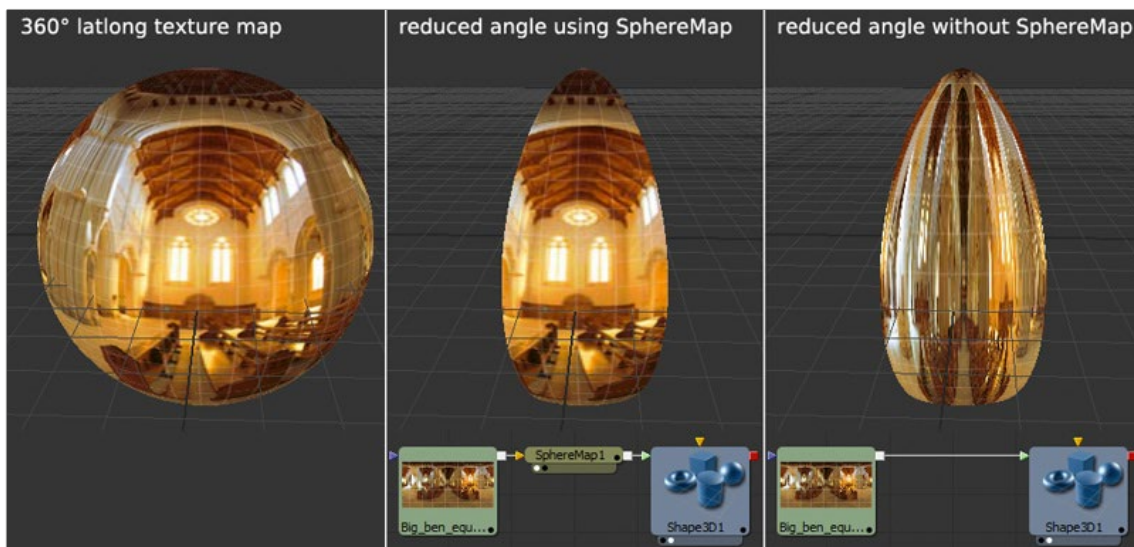
Spherical mapping differences

> **NOTE:** If you pipe the texture directly into the sphere, it is also mirrored horizontally. You can change this by using a Transform node first.

Texture 2D [3Tx]
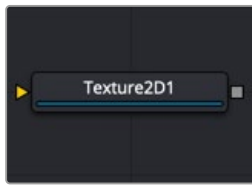
The Texture 2D node

Texture Node Overview

The Texture 2D node sets metadata of an image being used for a texture map. By default, an image will be (0,0) to (1,1) UV, but that can be changed. The Texture node relies on the presence of U and V Map channels in 3D rendered images. If these channels are not present, this node has no effect.

> **NOTE:** Background pixels may have U and V values of 0.0, which set those pixels to the color of the texture's corner pixel. To restrict texturing to specific objects, use an effect mask based on the Alpha of the object, or its Object or Material ID channel. For more information, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Inputs

— **Image Input:** The orange image input expects a 2D image.

Basic Node Setup

The Texture 2D node below takes a 2D gradient from the Background node and sets the UV metadata for it. The texture is then applied to the FBX geometry based on that metadata. If you have the option to use the UV Map tool, it is recommended because it may be faster and has onscreen controls.
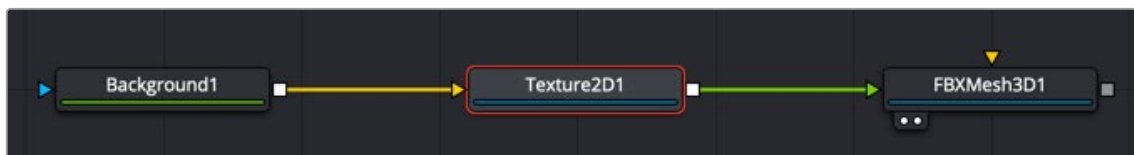
A Texture 2D node is used to set the 3D texture metadata for the input image.

Inspector

Texture 2D controls

Controls Tab

The Controls tab of the Inspector includes the following options.

U/V Offset

These sliders can be used to offset the texture along the U and V coordinates.

U/V Scale

These sliders can be used to scale the texture along the U and V coordinates.

Wrap Mode

If a texture is transformed in the texture space (using the controls below or the UV Map node), then it's possible that areas beyond the image borders will be mapped on the object. The Wrap Mode determines how the image is applied in these areas.

— **Wrap:** This wraps the edges of the image around the borders of the image.

— **Clamp:** The color at the edges of the images is used for texturing. This mode is similar to the Duplicate mode in the Transform node.

— **Black:** The image is clipped along its edges. A black color with Alpha = 0 is used instead.

— **Mirror:** The image is mirrored in both X and Y.

Texture Filtering Mode

The texture can be filtered differently depending on whether you are using the Software Renderer or OpenGL renderer in the Renderer 3D node. Within the two render engines, you can choose between high-quality anti-aliasing or low quality. The texture filtering mode provides different filtering options for the two render engines and the two anti-aliasing settings.

— **Nearest:** The simplest filtering technique is very fast but can cause artifacts when scaling textures.

— **Bilinear:** A standard isotropic filtering technique for scaling textures into multiple resolutions. Works well for magnification of textures.

— **Trilinear:** An extension of Bilinear filtering. Trilinear tends to be a better option when scaling down textures

— **Anisotropic:** The highest-quality filtering method that takes the camera orientation and polygon perspective into account.

— **SAT:** SAT (Summed Area Table) is a method of performing high-quality filtering, but it can require more memory than other options. Works very well on smaller bitmaps.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Texture Transform [3TT]
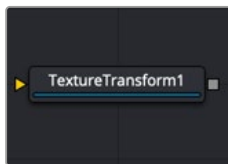
The Texture Transform node

Texture Transform Node Overview

The Texture Transform node can be used to translate, rotate, and scale the UVW texture coordinates of a 3D object. While the input can also be an image, the output is always a material.

Inputs

The Texture Transform node includes a single input that is used to connect the image or material you want to transform.

— **Material Input:** The orange Material input accepts a 2D image or 3D material whose texture coordinates are transformed using the controls in the Inspector.

Basic Node Setup

The Texture Transform node below is used to take in a 2D image, transform it, and output a material to be used on 3D geometry.
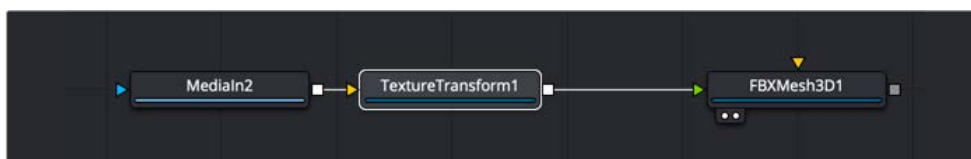
A Texture Transform node transforms a texture applied to 3D geometry.

Inspector
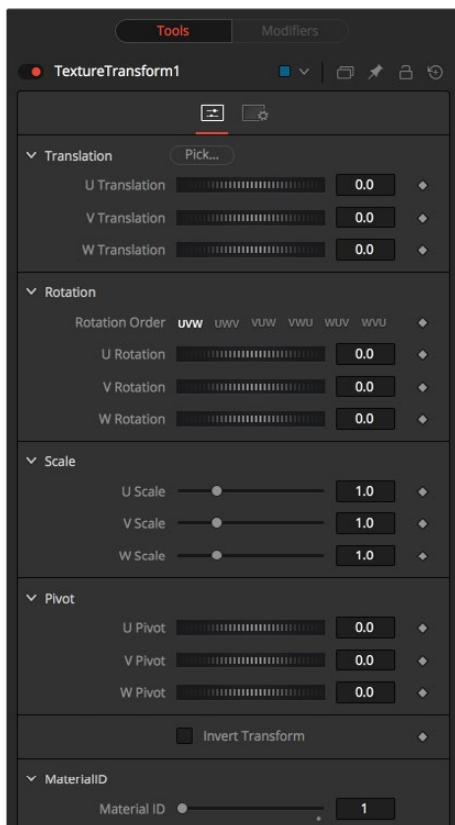
Texture Transform controls

> **NOTE:** Not all Wrap modes are supported by all graphics cards.

Controls Tab

The Controls tab for the Texture Transform node includes many common transform controls that are used to transform the texture using UVW coordinates.

Translation

The U, V, W translation sliders shift the texture along U, V, and W axes.

Rotation

Rotation Order buttons set the order in which the rotation is applied. In conjunction with the buttons, the UVW dials define the rotation around the UVW axes.

Scale

U, V, W sliders scale the texture along the UVW axes.

Pivot

U, V, W Pivot sets the reference point for rotation and scaling.

Material ID

This slider sets the numeric identifier assigned to this material. This value is rendered into the MatID auxiliary channel if the corresponding option is enabled in the renderer.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other 3D nodes. These common controls are described in the following "The Common Controls" section.

The Common Controls

Nodes that handle 3D geometry share a number of identical controls in the Inspector. This section describes controls that are common among 3D Texture nodes.

Settings Tab

Common Settings 3D controls

The Common Settings tab can be found on most tools in Fusion. The following controls are specific settings for 3D nodes.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, fields for each input on a node are displayed. Dragging a connected node from the node tree into the field hides that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line reappears.

Comment Tab

The Comment tab contains a single text control that is used to add comments and notes to the tool. When a note is added to a tool, a small red dot icon appears next to the setting's tab icon, and a text bubble appears on the node. To see the note in the Node Editor, hold the mouse pointer over the node for a moment. The contents of the Comments tab can be animated over ime, if required.

Scripting Tab

The Scripting tab is present on every tool in Fusion. It contains several edit boxes used to add scripts that process when the tool is rendering. For more details on the contents of this tab, please consult the scripting documentation.

Blur Nodes

This chapter details the Blur nodes available in Fusion. The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Blur [Blur]

The Blur node

Blur Node Introduction

The Blur node does exactly what its name implies – it blurs the input image. This is one of the most commonly used image-processing operations.

Inputs

The two inputs on the Blur node are used to connect a 2D image and an effect mask that can be used to limit the blurred area.

— **Input:** The orange input is used for the primary 2D image that is blurred.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the blur to only those pixels within the mask. An effect mask is applied to the tool after the tool is processed.

Basic Node Setup

The Blur node, like many 2D image-processing nodes, receives a 2D image like the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.
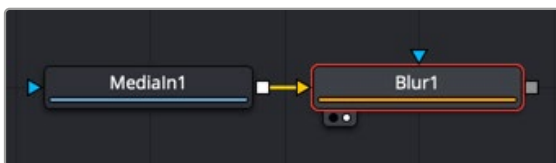
A Blur node applied to a MediaIn1 node

Inspector
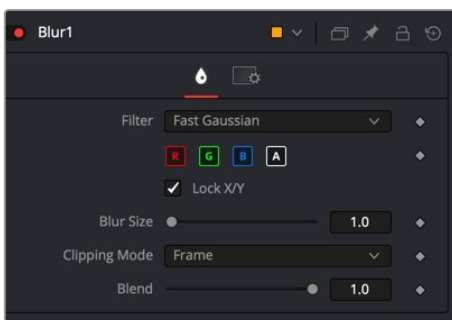
Blur controls

NOTE: Since a perfect Gaussian filter would require examining an infinite number of pixels, all practical Gaussians are, of necessity, approximations. The algorithm Fusion uses is a highly-optimized approach that has many strengths, but can create visible ringing around the edges in certain extreme cases. This ringing appears only when blurring float-depth images and is normally far below the limits of visibility, especially in final renders or HiQ mode, but may appear in subsequent processing. If you experience this, selecting the Multi-box filter may be a good choice.

Controls Tab

The Controls tab contains the primary controls necessary for customizing the blur operation, including five filter algorithms.

Filter

The Filter menu is where you select the type of filter used to create the blur.

— **Box Blur:** This option is faster than the Gaussian blur but produces a lower-quality result.

— **Bartlett:** This option is a more subtle, anti-aliased blur filter.

— **Multi-box:** Multi-box uses a Box filter layered in multiple passes to approximate a Gaussian shape. With a moderate number of passes (e.g., four), a high-quality blur can be obtained, often faster than the Gaussian filter and without any ringing.

— **Gaussian:** Gaussian applies a smooth, symmetrical blur filter, using a sophisticated constant-time Gaussian approximation algorithm.

— **Fast Gaussian:** Gaussian applies a smooth, symmetrical blur filter, using a sophisticated constant-time Gaussian approximation algorithm. This mode is the default filter method.

Color Channels (RGBA)

The filter defaults to operating on R, G, B, and A channels. Selective channel filtering is possible by clicking each channel button to make them active or inactive.

NOTE: This is not the same as the RGBA checkboxes found under the common controls. The node takes these selections into account before it processes the image, so deselecting a channel causes the node to skip that channel when processing, speeding up the rendering of the effect. In contrast, the channel controls under the Common Controls tab are applied after the node has processed.

Lock X/Y

Locks the X and Y Blur sliders together for symmetrical blurring. This is enabled by default.

Blur Size

Sets the amount of blur applied to the image. When the Lock X and Y control is deselected, independent control over each axis is provided.

Clipping Mode

This option determines how edges are handled when performing domain-of-definition rendering. This is profoundly important for nodes like Blur, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that would normally be outside the upstream DoD is treated as black/transparent.

Blend

The Blend slider determines the percentage of the affected image that is mixed with original image. It blends in more of the original image as the value gets closer to 0.

This control is a cloned instance of the Blend slider in the Common Controls tab. Changes made to this control are simultaneously made to the one in the common controls.

Examples

Following is a comparison of Blur filters visualized as "cross-sections" of a filtered edge. As you can see, Box creates a linear ramp, while Bartlett creates a somewhat smoother ramp. Multi-box and Gaussian are indistinguishable unless you zoom in really close on the slopes. They both lead to even smoother ramps, but as mentioned above, Gaussian overshoots slightly and may lead to negative values if used on floating-point images.

Blur filters visualized as "cross sections" of a filtered edge

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Blur nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Defocus [Dfo]
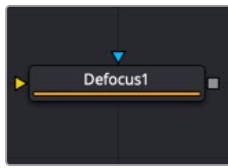
The Defocus node

Defocus Node Introduction

The Defocus node simulates the effects of an out-of-focus camera lens, including blooming and image flaring. It provides a fast Gaussian mode, as well as a more realistic but slower Lens mode.

Inputs

The two inputs on the Defocus node are for connecting a 2D image and an effect mask that can be used to limit the simulated defocused area.

— **Input:** The orange input is used for the primary 2D image for defocusing.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the defocus to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The Defocus node receives a 2D image like the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.

A Defocus node applied to a MediaIn1 node

Inspector
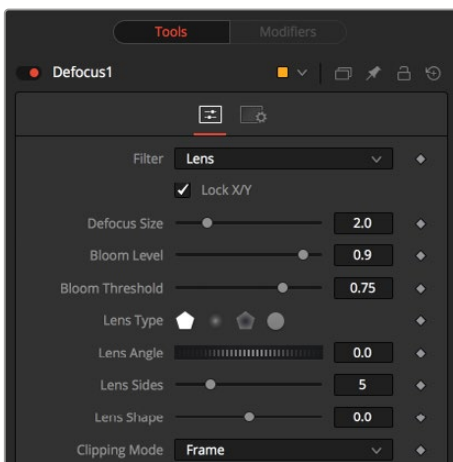
Defocus controls

Controls Tab

The Controls tab contains all the primary controls necessary for customizing the defocus operation.

Filter

Use this menu to select the exact method applied to create the defocus. Gaussian applies a simplistic effect, while Lens mode creates a more realistic defocus. Lens mode takes significantly longer than Gaussian.

Lock X/Y

When Lock X/Y is selected, this performs the same amount of defocusing to both the X- and Y-axis of the image. Deselect to obtain individual control.

Defocus Size

The Defocus Size control sets the size of the defocus effect. Higher values blur the image by greater amounts and produce larger blooms.

Bloom Level

The Bloom Level control determines the intensity and size of the blooming applied to pixels that are above the bloom threshold.

Bloom Threshold

Pixels with values above the set Bloom Threshold are defocused and have a glow applied (blooming). Pixels below that value are only defocused.

The following four lens options are available only when the Filter is set to Lens.

— **Lens Type:** The basic shape used to create the "bad bokeh" effect. This can be refined further with the Angle, Sides, and Shape sliders.

— **Lens Angle:** Defines the rotation of the shape. Best visible with NGon lens types. Because of the round nature of a circle, this slider has no visible effect when the Lens Type is set to Circle.

— **Lens Sides:** Defines how many sides the NGon shapes have. Best visible with NGon lens types. Because of the round nature of a circle, this slider has no visible effect when the Lens Type is set to Circle.

— **Lens Shape:** Defines how pointed the NGons are. Higher values create a more pointed, starry look. Lower values create smoother NGons. Best visible with NGon lens types and Lens Sides between 5 and 10. Because of the round nature of a circle, this slider has no visible effect when the Lens Type is set to Circle.

Clipping Mode

This option determines how edges are handled when performing domain-of-definition rendering. This is profoundly important for nodes like Blur, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that would normally be outside the upstream DoD is treated as black/transparent.
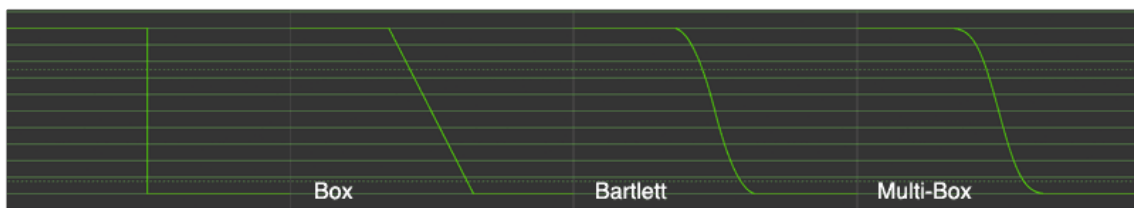
Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Blur nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Directional Blur [DrBl]
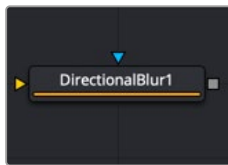
The Directional Blur node

Directional Blur Node Introduction

This node is used to create Directional and Radial blurs. It is useful for creating simulated motion blur and light ray-type effects. Directional Blur affects all channels (RGBA).

Inputs

The two inputs on the Directional Blur node are used to connect a 2D image and an effect mask which can be used to limit the blurred area.

— **Input:** The orange input is used for the primary 2D image that has the directional blur applied.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the directional blur to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The Directional Blur node receives a 2D image like the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.

A Directional Blur node applied to a MediaIn1 node

Inspector
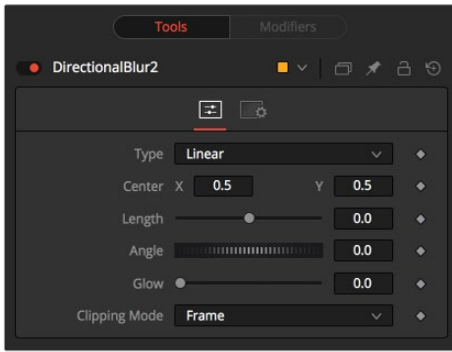
Directional Blur controls

Controls Tab

The Controls tab contains all the primary controls necessary for customizing the directional blur operation.

Type

This menu is used to select the type of directional blur to be applied to the image.

— **Linear:** Linear distorts the image in a straight line, resembling the scenery that appears in the window of a speeding train.
— **Radial:** Radial creates a distortion that originates at some arbitrary center, radiating outward the way that a view would appear if one were at the head of the train looking forward.
— **Centered:** The Centered button produces a similar result to linear, but the blur effect is equally distributed on both sides of the original.
— **Zoom:** Zoom creates a distortion in the scale of the image smear to simulate the zoom streaking of a camera filming with a slow shutter speed.

Center X and Y

This coordinate control and related viewer crosshair affects the Radial and Zoom Motion Blur types only. It is used to position where the blurring effect starts.

Length

Length adjusts the strength and heading of the effect. Values lower than zero cause blurs to head opposite the angle control. Values greater than the slider maximum may be typed into the slider's edit box.

Angle

In both Linear and Center modes, this control modifies the direction of the directional blur. In the Radial and Zoom modes, the effect is similar to the camera spinning while looking at the same spot. If the setting of the length slider is other than zero, the effect creates a whirlpool effect.

Glow

This adds a Glow to the directional blur, which can be used to duplicate the effect of increased camera exposure to light caused by longer shutter speeds.

Clipping Mode

This option determines how edges are handled when performing domain-of-definition rendering. This is profoundly important for nodes like Blur, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that would normally be outside the upstream DoD is treated as black/transparent.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Blur nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Glow [Glo]

The Glow nozzde

Glow Node Introduction

A Glow is created by blurring an image, and then brightening the blurred result and mixing it back with the original. The Glow node provides a variety of variations on this theme. For example, a Bartlett glow is a high-quality glow with a smoother drop-off; however, it is more processor-intensive at larger sizes.

Inputs

The Glow node has three inputs: an orange one for the primary 2D image input, a blue one for an effect mask, and a third white input for a Glow mask.

— **Input:** The orange input is used for the primary 2D image that has the glow applied.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input restricts the source of the glow to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

— **Glow Mask:** The Glow node supports pre-masking using the white glow mask input. A Glow pre-mask filters the image before applying the glow. The glow is then merged back over the original image. This is different from a regular effect mask that clips the rendered result.

The Glow mask allows the glow to extend beyond the borders of the mask, while restricting the source of the glow to only those pixels within the mask.

Glow masks are identical to Effect masks in every other respect.

Basic Node Setup

The Glow node receives a 2D image like the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.
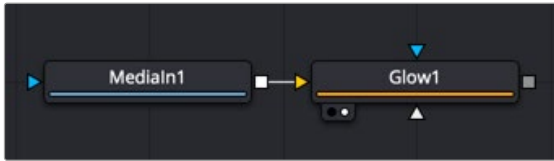
A Glow node applied to a MediaIn1 node in DaVinci Resolve
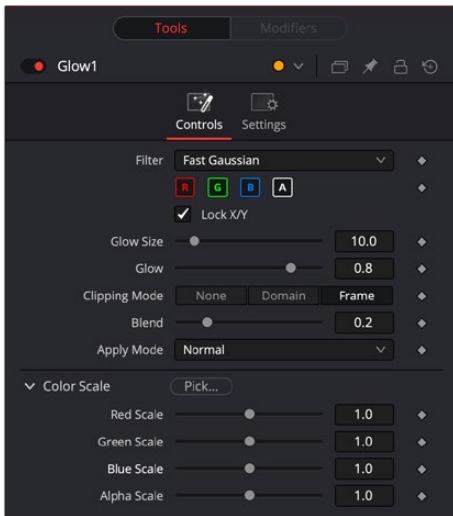
Inspector

Glow controls

Controls Tab

The Controls tab contains all the primary controls necessary for customizing the glow operation. A Color Scale section at the bottom of the Inspector can be used for tinting the glow.

Filter

Use this menu to select the method of Blur used in the filter. The selections are described below.

— **Box:** A simple but very fast Box filter.
— **Bartlett:** Bartlett adds a softer, subtler glow with a smoother drop-off but may take longer to render than Box.

- **Multi-box:** Multi-box uses a Box filter layered in multiple passes to approximate a Gaussian shape. With a moderate number of passes (e.g., four), a high-quality blur can be obtained, often faster than the Gaussian filter, and without any ringing.
- **Gaussian:** Gaussian adds a soft glow, blurred by the Gaussian algorithm.
- **Fast Gaussian:** Fast Gaussian adds a soft glow, blurred by the Gaussian algorithm. This is the default method.
- **Blend:** Blend adds a nonlinear glow that is evenly visible in the whites and blacks.
- **Hilight:** Hilight adds a glow without creating a halo in the surrounding pixels.
- **Solarize:** Solarize adds a glow and solarizes the image.

Color Channels (RGBA)

This filter defaults to operating on R, G, B, and A channels. Selective channel filtering is possible by clicking each channel to make them active or inactive.

> **NOTE:** This is not the same as the RGBA checkboxes found under the common controls. The node takes these selections into account before it processes the image, so deselecting a channel causes the node to skip that channel when processing, speeding up the rendering of the effect. In contrast, the channel controls under the Common Controls tab are applied after the node has processed.

Lock X/Y

When Lock X/Y is checked, both the horizontal and vertical glow amounts are locked. Otherwise, separate amounts of glow may be applied to each axis.

Glow Size

Glow Size determines the size of the glow effect. Larger values expand the size of the glowing highlights of the image.

Num Passes

Only available in Multi-box mode. Larger values lead to a smoother distribution of the effect, but also increase render times. It's good to find the line between desired quality and acceptable render times.

Glow

The Glow slider determines the intensity of the glow effect. Larger values tend to completely blow the image out to white.

Clipping Mode

This option determines how edges are handled when performing domain-of-definition rendering. This is profoundly important for nodes like Blur, which may require samples from portions of the image outside the current domain.
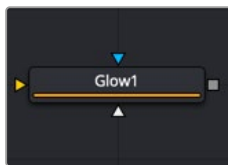
- **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that would normally be outside the upstream DoD is treated as black/transparent.

Blend

The Blend slider determines the percentage of the affected image that is mixed with original image. It blends in more of the original image as the value gets closer to 0.

This control is a cloned instance of the Blend slider in the Common Controls tab. Changes made to this control are simultaneously made to the one in the common controls.

Apply Mode

Three Apply Modes are available when it comes to applying the glow to the image.

— **Normal:** Default. This mode simply adds the glow directly over top of the original image.
— **Merge Under:** Merge Under places the glow beneath the image, based on the Alpha channel. Threshold mode permits clipping of the threshold values.
— **Threshold:** This control clips the effect of the glow. A new range slider appears. Pixels in the glowed areas with values below the low value are pushed to black. Pixels with values greater than high are pushed to white.
— **High-Low Range Control:** Available only in Threshold mode. Pixels in the glowed areas with values below the low value are pushed to black. Pixels with values greater than high are pushed to white.

Color Scale (RGBA)

These Scale sliders can be used to adjust the amount of glow applied to each color channel individually, by tinting the glow.

By click and holding on the Pick button, then dragging the pointer over the viewer, you can select a specific color from the image.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Blur nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Sharpen [Shrp]
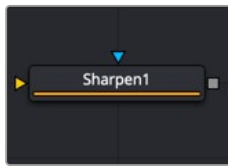
The Sharpen node

Sharpen Node Introduction

The Sharpen node uses a convolution filter to enhance detail in an image overall or to an individual channel.

Inputs

The two inputs on the Sharpen node are used to connect a 2D image and an effect mask that can limit the area affected by the sharpen.

— **Input:** The orange input is used for the primary 2D image for sharpening.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the sharpen to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The Sharpen node receives a 2D image like the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.

A Sharpen node applied to a MediaIn1
node in DaVinci Resolve

Inspector
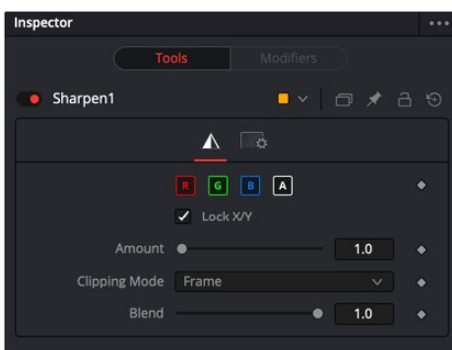
Sharpen controls

Controls Tab

The Controls tab contains all the primary controls necessary for customizing the sharpen operation.

Color Channels (RGBA)

This filter defaults to operating on R, G, B, and A channels. Selective channel filtering is possible by clicking the channel buttons to make them active or inactive.

> **NOTE:** This is not the same as the RGBA checkboxes found under the common controls. The node takes these selections into account before it processes the image, so deselecting a channel causes the node to skip that channel when processing, speeding up the rendering of the effect. In contrast, the channel controls under the Common Controls tab are applied after the node has processed.

Lock X/Y

This locks the X and Y Sharpen sliders together for symmetrical sharpening. This is checked by default.

Amount

This slider sets the amount of sharpening applied to the image. When the Lock X/Y control is deselected, independent control over each axis is provided.

Clipping Mode

This option determines how edges are handled when performing domain-of-definition rendering. This is profoundly important for nodes like Blur, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that would normally be outside the upstream DoD is treated as black/transparent.

Blend

The Blend slider determines the percentage of the affected image that is mixed with original image. It blends in more of the original image as the value gets closer to 0.

This control is a cloned instance of the Blend slider in the Common Controls tab. Changes made to this control are simultaneously made to the one in the common controls.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Blur nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Soft Glow [SGlo]

The Soft Glow node

Soft Glow Node Introduction

The Soft Glow node is similar to the Glow node but performs additional processing of the image to create a much softer, more natural glow.

This node is perfect for atmospheric haze around planets, skin tones, and simulating dream like environments.

Inputs

Like the Glow node, Soft Glow also has three inputs: an orange one for the primary image input, a blue one for an effect mask, and a third white input for a Glow mask.

— **Input:** The orange input is used for the primary 2D image for the soft glow.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the soft glow to only those pixels within the mask. An effect mask is applied to the tool after it is processed.
— **Glow Mask:** The Soft Glow node supports pre-masking using the white glow mask input. A Glow pre-mask filters the image before applying the soft glow. The soft glow is then merged back over the original image. This is different from a regular effect mask that clips the rendered result.

The Glow mask allows the soft glow to extend beyond the borders of the mask, while restricting the source of the soft glow to only those pixels within the mask.

Glow masks are identical to effect masks in every other respect.

Basic Node Setup

The Soft Glow node receives a 2D image like the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.

A Soft Glow node applied to a MediaIn1 node
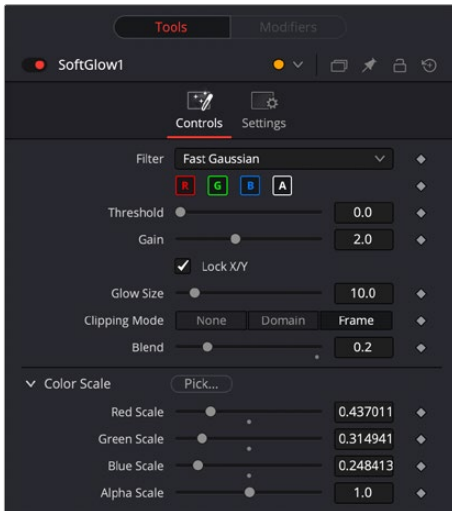
Inspector

Soft Glow controls

Controls Tab

The Controls tab contains all the primary controls necessary for customizing the soft glow operation. A color scale section at the bottom of the Inspector can be used for tinting the soft glow.

Filter

Use this menu to select the method of Blur used in the filter. The selections are described below.

— **Box:** A simple but very fast Box filter.

— **Bartlett:** Bartlett adds a softer, subtler glow with a smoother drop-off but may take longer to render than Box.

— **Multi-box:** Multi-box uses a Box filter layered in multiple passes to approximate a Gaussian shape. With a moderate number of passes (e.g., four), a high-quality blur can be obtained, often faster than the Gaussian filter and without any ringing.

— **Gaussian:** Gaussian adds a soft glow, blurred by the Gaussian algorithm. This is the default method.

Color Channels (RGBA)

The filter defaults to operating on R, G, B, and A channels. Selective channel filtering is possible by clicking the channel buttons to make them active or inactive.

> **NOTE:** This is not the same as the RGBA checkboxes found under the common controls. The node takes these selections into account before it processes the image, so deselecting a channel causes the node to skip that channel when processing, speeding up the rendering of the effect. In contrast, the channel controls under the Common Controls tab are applied after the node has processed.

Threshold

This control is used to limit the effect of the soft glow. The higher the threshold, the brighter the pixel must be before it is affected by the glow.

Gain

The Gain control defines the brightness of the glow.

Lock X/Y

When Lock X/Y is checked, both the horizontal and vertical glow amounts are locked. Otherwise, separate amounts of glow may be applied to each axis of the image.

Glow Size

This amount determines the size of the glow effect. Larger values expand the size of the glowing highlights of the image.

Num Passes

Available only in Multi-box mode. Larger values lead to a smoother distribution of the effect, but also increase render times. It's good to find the line between desired quality and acceptable render times.

Clipping Mode

This option determines how edges are handled when performing domain-of-definition rendering. This is profoundly important for nodes like Blur, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that would normally be outside the upstream DoD is treated as black/transparent.

Blend

The Blend slider determines the percentage of the affected image that is mixed with original image. It blends in more of the original image as the value gets closer to 0.

This control is a cloned instance of the Blend slider in the Common Controls tab. Changes made to this control are simultaneously made to the one in the common controls.

Color Scale (RGBA)

These Scale sliders are used to adjust the amount of glow applied to each color channel individually, by tinting the glow.

By click and holding on the Pick button, then dragging the pointer over the viewer, you can select a specific color from the image.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Blur nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Unsharp Mask [UsM]

The Unsharp Mask node

Unsharp Mask Introduction

Unsharp masking is a technique used to sharpen only the edges within an image. This node is most often used to correct for blurring and loss of detail in low-contrast images; for example, to extract useful detail from long exposure shots of faraway galaxies.

This filter extracts a range of frequencies from the image and blurs them to reduce detail. The blurred result is then compared to the original images. Pixels with a significant difference between the original and the blurred image are likely to be an edge detail. The pixel is then brightened to enhance it.

Inputs

The two inputs on the Unsharp Mask node are used to connect a 2D image and an effect mask for limiting the effect.

— **Input:** The orange input is used for the primary 2D image for the Unsharp Mask.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Unsharp Mask to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The Unsharp Mask node receives a 2D image like the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.

An Unsharp mask node applied to a MediaIn1 node

Inspector
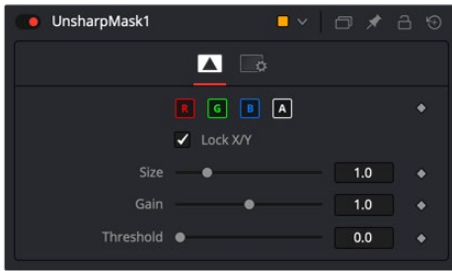
Unsharp Mask controls

Controls Tab

The Controls tab contains all the primary controls necessary for customizing the Unsharp Mask operation.

Color Channels (RGBA)

The filter defaults to operating on R, G, B, and A channels. Selective channel filtering is possible by clicking the channel buttons to make them active or inactive.

> **NOTE:** This is not the same as the RGBA checkboxes found under the common controls. The node takes these selections into account before it processes the image, so deselecting a channel causes the node to skip that channel when processing, speeding up the rendering of the effect. In contrast, the channel controls under the Common Controls tab are applied after the node has processed.

Lock X/Y

When Lock X/Y is checked, both the horizontal and vertical sharpen amounts are locked. Otherwise, separate amounts of glow may be applied to each axis of the image.

Size

This control adjusts the size of blur filter applied to the extracted image. The higher this value, the more likely it is that pixels are identified as detail.

Gain

The Gain control adjusts how much gain is applied to pixels identified as detail by the mask. Higher values create a sharper image.

Threshold

This control determines the frequencies from the source image to be extracted. Raising the value eliminates lower-contrast areas from having the effect applied.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Blur nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Vari Blur [VBl]

The Vari Blur node

Vari Blur Node Introduction

The Vari Blur node gives a true per-pixel variable blur, using a second image to control the amount of blur for each pixel. It is somewhat similar in effect to the Depth Blur node but uses a different approach for frequently cleaner results.

Inputs

There are two inputs on the Vari Blur node for the primary image: the blur map image, and an effect mask.

— **Input:** The gold image input is a required connection for the primary image you wish to blur.
— **Blur Image:** The green input is also required, but it can accept a spline shape, text object, still image, or movie file as the blur map image. Once connected, you can choose red, green, blue, Alpha, or luminance channel to create the shape of the blur.
— **Effect Mask:** The optional blue effect mask input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Vari Blur to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The Vari Blur node receives a 2D image like the MediaIn1 shown below. A gradient Background tool connects to the Blur image input to control the areas affected by the blur. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.
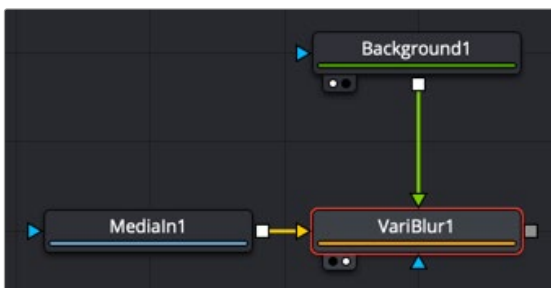
A Vari Blur node applied to a MediaIn1 node and a gradient background directing the blurred regions

Inspector
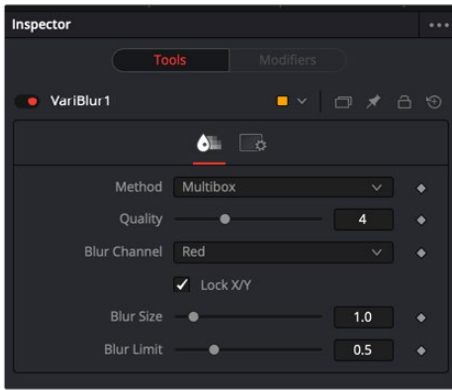
Vari Blur controls

Controls Tab

The Controls tab contains all the primary controls necessary for customizing the Vari Blur operation.

Method

Use this menu to select the method of Blur used in the filter. The selections are described below.

— **Soften:** This method varies from a simple Box shape to a Bartlett triangle to a decent-looking Smooth blur as Quality is increased. It is a little better at preserving detail in less-blurred areas than Multi-box.

— **Multi-box:** Similar to Soften, this gives a better Gaussian approximation at higher Quality settings.

— **Defocus:** Produces a flat, circular shape to blurred pixels that can approximate the look of a defocus.

Quality

Increasing Quality gives smoother blurs, at the expense of speed. Quality set to 1 uses a very fast but simple Box blur for all Method settings. A Quality of 2 is usually sufficient for low Blur Size values. A Quality of 4 is generally good enough for most jobs unless Blur Size is particularly high.

Blur Channel

This selects which channel of the Blur Image controls the amount of blurring applied to each pixel.

Lock X/Y

When selected, only a Blur Size control is shown, and changes to the amount of blur are applied to both axes equally. If the checkbox is cleared, individual controls appear for both X and Y Blur Size.

Blur Size

Increasing this control increases the overall amount of blur applied to each pixel. Those pixels where the Blur image is black or nonexistent are blurred, despite the Blur Size.

Blur Limit

This slider limits the useable range from the Blur image. Some Z-depth images can have values that go to infinity, which skew blur size. The Blur Limit is a way to keep values within range.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Blur nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Vector Motion Blur [VMB]
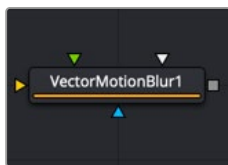
The Vector Motion Blur node

Vector Motion Blur Introduction

This node is used to create directional blurs based on a Motion Vector map or AOV (Arbitrary Output Variable) channels exported from 3D-rendering software like Arnold, Renderman, or VRay. You can also generate motion vectors using the Optical Flow node in Fusion.

The vector map is typically two floating-point images: one channel specifies how far the pixel is moving in X, and the other specifies how far the pixel is moving in Y. These channels may be embedded in OpenEXR or RLA/RPF images, or may be provided as separate images using the node's Vectors input.

The vector channels should use a float16 or float32 color depth, to provide + and – values.

A value of 1 in the X channel would indicate that pixel has moved one pixel to the right, while a value of –10 indicates ten pixels of movement to the left.

Inputs

The Vector Motion Blur node has three inputs for a 2D image, a motion vector pass, and an effect mask.

— **Input:** The required orange input is for a 2D image that receives the motion blur.
— **Vectors:** The green input is also required. This is where you connect a motion vector AOV rendered from a 3D application or an EXR file generated from the Optical Flow node in Fusion.
— **Vector Mask:** The white Vector Mask input is an optional input that masks the image before processing.
— **Effect Mask:** The common blue input is used for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input restricts the source of the motion blur to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The Vector Motion Blur node receives a 2D image like the IMAGE shown below. A MediaIn or Loader node containing motion vectors is connected to the Vector's input. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.

A Vector Motion Blur node applied to a MediaIn or Loader node with motion vectors connected to the Vectors input

Inspector
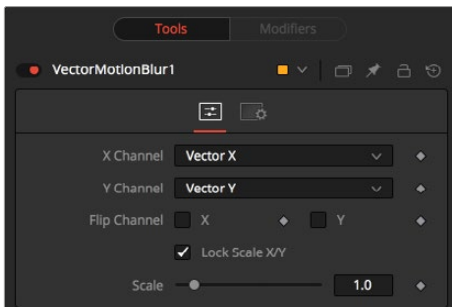
Vector Motion Blur node

Controls Tab

The Controls tab contains all the primary controls necessary for customizing the Vector Motion Blur operation.

X Channel

Use this menu to select which channel of the image provides the vectors for the movement of the pixels along the X-axis.

Y Channel

Use this menu to select which channel of the image provides the vectors for the movement of the pixels along the Y-axis.

Flip Channel

These checkboxes can be used to flip, or invert, the X and Y vectors. For instance, a value of 5 for a pixel in the X-vector channel would become –5 when the X checkbox is enabled.

Lock Scale X/Y

Selecting this checkbox provides access to separate sliders for X and Y Scale. By default, only a single Scale slider is provided.

Scale

The X and Y vector channel values for a pixel are multiplied by the value of this slider. For example, given a scale of 2 and a vector value of 10, the result would be 20. This slider splits to show Scale X and Scale Y if the Lock Scale X/Y checkbox is not enabled.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Blur nodes. These common controls are described in the following "The Common Controls" section.

The Common Controls

Nodes that handle blur operations share several identical controls in the Inspector. This section describes controls that are common among Blur nodes.

Inspector
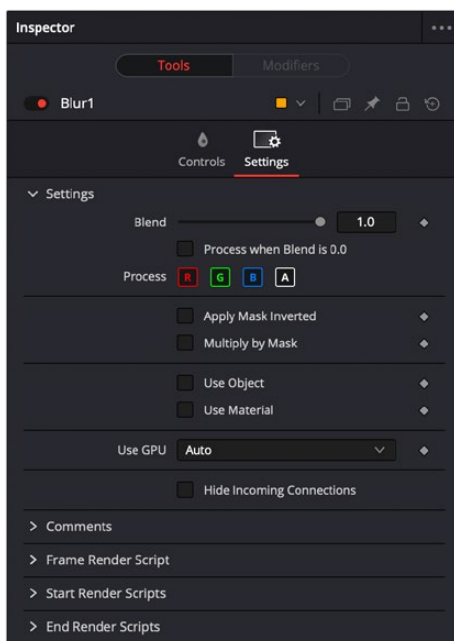
Common Blur settings

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Blur category. The Settings controls are even found on third-party Blur-type plugin tools. The controls are consistent and work the same way for each tool.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this causes the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if processing of this node is scripted to trigger another task, but the value of the node is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the Red button on a Blur tool is deselected, the blur is first applied to the image, and then the red channel from the original input is copied back over the red channel of the result.

There are some exceptions, such as tools where deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option causes the RGB values of the masked image to be multiplied by the mask channel's values. This causes all pixels of the image not in the mask (i.e. set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports object and material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels are used, if present. The specific material ID or object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID is used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker to grab IDs from the image displayed in the viewer. The image or sequence must have been rendered from a 3D software package with those channels included.

Use GPU

The GPU menu has three settings. Disable turns off GPU hardware accelerated rendering. Enabled uses the GPU hardware for rendering the node. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available.

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A Quality setting of 2 causes Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one whole frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Comments

The Comments field is used to add notes to a tool. Click in the field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Color Nodes

This chapter details the Color nodes available in Fusion. The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Auto Gain [AG]

The Auto Gain node

Auto Gain Node Introduction

The Auto Gain node automatically adjusts the tonal range of an image, setting the darkest and brightest pixels to user-selected values. By default, the darkest pixels get pushed to black, the brightest pixels get pushed to white, and pixels in between get stretched to cover the tonal range evenly.

This can be useful when compensating for variations in lighting, dealing with low-contrast images, or visualizing the full color range of float images (although the viewer's View Normalized Image option is generally more suitable for this).

Inputs

The two inputs on the Auto Gain node are the input and effect mask.

— **Input:** The orange input connects the primary 2D image for the auto gain.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the auto gain adjustment to only those pixels within the mask. An effect mask is applied to the tool after the tool is processed.

Basic Node Setup

The Auto Gain node, like many 2D image-processing nodes, receives a 2D image like a Loader node or the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.

An Auto Gain node applied to a MediaIn1 node

Inspector
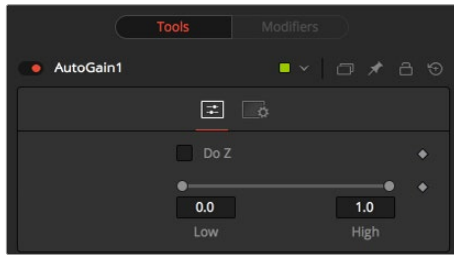
Auto Gain controls

Controls Tab

The Controls tab contains the few primary controls necessary for customizing the AutoGain operation.

> **NOTE:** Variations over time in the input image can cause corresponding variations in the levels of the result. For example, if a bright object moves out of an otherwise dark shot, the remaining scene gets suddenly brighter, as the remaining darker values get stretched to white. This also applies to sudden depth changes when Do Z is applied; existing objects may be pushed forward or backward when a near or far object enters or leaves the scene.

Do Z

Select the Do Z checkbox to apply the Auto Gain effect to the Z or Depth channels. This can be useful for matching the ranges of one Z-channel to another, or to view a float Z-channel in the RGB values.

Range

This Range control sets the black point and white point in the image. All tonal values in the image rescale to fit within this range.

> **EXAMPLE** Create a horizontal gradient with the Background node. Set one color to dark gray (RGB Values 0.2). Set the other color to light gray (RGB Values 0.8).
>
> Add an Auto Gain node and set the Low value to 0.0 and the High value to 0.5. This causes the brightest pixels to be pushed down to 0.5, and the darkest pixels get pushed to black. The remainder of the pixel values scale between those limits.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Brightness Contrast [BC]

The Brightness Contrast node

Brightness Contrast Node Introduction

The Brightness Contrast node adjusts the gain, brightness, contrast, gamma, and saturation of an image. The order of the controls represents the order in which the operations are applied. For example, gamma gets applied before contrast but after gain. The Brightness Contrast is also reversible using the Forward and Reverse buttons. So color corrections, once applied, can be reversed further downstream.

For this to work best, image processing should operate in 32-bit floating point.

Inputs

The two inputs on the Brightness Contrast node are the input and effect mask.

— **Input:** The orange input connects the primary 2D image for the brightness contrast.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Brightness Contrast adjustment to only those pixels within the mask. An effect mask is applied to the tool after the tool is processed.

Basic Node Setup

The Brightness Contrast node, like many 2D image-processing nodes, receives a 2D image like a Loader node or the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.

A Brightness Contrast node applied to a MediaIn1 node

Inspector
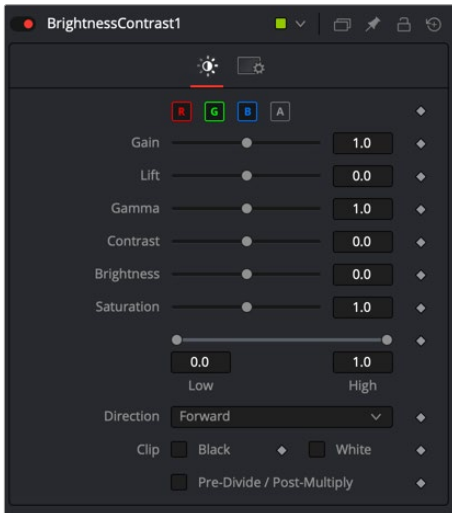
Brightness Control controls

Controls Tab

The Controls tab contains all the primary controls necessary for customizing the brightness, contrast operations.

Color Channels (RGBA)

The filter defaults to operating on R, G, B, and A channels. Selective channel filtering is possible by clicking each channel button to make them active or inactive.

> **NOTE:** This is not the same as the RGBA checkboxes found under the common controls. The node takes these selections into account before it processes the image, so deselecting a channel causes the node to skip that channel when processing, speeding up the rendering of the effect. In contrast, the channel controls under the Common Controls tab get applied after the node has processed.

Gain

The gain slider is a multiplier of the pixel value. A Gain of 1.2 makes a pixel that is R0.5 G0.5 B0.4 into R0.6 G0.6, B0.48 (i.e., 0.4 * 1.2 = 0.48) while leaving black pixels unaffected. Gain affects higher values more than it affects lower values, so the effect is most influential in the midrange and top range of the image.

Lift

While Gain scales the color values around black, Lift scales the color values around white. The pixel values get multiplied by the value of this control. A Lift of 0.5 makes a pixel that is R0.0 G0.0 B0.0 into R0.5 G0.5, B0.5 while leaving white pixels unaffected. Lift affects lower values more than it affects higher values, so the effect is most influential in the midrange and low range of the image.

Gamma

Values higher than 1.0 raise the Gamma (mid-gray), whereas lower values decrease it. The effect of this node is not linear, and existing black or white points are not affected at all. Pure gray colors are affected the most.

Contrast

Contrast is the range of difference between the light to dark areas. Increasing the value of this slider increases the contrast, pushing color from the midrange toward black and white. Reducing the contrast causes the colors in the image to move toward midrange, reducing the difference between the darkest and brightest pixels in the image.

Brightness

The value of the Brightness slider gets added to the value of each pixel in the image. This control's effect on an image is linear, so the effect is applied identically to all pixels regardless of value.

Saturation

Use this control to increase or decrease the amount of Saturation in the image. A saturation of 0 has no color, reducing the image to grayscale.

Low and High

This range control is similar to the Gain control in some respects. If Low gets anchored at 0.0 and the High value gets reduced from 1.0, the effect is identical to increasing the gain. High values get multiplied by the inverse of the high value. (e.g., if high is 0.75, each pixel is multiplied by 1/0.75, or 1.3333).

Leaving the high anchored at 1.0 and increasing the low is the same as inverting the image colors and increasing the gain and inverting it back again. This pushes more of the image toward black without affecting the whites at all.

Direction

Forward applies all values normally. Reverse effectively inverts all values.

Clip Black/White

The Clip Black and Clip White checkboxes clip out-of-range color values that can appear in an image when processing in floating-point color depth. Out-of-range colors are below black (0.0) or above white (1.0). These checkboxes have no effect on images processed at 8-bit or 16-bit per channel, as such images cannot have out-of-range values.

Pre-Divide/Post-Multiply

Selecting the Pre-Divide/Post-Multiply checkbox causes the image pixel values to be divided by the Alpha values before the color correction, and then re-multiplied by the Alpha value after the correction.

This helps to prevent the creation of illegally additive images when color correcting images with premultiplied Alpha channels.

Common Controls

Settings Tab

The Settings tab in the Inspector appears in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Channel Booleans [Bol]

The Channel Booleans node

Channel Booleans Node Introduction

The Channel Booleans node applies a variety of mathematical and logical operations on the channels in an image. This node works by using one image's channels to modify another image's channels. If a foreground input is not available, selecting options that use color channels from the foreground ends up using the background input's color channels instead.

> **NOTE:** Be aware of another similarly named Channel Boolean (3Bol), which is a 3D node used to remap and modify channels of 3D materials. When modifying 2D channels, use the Channel Booleans (with an "s") node (Bol).

Inputs

There are four inputs on the Channel Booleans node in the Node Editor, but only the orange Background input is required.

— **Background:** This orange input connects a 2D image that gets adjusted by the foreground input image.
— **Effect Mask:** The blue effect mask input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the channel booleans adjustment to only those pixels within the mask.
— **Foreground:** The green foreground input connects a 2D image that is used to adjust the background input image.
— **Matte:** The white matte input can be used to combine external mattes with the foreground and background operations.

Basic Node Setup

The Channel Booleans node is an extremely flexible tool used in many different ways. The example below copies the z-depth channel from the foreground input (green) into the background image (orange).
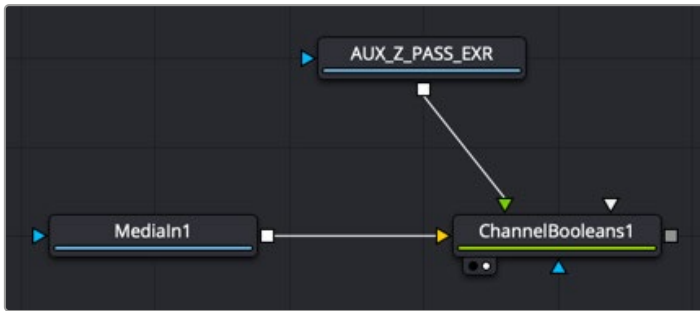
A Channel Booleans set to copy from foreground to background

Inspector
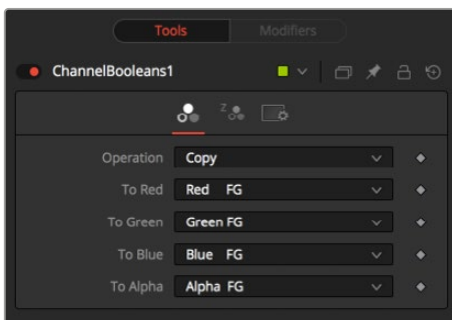
Channel Booleans controls

Color Channel Tab

On the Color Channels tab, the controls are divided into two columns.

On the left side are target channels for the image connected into the orange background input. The drop-down menu to the right lets you choose whether you want to modify the BG image with its channels (suffix BG after list name) or with the channels from an image connected into the green foreground input on the node (suffix FG in the drop-down list).

Operation

This menu selects the mathematical operation applied to the selected channels. The options are as follows:

Copy

Copy the value from one color channel to another. For example, copy the foreground red channel into the background's Alpha channel to create a matte.

— **Add:** Add the color values from one color channel to another channel.

— **Subtract:** Subtract the color values of one color channel from another color channel.

— **And:** Perform a logical AND on the color values from color channel to color channel. The foreground image generally removes bits from the color channel of the background image.

— **Or:** Perform a logical OR on the color values from color channel to color channel. The foreground image generally adds bits from the color channel of the background image.

— **Exclusive Or:** Perform a logical XOR on the color values from color channel to color channel. The foreground image generally flips bits from the color channel of the background image.

— **Multiply:** Multiply the values of a color channel. This gives the appearance of darkening the image as the values scale from 0 to 1. White has a value of 1, so the result would be the same. Gray has a value of 0.5, so the result would be a darker image or, in other words, an image half as bright.

— **Divide:** Divide the values of a color channel. This gives the appearance of lightening the image as the values scale from 0 to 1.

— **Maximum:** Compare the two images and take the maximum, or brightest, values from each image.

— **Minimum:** Compare the two images and take the minimum, or darkest, values from each image.

— **Negative:** Invert the FG input to make a negative version of the image.

— **Solid:** Solid sets a channel to a full value of 255. This is useful for setting the Alpha to full value.

— **Clear:** Clear sets a channel to a value of zero. This is useful for clearing the Alpha.

— **Difference:** Difference subtracts the greater color values of one color channel from the lesser values of another color channel.

— **Signed Add:** Signed Add subtracts areas that are lower than mid-gray and adds areas that are higher than mid-gray, which is useful for creating effects with embossed gray images.

To Red, To Green, To Blue, To Alpha

These menus represent the four color channels of the output image. Use the drop-down menu to select which channel from the source images produces the output channel.

The default setting copies the channels from the foreground channel. Select any one of the four color channels, as well as several auxiliary channels like Z-buffer, saturation, luminance, and hue.

Inspector
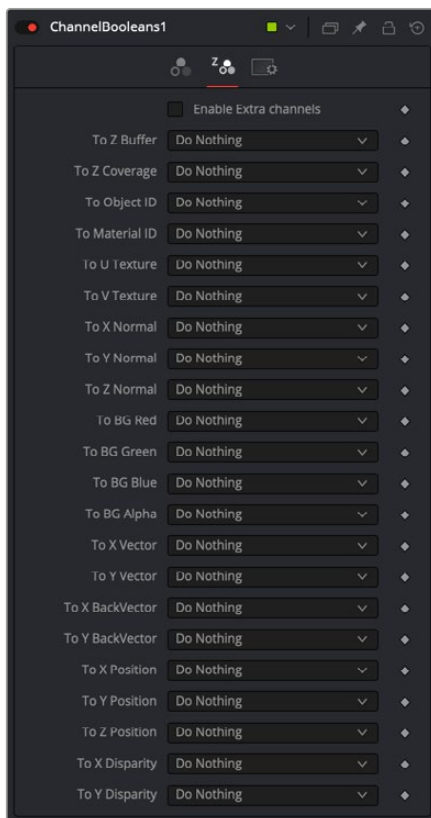
Aux Channel Inspector

Aux Channel Tab

This tab includes a series of menus where you select a source for the auxiliary channels of the output image.

Enable Extra Channels

When the Enable Extra Channels checkbox is selected, the Channel Booleans node can output images with channels beyond the usual RGBA. Once enabled, the remaining controls in the Aux Channels tab can copy data into the auxiliary channels.

> **EXAMPLES** To copy the Alpha channel of one image to its color channels, set the red, green, and blue channels to Alpha BG. Set the Operation to Copy.
>
> To copy the Alpha channel from another image, set operation type to Alpha FG.
>
> To replace the existing Alpha channel of an image with the Alpha of another image, choose "Do Nothing" for To Red, To Green, and To Blue and "Alpha FG" for To Alpha. Pipe the image containing the Alpha into the foreground input on the Channel Booleans node. Set Operation: "Copy." The same operation is available in the Matte Control node.
>
> To combine any mask into an Alpha channel of an image, choose "Do Nothing" for To Red, To Green, and To Blue and "Matte" for To Alpha. Pipe the mask into the foreground input on the Channel Booleans node. Set Operation: "Copy."
>
> To subtract the red channel's pixels of another image from the blue channel, choose "Do Nothing" for To Red and To Green and "Red FG" for To Blue. Pipe the image containing the red channel to subtract into the foreground input on the Channel Booleans node. Set Operation: "Subtract."

Common Controls

Settings Tab

The Settings tab in the Inspector appears in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Color Corrector [CC]

The Color Corrector node

Color Corrector Node Introduction

The Color Corrector node is a comprehensive color node with histogram matching, and equalization, hue shifting, tinting, and color suppression.

Controls in the Color Corrector node are separated into four tabs: Correction, Ranges, Options, and Settings.

Inputs

The Color Corrector node includes four inputs in the Node Editor.

— **Input:** This orange input is the only required connection. It connects a 2D image for color correction.
— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the color corrector adjustment to only those pixels within the mask. An effect mask is applied to the tool after the tool is processed.
— **Match Reference:** The green input is used to connect an image that can be a reference for histogram matching.
— **Match Mask:** This optional white input accepts any mask much like an effect mask. However, this mask defines of the area to match during a Histogram Match. It offers more flexibility in terms of shape than the built-in Match rectangle in the Inspector.

Basic Node Setup

The Color Corrector node, like many 2D image-processing nodes, receives a 2D image like a Loader node or the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.

A Color Corrector node applied to a MediaIn1 node

Inspector

Color Corrector controls

Correction Tab Colors Menu

The main Correction tab is further separated into four types of correction methods: colors, levels, histogram, and suppress. Selecting one from the menu at the top of the Correction tab causes that method's controls to appear. The Color method is described in detail below.

Range

This menu determines the tonal range affected by the color correction controls in this tab. The menu can be set to Shadows, Midtones, Highlights, and Master, where Master is the default affecting the entire image.

The selected range is maintained throughout the Colors, Levels, and Suppress sections of the Color Corrector node.

Adjustments made to the image in the Master channel are applied to the image after any changes made to the Highlight, Midtone, and Shadow ranges.

> **NOTE:** The controls are independent for each color range. For example, adjusting the Gamma control while in Shadows mode does not change or affect the value of the Gamma control for the Highlights mode. Each control is independent and applied separately.

Color Wheel

The color wheel provides a visual representation of adjustments made to Hue and Saturation, as well as any tinting applied to the image. Adjustments can be made directly by dragging the color indicator, or by entering values in the numeric boxes under the color wheel.

The tinting is represented in the color wheel color indicator that shows the color and strength of the tint. The Highlight setting uses a black outline for the color indicator. The Midtones and Shadows use gray color indicators. The Master color indicator is also black, but it has a white M in the center to distinguish it from the others.

The mouse can position the color indicator for each range only when the applicable range is selected. For example, the Highlight color indicator cannot be moved when the Master range is selected.

Holding down the Command or Ctrl key while dragging this indicator allows you to make finer adjustments by reducing the control's sensitivity to mouse movements. Holding down the Shift key limits the movement of the color indicator to a single axis, allowing you to restrict the effect to either tint or strength.

Tint Mode

This menu is used to select the speed and quality of the algorithm used to apply the hue and saturation adjustments. The default is Better, but for working with larger images, it may be desirable to use a faster method.

Hue

This slider is a clone of the Hue control located under the color wheel. The slider makes it easier to make small adjustments to the value with the mouse. The Hue control provides a method of shifting the hue of the image (or selected color range) through the color spectrum. The control value has an effective range between -0.1 and 1.0, which represents the angle of rotation in a clockwise direction. A value of 0.25 would be 90 degrees (90/360) and would have the effect of shifting red toward blue, green to red, and so on.

Hue shifting can be done by dragging the slider, entering a value directly into the text control, or by placing the mouse above the outer ring of the color wheel and dragging the mouse up or down. The outer ring always shows the shifted colors compared to the original colors shown in the center of the wheel.

Saturation

This slider is a clone of the Saturation control located under the color wheel. The slider makes it easier to make small adjustments to the value with the mouse. The Saturation control is used to adjust the intensity of the color values. A saturation of 0 produces gray pixels without any color component, whereas a value of 1.0 produces no change in the chroma component of the input image. Higher values generate oversaturated values with a high color component.

Saturation values can be set by dragging the slider, entering a value directly into the text control, or by dragging the mouse to the left and right on the outer ring of the color wheel control.

Channel

This menu is set for the Histogram, Color, and Levels sections of the Color Corrector node. When the red channel is selected, the controls in each mode affect the red channel only, and so on.

The controls are independent, so switching to blue does not remove or eliminate any changes made to red, green, or Master. The animation and adjustments made to each channel are separate. This menu simply determines what controls to display.

Contrast

Contrast is the range of difference between the light to dark areas. Increasing the value of this slider increases the contrast, pushing color from the midrange toward black and white. Reducing the contrast causes the colors in the image to move toward midrange, reducing the difference between the darkest and brightest pixels in the image.

Gain

The Gain slider is a multiplier of the pixel value. A gain of 1.2 makes a pixel that is R0.5 G0.5 B0.4 into R0.6 G0.6, B0.48 (i.e., 0.4 * 1.2 = 0.48), while leaving black pixels totally unaffected. Gain affects higher values more than it affects lower values, so the effect is strongest in the midrange and top range of the image.

Lift

While Gain scales the color values around black, Lift scales the color values around white. The pixel values are multiplied by the value of this control. A Lift of 0.5 makes a pixel that is R0.0 G0.0 B0.0 into R0.5 G0.5, B0.5, while leaving white pixels totally unaffected. Lift affects lower values more than it affects higher values, so the effect is strongest in the midrange and low range of the image.

Gamma

Values higher than 1.0 raise the Gamma (mid gray), whereas lower values decrease it. The effect of this node is not linear, and existing black or white points are not affected at all. Pure gray colors are affected the most.

Brightness

The value of the Brightness slider is added to the value of each pixel in your image. This control's effect on an image is linear, so the effect is applied identically to all pixels despite value.

Reset All Color Changes

Selecting this button returns all color controls in this section to their default values.
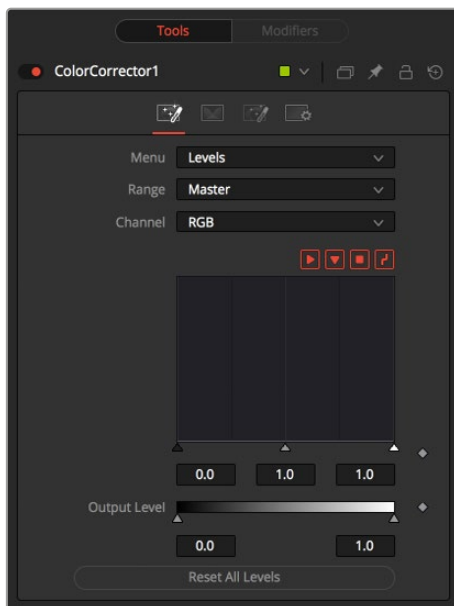
Color Corrector Levels controls

Correction Tab Levels Menu

The main Correction tab is further separated into four types of correction methods: colors, levels, histogram, and suppress. When Levels is selected from the menu, you can remap the white and black points of an image, with a Gamma control to adjust midtones. A histogram provides a view of the tonal distribution in the image to help guide your adjustments. The Level method is described in detail below.

Range

Identical to the Range menu when Color is selected in the Menu, the Range menu determines the tonal range affected by the color correction controls in this tab. The menu can be set to Shadows, Midtones, Highlights, and Master, where Master is the default affecting the entire image.

The selected range is maintained throughout the Colors, Levels, and Suppress sections of the Color Corrector node.

Adjustments made to the image in the Master channel are applied to the image after any changes made to the Highlights, Midtones, and Shadows ranges.

> **NOTE:** The controls are independent for each color range. For example, adjusting the Gamma control while in Shadows mode does not change or affect the value of the Gamma control for the Highlights mode. Each control is independent and applied separately.

Channel

This menu is used to select and display the histogram for each color channel or for the Master channel.

Histogram Display

A histogram is a chart that represents the distribution of color values in the scene. The chart reads from left to right, with the leftmost values representing the darkest colors in the scene and the rightmost values representing the brightest. The more pixels in an image with the same or similar value, the higher that portion of the chart is.

Luminance is calculated per channel; therefore, the red, green, and blue channels all have their own histogram, and the combined result of these comprises the Master Histogram.

To scale the histogram vertically, place the mouse pointer inside the control and drag the pointer up to zoom in or down to zoom out.

Display Selector Toolbar

The Display Selector toolbar at the top of the histogram provides a method of enabling and disabling components of the histogram display. Hold the mouse pointer over the button to display a tooltip that describes the button's function.

— **Input Histogram:** This enables or disables the display of the input image's histogram.
— **Reference Histogram:** This enables or disables the display of the reference image's histogram.
— **Output Histogram:** This enables or disables the display of the histogram from the post-color-corrected image.
— **Corrective Curve:** This toggles the display of a spline used to visualize exactly how auto color corrections applied using a reference image are affecting the image. This can be useful when equalizing luminance between the input and reference images.

Histogram Controls

These controls along the bottom of the histogram display are used to adjust the input image's histogram, compressing or shifting the ranges of the selected color channel.

The controls can be adjusted by dragging the triangles beneath the histogram display to the left and right.

Shifting the High value toward the left (decreasing the value) causes the histogram to slant toward white, shifting the image distribution toward white. The Low value has a similar effect in the opposite direction, pushing the image distribution toward black.

Output Level

The Output Level control can apply clipping to the image, compressing the histogram. Decreasing the High control reduces the value of pixels in the image, sliding white pixels down toward gray and gray pixels toward black.

Adjusting the Low control toward High does the opposite, sliding the darkest pixels toward white.

If the low value were set to 0.1, pixels with a value of 0.0 would be set to 0.1 instead, and other values would increase to accommodate the change. The best way to visualize the effect is to observe the change to the output histogram displayed above.

Reset All Levels

Clicking this button resets all the controls in the Levels section to their defaults.
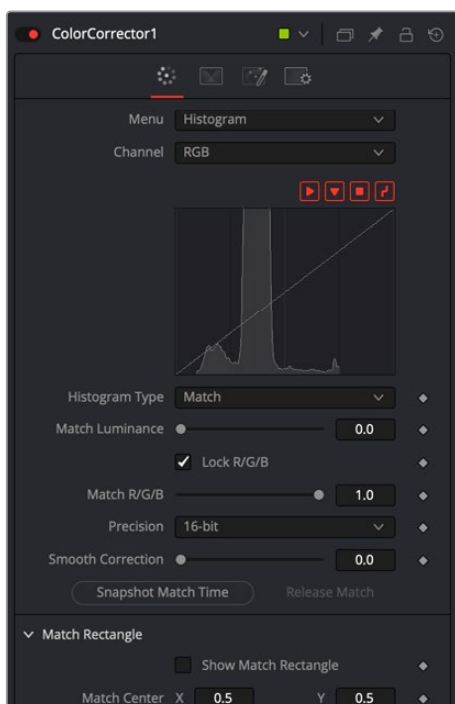
Color Correction in Histogram mode

Correction Tab Histogram Menu

When the menu is set to Histogram, a histogram display is produced of the input image. If a reference image is also provided, the histogram for the reference image is also displayed. The controls in this tab are primarily used to match one image to another, using either the Equalize or Match modes of the Color Corrector.

Channel

This menu is used to select and display the histogram for each color channel or for the Master channel.

Histogram Display

A histogram is a chart that represents the distribution of color values in the scene. The chart reads from left to right, with the leftmost values representing the darkest colors in the scene and the rightmost values representing the brightest. The more pixels in an image with the same or similar value, the higher that portion of the chart is.

Luminance is calculated per channel; therefore, the red, green, and blue channels all have their own histogram, and the combined result of these comprises the Master Histogram.

To scale the histogram vertically, place the mouse pointer inside the control and drag the pointer up to zoom in or down to zoom out.

Display Selector Toolbar

The Display Selector toolbar at the top of the histogram provides a method of enabling and disabling components of the histogram display. Hold the mouse pointer over the button to display a tooltip that describes the button's function.

— **Input Histogram:** This enables or disables the display of the input image's histogram.
— **Reference Histogram:** This enables or disables the display of the reference image's histogram.
— **Output Histogram:** This enables or disables the display of the histogram from the post-color-corrected image.
— **Corrective Curve:** This toggles the display of a spline used to visualize exactly how auto color corrections applied using a reference image are affecting the image. This can be useful when equalizing luminance between the input and reference images.

Float Images and Histogram Equalization or Matching

By using the Histogram Match or Equalize methods on a float image, the color depth of the output image is converted to 16-bit integer. Two-dimensional histograms are not well suited to working with the extreme dynamic range of float images, so these operations always revert to 16-bit integer processing.

Histogram Type

Each of these menu options enables a different type of color correction operation.

— **Keep:** Keep produces no change to the image, and the reference histogram is ignored.
— **Equalize:** Selecting Equalize adjusts the source image so that all the color values in the image are equally represented—in essence, flattening the histogram so that the distribution of colors in the image becomes more even.
— **Match:** The Match mode modifies the source image based on the histogram from the reference image. It is used to match two shots with different lighting conditions and exposures so that they appear similar.

  When selected, the Equalize and Match modes reveal the following controls.

— **Match/Equalize Luminance:** This slider affects the degree that the Color Corrector node attempts to affect the image based on its luminance distribution. When this control is zero (the default), matching and equalization are applied to each color channel independently, and the luminance, or combined value of the three color channels, is not affected.

  If this control has a positive value when equalizing the image, the input image's luminance distribution is flattened before any color equalization is applied.

If this control has a positive value when the correction mode is set to Match, the luminance values of the input are matched to the reference before any correction is applied to the R, G, and B channels.

The Luminance and RGB controls can have a cumulative effect, and generally they are not both set to full (1.0) simultaneously.

— **Lock R/G/B:** When this checkbox is selected, color matching is applied to all color channels equally. When the checkbox is not selected, individual controls for each channel appear.

Equalize/Match R/G/B

The name of this control changes depending on whether the Equalize or Match modes have been selected. The slider can be used to reduce the correction applied to the image to equalize or match it. A value of 1.0 causes the full effect of the Equalize or Match to be applied, whereas lower values moderate the result.

Precision

This menu determines the color fidelity used when sampling the image to produce the histogram. 10-bit produces higher fidelity than 8-bit, and 16-bit produces higher fidelity than 10-bit.

Smooth Correction

Often, color equalization and matching operations introduce posterization in an image, which occurs because gradients in the image have been expanded or compressed so that the dynamic range between colors is not sufficient to display a smooth transition. This control can be used to smooth the correction curve, blending some of the original histogram back into the result for a more even transition.

Snapshot Match Time

Click this button to take a freeze snapshot of the current reference histogram, storing its current state as a snapshot in memory. If the reference histogram is not snapshot, the reference histogram is updated from frame to frame. This can cause flickering and phasing of the correction as the node tries to match a changing source to a changing reference.

Release Match

Click this button to release the current snapshot of the histogram and return to using the live reference input.

Reset All Histogram Changes

Selecting this button removes all changes made to the histogram, returning the controls to default and setting the mode back to Keep.

Correction Tab Suppress Menu

Color Suppression provides a mechanism for removing an unwanted color component from the image. The Color Wheel control is similar to that shown in the Colors section of the node, but this one is surrounded by six controls, each representing a specific color along the wheel.

To suppress a color in the selected range, drag the control that represents that color toward the center of the color wheel. The closer the control is to the center, the more that color is suppressed from the image.

Color Corrector Suppression controls

Suppression Angle

Use the Suppression Angle control to rotate the controls on the suppression wheel and zero in on a specific color.

Reset All Suppression

Clicking this control resets the suppression colors to 1.0, the default value.
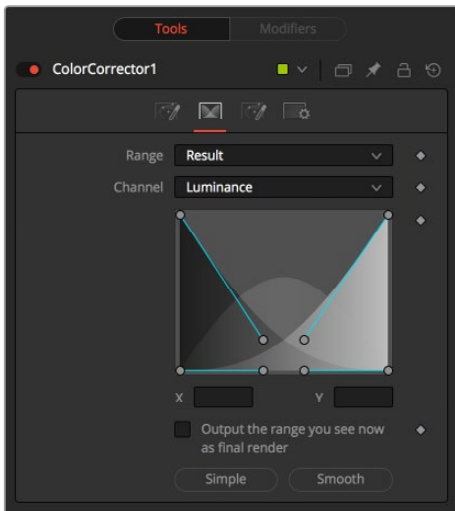
Color Corrector Luminance controls

Ranges Tab

The Ranges tab contains the controls used to specify which pixels in an image are considered to be shadows and which are considered to be highlights. The midrange is always calculated as pixels not already included in the shadows or the highlights.

Range

This menu is used to select the tonal range displayed in the viewers. They help to visualize the pixels in the range. When the Result menu option is selected, the image displayed by the color corrector in the viewers is that of the color corrected image. This is the default.

Selecting one of the other menu options switches the display to a grayscale image showing which pixels are part of the selected range. White pixels represent pixels that are considered to be part of the range, and black pixels are not in the range. For example, choosing Shadows would show pixels considered to be shadows as white and pixels that are not shadows as black. Mid gray pixels are only partly in the range and do not receive the full effect of any color adjustments to that range.

Channel

The Channel menu in this tab can be used to examine the range of a specific color channel. By default, Fusion displays the luminance channel when the color ranges are examined.

Spline Display

The ranges are selected by manipulating the spline handles. There are four spline points, each with one Bézier handle. The two handles at the top represent the start of the shadow and highlight ranges, whereas the two at the bottom represent the end of the range. The Bézier handles are used to control the falloff.

The midtones range has no specific controls since its range is understood to be the space between the shadow and the highlight ranges.

The X and Y text controls below the spline display can be used to enter precise positions for the selected Bézier point or handle.

Output the Range You See Now as Final Render

Selecting this checkbox causes the monochrome display of the range shown in the viewers to be output as the final render. Normally, the Color node outputs the full RGBA image, even if the node were left to display one of the color ranges in the view instead. This control makes it possible to use the Color Corrector node to generate a range's matte for use as an effect mask in other nodes.

Preset Simple/Smooth Ranges

These two buttons can be used to return the spline ranges to either Smooth (default) or Simple (linear) settings.
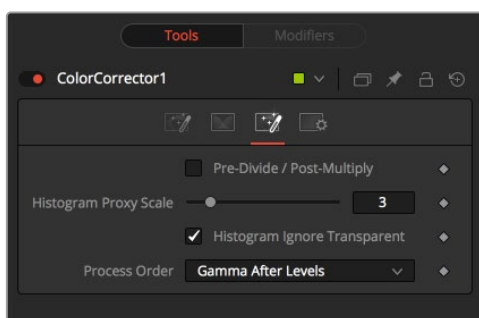
Color Corrector controls

Options Tab

The Options tab includes a few very important processing operations including a simple solution when color correcting premultiplied Alpha channels.

Pre-Divide/Post-Multiply

Selecting this option divides the color channels by the value of the Alpha before applying the color correction. After the color correction, the color values are re-multiplied by the Alpha to produce a

properly additive image. This is crucial when performing an additive merge or when working with CG images generated with premultiplied Alpha channels.

Histogram Proxy Scale

The Histogram Proxy Scale determines the precision used when creating and calculating histograms. Lower values represent higher precision, and higher values produce a rougher, generalized histogram.

Process Order

This menu is used to select whether adjustments to the image's gamma are applied before or after any changes made to the images levels.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Color Curves [CCv]
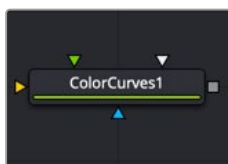
The Color Curves node

Color Curves Node Introduction

The Color Curves node is a spline-based node for performing Lookup table (LUT) color manipulations. A separate spline is provided for each color channel. The effect can be animated or dissolved and can be applied to the image using RGB, YUV, YIQ, CMY, or HLS color spaces.

The LUT view in the Color Corrector can be scaled using the + and - keys on the numeric keypad. The color curves LUT fully supports out-of-range values—i.e., pixels with color values above 1.0 or below 0.0.

The splines shown in this LUT view are also available from the Spline Editor, should greater precision be required when adjusting the controls.

Inputs

The Color Curves node includes three inputs in the Node Editor.

— **Input:** This orange input is the only required connection. It connects a 2D image that is adjusted by the color curves.

— **Effect Mask:** The optional effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the color curves adjustment to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

— **Reference Image:** The optional green input is used to connect a second 2D image that can be used for reference matching.

— **Match Mask:** This optional white input accepts any mask much like an effect mask. However, this mask defines of the area to match during a Match. It offers more flexibility in terms of shape than the built-in Match reference rectangle in the Inspector.

Basic Node Setup

The Color Curves node, like many 2D image-processing nodes, receives a 2D image like a Loader node or the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.

A Color Curves node applied to a MediaIn1 node
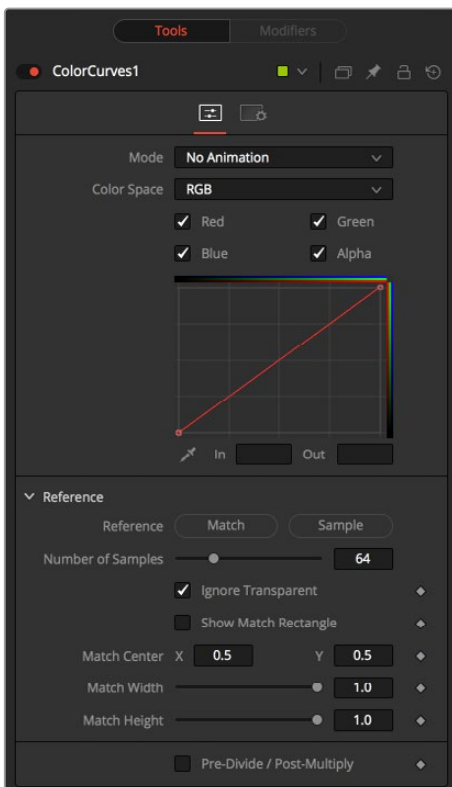
Inspector

Color Curves controls

Controls Tab

The Controls tab for the color curves is divided into two sections. The top half of the Inspector includes the curves and LUT controls. The bottom half is dedicated primarily to matching the reference image.

Mode

The Mode options change between Animated and Dissolve modes. The default mode is No Animation, where adjustments to the curves are static. Setting the mode provides a change spline for each channel, allowing the color curve to be animated over time.

Dissolve mode is essentially obsolete and is included for compatibility reasons only.

Color Space

The splines in the LUT view represent color channels from a variety of color spaces. The default is Red, Green, and Blue. The options in this menu allow an alternate color space to be selected.

A detailed description of the color spaces available here are below:

— **RGB (Red, Green, Blue):** Fusion uses the RGB color space, and most nodes and displays interpret the primary channels of an image as Red, Green, and Blue.

— **YUV (Luma, Blue Chroma, and Red Chroma):** The YUV color space is used in the analog broadcast of PAL video. Historically, this format was often used to color correct images, because of its familiarity to a large percentage of video engineers. Each pixel is described in terms of its Luminance, Blue Chroma, and Red Chroma components.

— **HLS (Hue, Luminance, and Saturation):** Each pixel in the HLS color space is described in terms of its Hue, Luminance, and Saturation components.

— **YIQ (Luma, In Phase, and Quadrature):** The YIQ color space is used in the analog broadcast of NTSC video. This format is much rarer than YUV and almost never seen in production. Each pixel is described in terms of its Luminance, Chroma (in-phase or red-cyan channel) and Quadrature (magenta-green) components.

— **CMY (Cyan, Magenta, and Yellow):** Although more common in print, the CMY format is often found in computer graphics from other software packages. Each pixel is described in terms of its Cyan, Magenta, and Yellow components. CMY is nonlinear.

Color Channels (RGBA)

Use the Color Channel controls to select which channel's spline is currently active for editing. The labels of these controls change to reflect the names of the channels for the current color space. Normally, they are read as Red, Green, and Blue. If the Color Curves node is operating in YUV color space, they are read as Y, U, and V instead.

These controls do not restrict the effect of the node to a specific channel. They only select whether the spline for that channel is editable. These controls are most often used to ensure that adding or moving points on one channel's spline do not unintentionally affect a different channel's spline.

Spline Window

The Spline Window displays a standard curve editor for each RGBA channel. These splines can be edited individually or as a group, depending on the color channels selected above.

The spline defaults to a linear range, from 0 in/0 out at the bottom left to the 1 in/1 out at the top right. At the default setting, a color processes to the same value as the output. If a point is added in the middle at 0.5 in/0.5 out, and the point is moved up, this raises the mid color of the image brighter.

The spline curves allow precise control over color ranges, so specific adjustments can be made without affecting other color values.

In and Out

Use the In and Out controls to manipulate the precise values of a selected point. To change a value, select a point and enter the in/out values desired.

Eyedropper (Pick)

Click the Eyedropper icon, also called the Pick button, and select a color from an image in the display to automatically set control points on the spline for the selected color. The new points are drawn with a triangular shape and can only be moved vertically (if point is locked, only the Out value can change).

Points are only added to enabled splines. To add points only on a specific channel, disable the other channels before making the selection.

One use for this technique is white balancing an image. Use the Pick control to select a pixel from the image that should be pure gray. Adjust the points that appear so that the Out value is 0.5 to change the pixel colors to gray.

Use the contextual menu's Locked Pick Points option to unlock points created using the Pick option, converting them into normal points.

Reference

The Reference section includes controls that handle matching to sample areas of the connected reference image.

— **Match Reference:** The Match Reference button adds points on the curve to match an image connected to the green reference image input. The number of points used to match the image is based on the Number of Samples slider below.

— **Sample Reference:** Clicking the Sample Reference button samples the center scanline of the background image and creates a LUT of its color values. The number of points used to match the samples scanline is based on the Number of Samples slider below.

— **Number of Samples:** This slider determines how many points are used to match the curve to the range in the reference image.

— **Show Match Rectangle:** Enabling this checkbox displays a rectangle in the viewer showing the area on the reference image used during the match process. The match rectangle affects only the result of the Match Reference operation. The Sample reference is always done from the center scaling of the image.

— **Match Center:** The X and Y parameters allow you to reposition the match rectangle to sample a different area when matching.

— **Match Width:** Width controls the width of the match rectangle.

— **Match Height:** Heigh controls the height of the match rectangle.

— **Pre-Divide/Post-Multiply:** Selecting this checkbox causes the image's pixel values to be divided by the Alpha values prior to the color correction, and then re-multiplied by the Alpha value after the correction. This helps to avoid the creation of illegally additive images, particularly around the edges of a blue/green key or when working with 3D-rendered objects.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Color Gain [Clr]
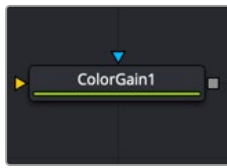
The Color Gain node

Color Gain Node Introduction

The Color Gain node contains options for adjusting the gain, gamma, saturation, and hue of the image. Many controls provided by the Color Gain node are also found in the Color Corrector node, but this simpler node may render more quickly. One feature that distinguishes the Color Gain node from the Color Corrector is its balance tab controls. These can be used to adjust the tinting of the colors in the highs, mids, and lows.

Inputs

The Color Gain node includes two inputs: one for the main image and the other for an effect mask.

— **Input:** This orange input is the only required connection. It connects a 2D image that gets adjusted by the color gain.
— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the color gain adjustment to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The Color Gain node, like many 2D image-processing nodes, receives a 2D image like a Loader node or the MediaIn1 shown below. The output continues the node tree by connecting to another 2D-image processing node or a Merge node.

A Color Gain node applied to a MediaIn1 node

Inspector

Color Gain controls

Gain Tab

The Gain tab provides control of individual RGBA Lift/Gamma/Gain parameters. These controls can quickly enable you to fix irregular color imbalances in specific channels.

Lock R/G/B

When selected, the Red, Green, and Blue channel controls for each effect are combined into one slider. Alpha channel effects remain separate.

Gain RGBA

The Gain RGBA controls multiply the values of the image channel in a linear fashion. All pixels are multiplied by the same factor, but the effect is larger on bright pixels and smaller on dark pixels. Black pixels do not change because multiplying any number times 0 is always 0.

Lift RGBA

While Gain scales the color values around black, Lift scales the color values around white. The pixel values are multiplied by the value of this control. A Lift of 0.5 makes a pixel that is R0.0 G0.0 B0.0 into R0.5 G0.5, B0.5, while leaving white pixels totally unaffected. Lift affects lower values more than it affects higher values, so the effect is strongest in the midrange and low range of the image.

Gamma RGBA

The Gamma RGBA controls affect the brightness of the midrange in the image. The effect of this node is nonlinear. White and black pixels in the image are not affected when gamma is modified, whereas pure grays are affected most by changes to this parameter. Large changes to this control tend to push midrange pixels into black or white, depending on the value used.

Pre-Divide/Post-Multiply

Selecting this checkbox causes the image pixel values to be divided by the Alpha values prior to the color correction, and then re-multiplied by the Alpha value after the correction. This helps when attempting to color correct images with premultiplied Alpha channels.

Saturation Tab

This Setting tab includes controls for the intensity of the colors in the individual RGB channels.
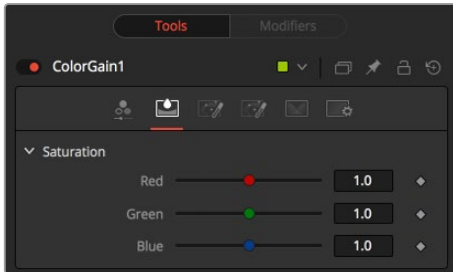
Color Gain Saturation setting tab

RGB Saturation

When adjusting an individual channel, a value of 0.0 strips out all that channel's color. Values greater than one intensify the color in the scene, pushing it toward the primary color.

Balance Tab

This tab in the Color Gain node offers controls for adjusting the overall balance of a color channel. Independent color and brightness controls are offered for the High, Mid, and Dark ranges of the image.

Colors are grouped into opposing pairs from the two dominant color spaces. Red values can be pushed toward Cyan, Green values toward Magenta and Blue toward Yellow. Brightness can be raised or lowered for each of the channels.
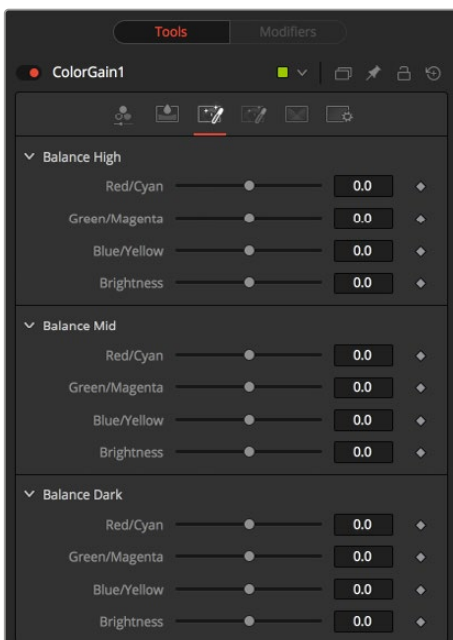
Color Gain Balance tab

CMY Brightness Highs/Mids/Darks

By default, the Balance sliders can be adjusted by -1 to +1, but values outside this range can be entered manually to increase the effect. A value of 0.0 for any slider indicates no change to the image channel. Positive and negative values indicate that the balance of the image channel has been pushed toward one color or the other in the pair.

Hue Tab

Use the Hue tab of the Color Gain node to shift the overall hue of the image, without affecting the brightness, or saturation. Independent controls of the High, Mid, and Dark ranges are offered by three sliders.

The following is the order of the hues in the RGB color space: Red, Yellow, Green, Cyan, Blue, Magenta and Red.
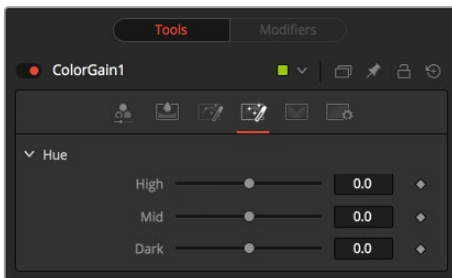
Color Gain Hue tab

High/Mid/Dark Hue

Values above 0 push the hue of the image toward the right (red turns yellow). Values below 0 push the hue toward the left (red turns magenta). At -1.0 or 1.0, the hue completes the cycle and returns to its original value.

The default range of the hue sliders is -1.0 to +1.0. Values outside this range can be entered manually.

Ranges Tab

The Ranges tab contains the controls used to specify which pixels in an image are considered to be shadows and which are considered to be highlights. The midrange is always calculated as pixels not included in either the shadows or the highlights.
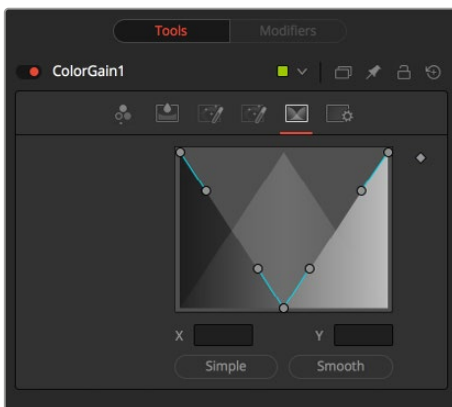
Color Gain Ranges tab

Spline Display

The ranges are selected by manipulating the spline handles. There are four spline points, each with one Bézier handle. The two handles at the top represent the start of the shadow and highlight ranges, whereas the two at the bottom represent the end of the range. The Bézier handles are used to control the falloff.

The midtones range has no specific controls since its range is understood to be the space between the shadow and the highlight ranges. The X and Y text controls below the Spline display can be used to enter precise positions for the selected Bézier point or handle.

Preset Simple/Smooth Ranges

These two buttons can be used to return the spline ranges to either Smooth (default) or Simple (linear) settings.

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Color Matrix [CMx]
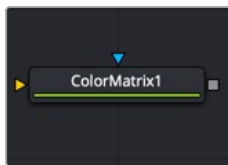
The Color Matrix node

Color Matrix Node Introduction

The ColorMatrix allows a vast number of operations to modify values individually in the different color channels.

Inputs

The Color Matrix node includes two inputs: one for the main image and the other for an effect mask.

— **Input:** This orange input is the only required connection. It connects a 2D image that is adjusted by the color matrix.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the color matrix adjustment to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The Color Matrix node, like many 2D image-processing nodes, receives a 2D image like a Loader node or the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.
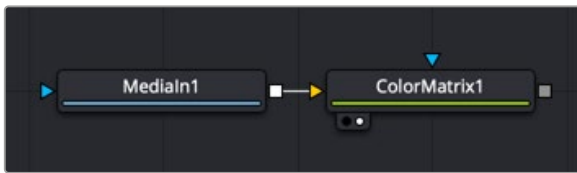
A Color Matrix node applied to a MediaIn1 node

Inspector
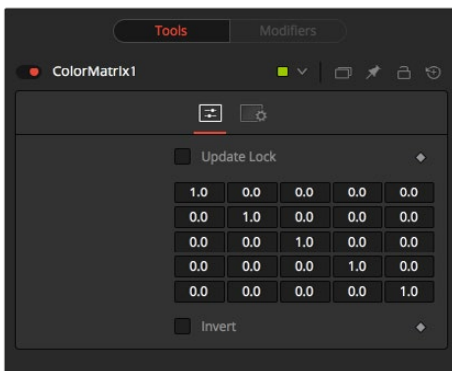
Color Matrix controls

Controls Tab

Color Matrix multiplies the RGBA channels based on the values entered in a 4 x 4 grid. The fifth column/row is an Add column.

Update Lock

When this control is selected, Fusion does not render the node. This is useful for setting up each value of the node, and then turning off Update Lock to render it.

Matrix

This defines what type of operation actually takes place. The horizontal rows define the output values of the node. From left to right, they are R, G, B, A, and Add. The vertical columns define the input values. From top to bottom, they are R, G, B, A, and Add. The Add column allows simple adding of values to the individual color channels.

By default, the output values are identical to the input values.

— 1.0 means 100% of the Red channel input is copied to the Red channel output.

— 1.0 means 100% of the Green channel input is copied to the Green channel output.

— 1.0 means 100% of the Blue channel input is copied to the Blue channel output.

— 1.0 means 100% of the Alpha channel input is copied to the Alpha channel output.

Written as mathematical equations, the default settings of the matrix would appear as follows:

```
[R out] = 1 * [R in] + 0 * [G in] + 0 * [B in] + 0 * [A in] + 0

[G out] = 0 * [R in] + 1 * [G in] + 0 * [B in] + 0 * [A in] + 0

[B out] = 0 * [R in] + 0 * [G in] + 1 * [B in] + 0 * [A in] + 0

[A out] = 0 * [R in] + 0 * [G in] + 0 * [B in] + 1 * [A in] + 0
```

Invert

Enabling this option inverts the Matrix. Think of swapping channels around, doing other operations with different nodes, and then copying and pasting the original ColorMatrix and setting it to Invert to get your channels back to the original.

Example 1: Invert

If you want to do a simple invert or negative of the color values, but leave the Alpha channel untouched, the matrix would look like this:
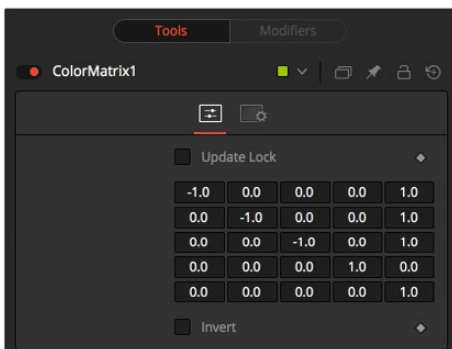
Color Matrix example

Observe the fact that we have to add 1 to each channel to push the inverted values back into the positive numbers.

Let's follow this example step by step by viewing the waveform of a 32-bit grayscale gradient.

1   The original grayscale.
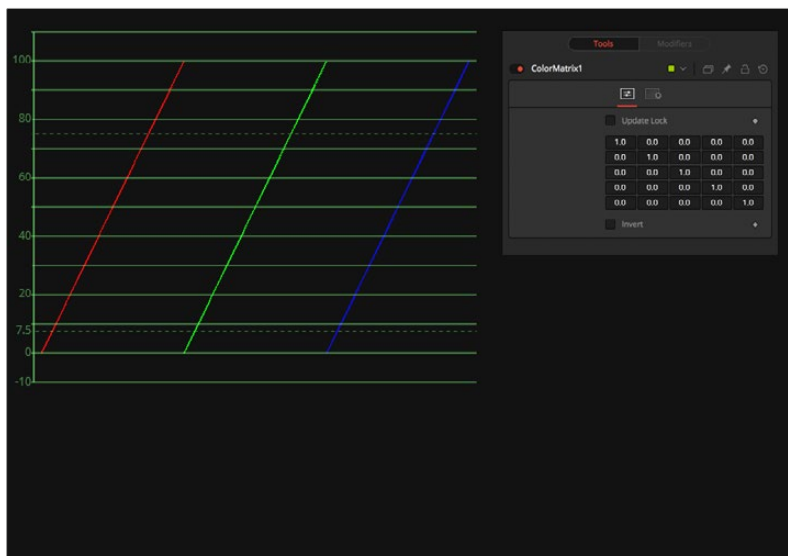
Original Grayscale

2   RGB set to -1. The values get inverted but fall below 0.
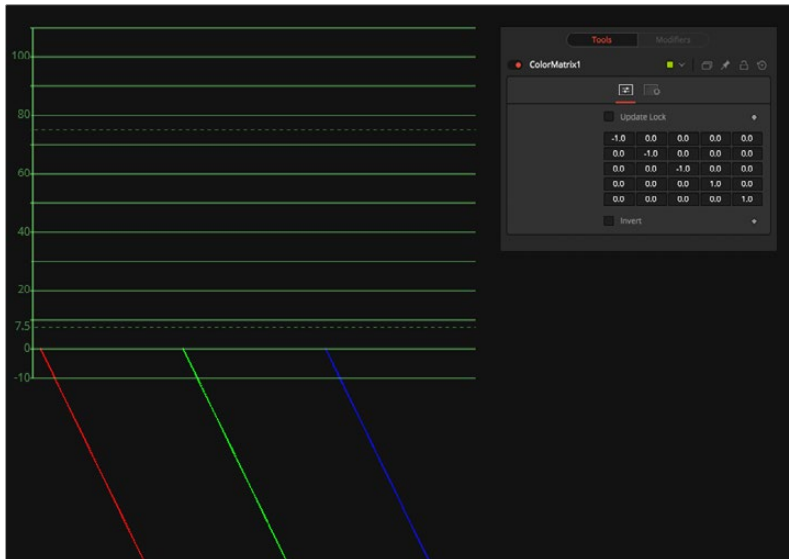
RGB set to -1

3   Adding 1 to each channel keeps the inversion but moves the values back into a positive range.
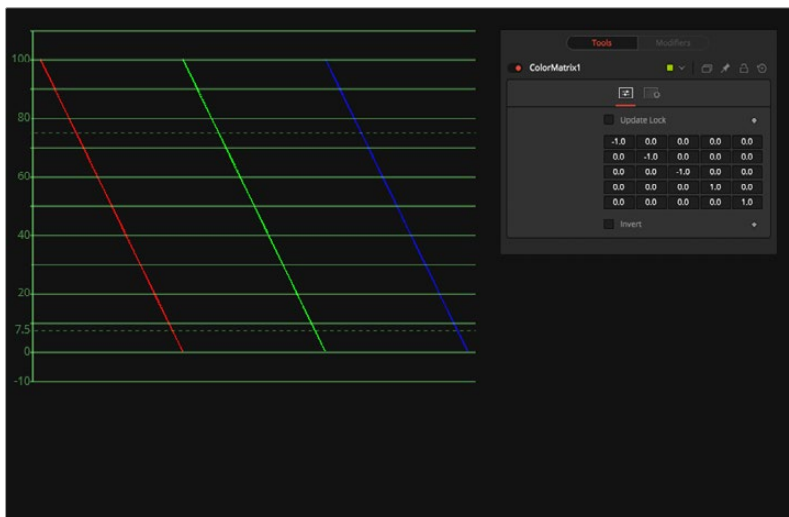
Adding 1 to each channel

Example 2: Brightness per Channel

This example influences the brightness of each channel individually. This subtracts 0.2 from the red channel, adds 0.314 to the green channel, and adds 0.75 to the blue channel, while keeping Alpha as it is.
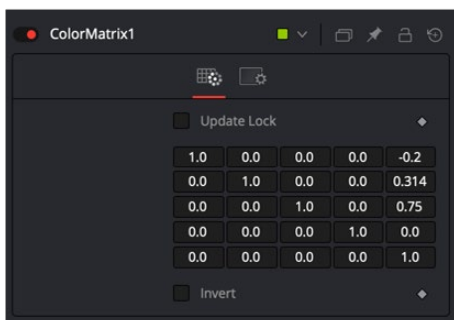
Brightness per channel example

Example 3: Copying Values

You can also copy color values back and forth between individual channels. In this example, the red channel contains the luminance values of the image based on thirds, and the green channel contains the luminance values based on the proper black-and-white conversion method, whereas the blue channel uses a third method based on getting more information from red and less from blue. The blue channel's brightness is also lowered by 0.1, and the Alpha channel is replaced with the original blue channel.
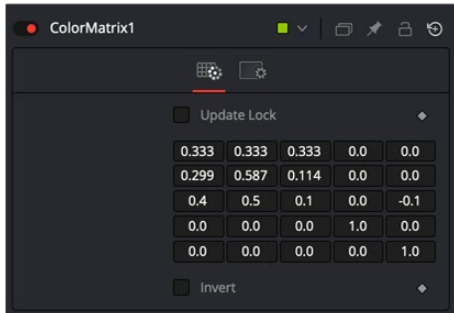
Copying Values example

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Color Space [CS]
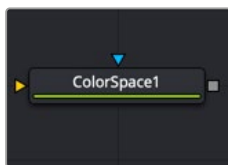
The Color Space node

Color Space Node Introduction

The Color Space node provides the ability to work on an image in a variety of alternate color space formats. By default, Fusion uses the RGB color space, and most nodes and displays interpret the primary channels of an image as Red, Green, and Blue.

Changing the color space from RGB causes most images to look odd, as Fusion's viewers still interpret the primary channels as Red, Green, and Blue. For example, viewing an image converted to YUV in one of the viewers shows the Y channel as Red, the U channel as Green, and the V channel as Blue.

Several common elements of the Fusion interface refer to the RGB channels directly. The four buttons commonly found on the Inspector's Settings tab to restrict the effect of the node to a single color channel are one example. When a conversion is applied to an image, the labels of these buttons remain R, G, and B, but the values they represent are from the current color space. (For example, Red is Hue, Green is Luminance, and Blue is Saturation for an RGB to HLS conversion. The Alpha value is never changed by the color space conversion.)

Inputs

The Color Space node includes two inputs: one for the main image and the other for an effect mask.

— **Input:** This orange input is the only required connection. It connects a 2D image that is converted by the color space operation.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the color space adjustment to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The Color Space node, like many 2D image-processing nodes, receives a 2D image like a Loader or the MediaIn1 shown below, processes the image, and then extends the node tree by connecting to another 2D image-processing node or a Merge node.

A Color Space node applies a conversion to a MediaIn1 node

Inspector
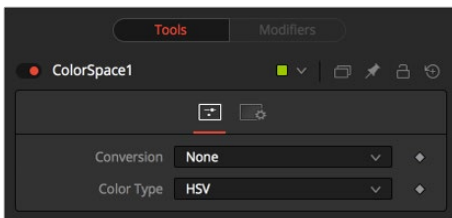
Color Space controls

Controls Tab

The Controls tab in the Color Space node consists of two menus. The top Conversion menu determines whether you are converting an image to RGB or from RGB. The bottom menu selects the alternative color space you are either converting to or from.

Conversion

This menu has three options. The None option has no effect on the image. When To Color is selected, the input image is converted to the color space selected in the Color Type control found below. When To RGB is selected, the input image is converted back to the RGB color space from the type selected in the Color Type menu (for example, YUV to RGB).

Color Type

This menu is used to select the color space conversion applied when the To Color conversion is selected. When the To RGB option is selected in the Conversion menu, the Color Type option should reflect the input image's current color space. There are eight color space options to choose from.

— **HSV (Hue, Saturation, and Value):** Each pixel in the HSV color space is described in terms of its Hue, Saturation, and Value components. Value is defined as the quality by which we distinguish a light color from a dark one or brightness. Decreasing saturation roughly corresponds to adding white to a paint chip on a palette. Increasing Value is roughly similar to adding black.

— **YUV (Luma, Blue Chroma, and Red Chroma):** The YUV color space is used in the analog broadcast of PAL video. Historically, this format was often used to color correct images because of its familiarity to a large percentage of video engineers. Each pixel is described in terms of its Luminance, Blue Chroma, and Red Chroma components.

— **YIQ (Luma, In Phase, and Quadrature):** The YIQ color space is used in the analog broadcast of NTSC video. This format is much rarer than YUV and almost never seen in production. Each pixel is described in terms of its Luminance, Chroma (in-phase or red-cyan channel), and Quadrature (magenta-green) components.

— **CMY (Cyan, Magenta, and Yellow):** Although more common in print, the CMY format is often found in computer graphics from other software packages. Each pixel is described in terms of its Cyan, Magenta, and Yellow components. CMY is nonlinear.

— **HLS (Hue, Luminance, and Saturation):** Each pixel in the HLS color space is described in terms of its Hue, Luminance, and Saturation components. The differences between HLS and HSV color spaces are minor.

— **XYZ (CIE Format):** This mode is used to convert a CIE XYZ image to and from RGB color spaces. CIE XYZ is a weighted space, instead of a nonlinear one, unlike the other available color spaces. Nonlinear in this context means that equal changes in value at different positions in the color space may not necessarily produce the same magnitude of change visually to the eye.

Expressed simply, the CIE color space is a perceptual color system, with weighted values obtained from experiments where subjects were asked to match an existing light source using three primary light sources.

This color space is most often used to perform gamut conversion and color space matching between image display formats because it contains the entire gamut of perceivable colors.

— **Negative:** The color channels are inverted. The color space remains RGBA.

— **BW:** The image is converted to black and white. The contribution of each channel to the luminance of the image is adjustable via slider controls that appear when this option is selected. The default values of these sliders represent the usual perceptual contribution of each channel to an image's luminance. The color space of the image remains RGBA.

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Copy Aux [CpA]

The Copy Aux node

Copy Aux Node Introduction

The Copy Aux node is used to shuffle channels between visible channels and auxiliary data channels in a single 2D image. Typically, these auxiliary channels are rendered from 3D applications. Auxiliary channels supported in the Copy Aux node include background color, z-depth, texture coordinates, coverage, object ID, material ID, normals, vectors, back vectors, and world position.

The Copy Aux node is mostly a convenience node, as the copying can also be accomplished with more effort using a Channel Booleans node. Where Channel Booleans deals with individual channels, Copy Aux deals with channel groups. By default, the Copy Aux node automatically promotes the depth of its output to match the depth of the aux channel.

Copy Aux also supports static normalization ranges. The advantage of static normalization versus the dynamic normalization that Fusion's viewers do is that colors remain constant over time. For example, if you are viewing Z or WorldPos values for a ball, you see a smooth gradient from white to black. Now imagine that some other 3D object is introduced into the background at a certain time. Dynamic normalization turns the ball almost completely white while the background object is now the new black. Dynamic normalization also causes flicker problems while viewing vector/disparity channels, which can make it difficult to compare the aux channels of two frames at different times visually.

Inputs

The Copy Aux node includes two inputs: one for the main image and the other for an effect mask.

— **Input:** This orange input is the only required connection. It connects a 2D image for the Copy Aux node operation.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Copy Aux operation to only those pixels within the mask. An effect mask is applied to the tool after the tool is processed.

Basic Node Setup

The Copy Aux node, like many 2D image-processing nodes, receives a 2D image like a Loader node or the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.
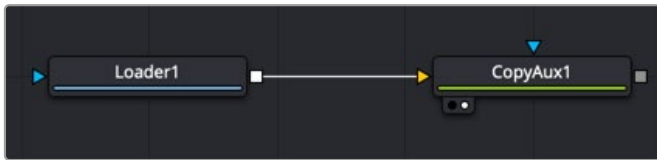
A Copy Aux node applied to a MediaIn1 node

Inspector
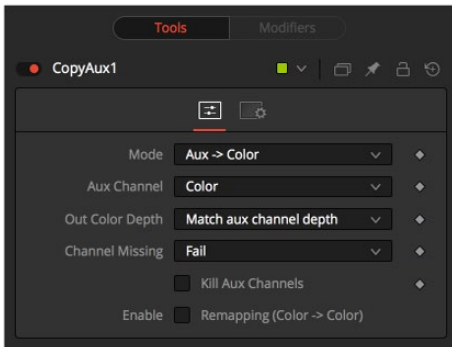
Copy Aux controls

Controls Tab

The Controls tab is used to copy auxiliary channel groups into RGBA channels. Although Copy Aux has quite a few options, most of the time you select only the channel to copy and ignore the remaining functionality.

Mode

The Mode menu determines whether the auxiliary channel is copied into the RGBA color channel (Aux to Color) or vice versa (Color to Aux). Using this option, you can use one Copy Aux node to bring an auxiliary channel into color, do some compositing operations on it, and then use another Copy Aux node to write the color back into the auxiliary channel. When the Mode is set to Color to Aux, all the options in the Controls tab except the Aux Channel menu are hidden.

Aux Channel

The Aux Channel menu selects the auxiliary channel to be copied from or written to depending on the current mode. When the aux channel abcd has one valid component, it is copied as aaa1, two valid components as ab01, three valid components as abc1, and four components as abcd. For example, the Z-channel is copied as zzz1, texture coordinates as uv01, and normals as nxnynz1.

Out Color Depth

Out Color Depth controls the color depth of the output image. Most aux channels contain float values or, if they are integer valued, they can contain values beyond 255. When you copy float values into an int8 or int16 image, this can be a problem since negative values and values over 1.0 can get clipped. In addition, precision can be lost. This option determines what happens if the depth of RGBA channels of the input image is insufficient to contain the copied aux channel.

Be careful when copying float channels into integer image formats, as they can get clipped if you do not set up Copy Aux correctly. For this node, all aux channels are considered to be float32 except ObjectID or MaterialID, which are considered to be int16.

— **Match Aux Channel Depth:** The bit depth of the RGBA channels of the output image is increased to match the depth of the aux channel. Specifically, this means that the RGBA channels of the output image are either int16 or float32. Be careful when using this option because, for example, if you normally have int8 color channels, you are now using 2x or 4x more memory for the color channels. Particularly, the Z, Coverage, TextureCoordinate, Normal, Vector, BackVector, WorldPosition, and Disparity channels are always output as float, and the Material/ObjectID channels are output as int16.

— **Match Source Color Depth:** The bit depth of the RGBA channels of the output image is the same as the input image. This can have some unexpected consequences. For example, if your input image is int8, the XYZ components of normals that are floating-point numbers in the [-1, 1] range are clipped to non-negative numbers [0, 1] range. As a more extreme example, consider what happens to Z values. Z values are floating-point numbers stored in the [-1e30, 0] range, and they all get truncated to the [0, 1] range, which means your Z-channel is full of zeroes.

— **Force Float32:** The bit depth of the RGBA channels of the output image is always float32.

Channel Missing

Channel Missing determines what happens if a channel is not present. For example, this determines what happens if you chose to copy Disparity to Color and your input image does not have a Disparity aux channel.

— **Fail:** The node fails and prints an error message to the console.

— **Use Default Value:** This fills the RGBA channels with the default value of zero for everything except Z, which is -1e30.

Kill Aux Channels

When this is checked, Copy Aux copies the requested channel to RGBA and then outputs a resulting image that is purely RGBA with other channels being killed. This is useful if you want to increase the number of frames of Copy Aux that can be cached for playback—for example, to play back a long sequence of disparity. A handy tip is that you can use the Kill Aux feature also with just Color to Aux > Color for a longer color playback.
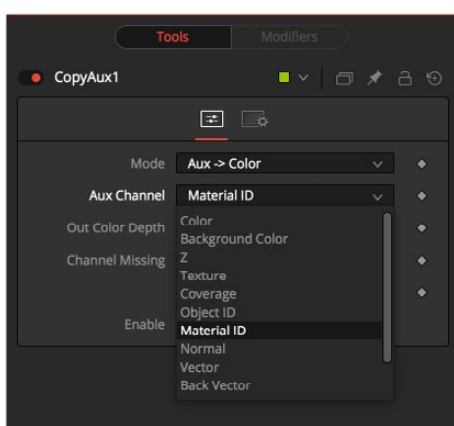
Kill Aux channels

Enable Remapping

When remapping is enabled, the currently selected aux channel is rescaled, linearly mapping the range according to the From and To slider selections as explained below. The Remapping options are applied before the conversion operation. This means you could set the From > Min-Max values to -1, 1 to rescale your normals into the [0, 1] range, or set them to [-1000, 0] to rescale your Z values from [-1000, 0] into the [0, 1] range before the clipping occurs.

Note that the Remapping options are per channel options. That means the default scale for normals can be set to [-1, +1] > [0, 1] and for Z it can be set [-1000, 0] > [0, 1]. When you flip between normals and Z, both options are remembered. One way this could be useful is that you can set up the remapping ranges and save this as a setting that you can reuse. The remapping can be useful to squash the aux channels into a static [0, 1] range for viewing or, for example, if you wish to compress normals into the [0, 1] range to store them in an int8 image.

— **From > Min:** This is the value of the aux channel that corresponds to To > Min.
— **From > Max:** This is the value of the aux channel that corresponds to To > Max. It is possible to set the max value less than the min value to achieve a flip/inversion of the values.
— **Detect Range:** This scans the current image to detect the min/max values and then sets the From > Min/ From > Max Value controls to these values.
— **Update Range:** This scans the current image to detect the min/max values and then enlarges the current [From > Min, From > Max] region so that it contains the min/max values from the scan.
— **To > Min:** This is the minimum output value, which defaults to 0.
— **To > Max:** This is the maximum output value, which defaults to 1.
— **Invert:** After the values have been rescaled into the [To > Min, To > Max] range, this inverts/flips the range.

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Gamut [Gmt]
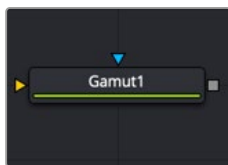
The Gamut node

Gamut Node Introduction

The Gamut node has controls to transform one color space to another and remove/add gamma curves. This node, along with the Cineon Log node, is primarily used to linearize incoming images and then reapply the applicable output gamma curve at the end of a node tree.

Inputs

The Gamut node includes two inputs: one for the main image and the other for an effect mask to limit the conversion area.

— **Input:** This orange input is the only required connection. It connects a 2D image output that is the source of the gamut conversion.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Gamut operation to only those pixels within the mask. An effect mask is applied to the tool after the tool is processed.

Basic Node Setup

A Gamut node is most often placed directly after the MediaIn node in DaVinci Resolve or a Loader node in Fusion Studio. Another Gamut node is usually placed at the end of a node tree before a MediaOut node in DaVinci Resolve or a Saver node in Fusion Studio.
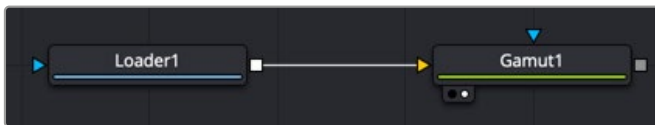
A Gamut node applied to a MediaIn1 node

Inspector
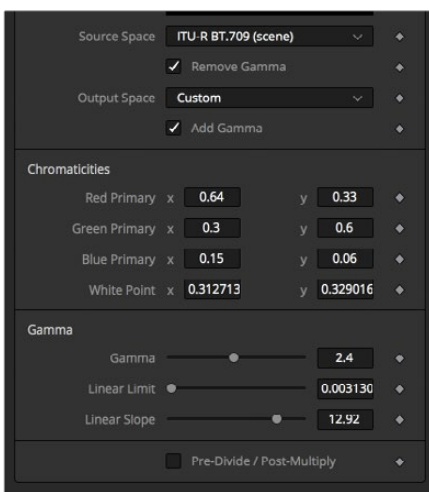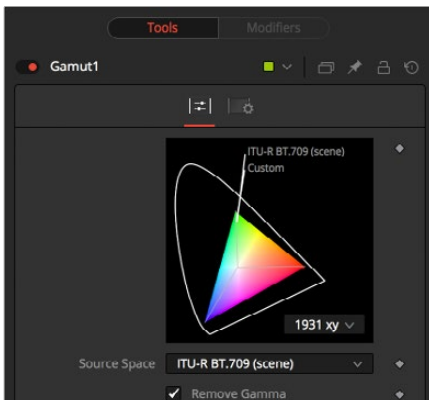
Gamut controls

Controls Tab

The Controls tab is where all the conversion operations take place. It has a section for incoming images and a section for the node's output. Which section you use depends on whether you are stripping an image of a gamma curve to make it linear or converting a linear image to a specific color space and gamma curve for output.

Source Space

Source Space determines the input color space of the image. When placed directly after a Loader node in Fusion or a MediaIn node in DaVinci Resolve, you would select the applicable color space based on how the image was created and check the Remove Gamma checkbox. The output of the node would be a linearized image. You leave this setting at No Change when you are adding gamma using the Output Space control and placing the node directly before the Saver node in Fusion or a MediaOut node in DaVinci Resolve.

DCI-P3

The DCI-P3 color space is most commonly used in association with DLP projectors. It is frequently provided as a color space available with DLP projectors and as an emulation mode for 10-bit LCD monitors such as the HP Dreamcolor and Apple's Pro Display XDR. This color space is defined in the SMPTE-431-2 standard.

Custom

The Custom gamut allows you to describe the color space according to CIE 1931 primaries and white point, which are expressed as XY coordinates, as well as by gamma, limit, and slope. For example, the DCI-P3 gamut mentioned above would have the following values if described as a Custom color space.

|		
Red Primary	0.68	0.32
Green Primary	0.265	0.69
Blue Primary	0.15	0.06
White Point	0.314	0.351
Gamma	2.6	–
Linear Limit	0.0313	–

To understand how these controls work, you could view the node attached to a gradient background in Waveform mode and observe how different adjustments modify the output.

Output Space

Output Space converts the gamut to the desired color space. For instance, when working with linearized images in a composite, you place the Gamut node just before the Saver node and use the Output Space to convert to the gamut of your final output file. You leave this setting at No Change when you want to remove gamma using the Source Space control.

> **NOTE:** When outputting to HD specification Rec. 709, Fusion uses the term Scene to refer to a gamma of 2.4 and the term Display for a gamma of 2.2.

Remove/Add Gamma

Select these checkboxes to do the gamut conversion in a linear or nonlinear gamma, or simply remove or add the applicable gamma values without changing the color space.

Pre-Divide/Post-Multiply

Selecting this checkbox causes the image's pixel values to be divided by the Alpha values prior to the color correction, and then re-multiplied by the Alpha value after the correction. This helps to avoid the creation of illegally additive images, particularly around the edges of a blue/green key or when working with 3D-rendered objects.

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Hue Curves [HCv]
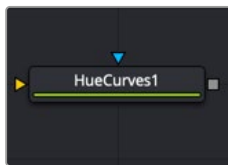
The Hue Curves node

Hue Curves Node Introduction

The Hue Curves node allows you to adjust the color in an image using a series of spline curves. Splines are provided to control the image's hue, saturation, and luminance as well as each individual color channel. An additional set of curves allows you to apply suppression to individual color channels.

The advantage of the Hue Curves node over other color correction nodes in Fusion is that the splines can be manipulated to restrict the node's effect to a very narrow portion of the image, or expanded to include a wide-ranging portion of the image. Additionally, these curves can be animated to follow changes in the image over time. Since the primary axis of the spline is defined by the image's hue, it is much easier to isolate a specific color from the image for adjustment.

Inputs

The Hue Curves node includes two inputs: one for the main image and the other for an effect mask to limit the color correction area.

— **Input:** This orange input is the only required connection. It connects a 2D image for the Hue Curves color correction.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Hue Curves operation to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The Hue Curves node, like many 2D image-processing nodes, receives a 2D image like a Loader node or the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.
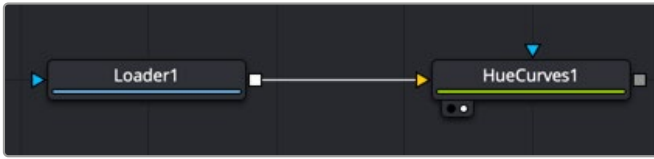
A Hue Curves node applied to a Loader node in Fusion Studio

Inspector
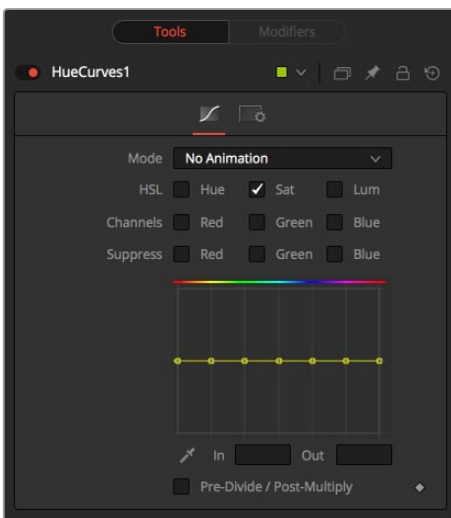
Hue Curves controls

Controls Tab

The Controls tab consists of color attribute checkboxes that determine which splines are displayed in the Spline window. The spline graph runs horizontally across with control points placed horizontally at each of the primary colors. You can manipulate these control points to change the selected color attribute.

Mode

The Mode options change between No Animation and Animated Points modes. The default mode is No Animation, where adjustments to the curves are applied consistently over time. Setting the Mode to Animated Points or Dissolve allows the color curve to be animated over time.

Dissolve mode is essentially obsolete and is included for compatibility reasons only.

Color Channel Checkboxes

These checkboxes define which splines are editable and are included when using the Eyedropper to pick a color in the image.

Any number of activated splines can be edited simultaneously; however it's more convenient to have only the currently modified spline active to avoid unwanted changes to other splines.

When using the Eyedropper icon, a point is created on all active splines, representing the selected color.

Spline Window

This graph display is the main interface element of the Hue Curves node, which hosts the various splines. In appearance, the node is very similar to the Color Curves node, but here the horizontal axis represents the image's hue, while the vertical axis represents the degree of adjustment. The Spline window shows the curves for the individual channels. It is a miniature Spline Editor. In fact, the curves shown in this window can also be found and edited in the Spline Editor.

The spline curves for all components are initially flat, with control points placed horizontally at each of the primary colors. From left to right, these are: Red, Yellow, Green, Cyan, Blue, and Magenta. Because of the cyclical design of the hue gradient, the leftmost control point in each curve is connected to the rightmost control point of the curve.

Right-clicking in the graph displays a contextual menu containing options for resetting the curves, importing external curves, adjusting the smoothness of the selected control points, and more.

In and Out

Use the In and Out controls to manipulate the precise values of a selected point. To change a value, select a point and enter the In/Out values desired.

Eyedropper

Left-clicking and dragging from the Eyedropper icon changes the current mouse cursor to an Eyedropper. While still holding down the mouse button, drag the cursor to a viewer to pick a pixel from a displayed image. This causes control points, which are locked on the horizontal axis, to appear on the currently active curves. The control points represent the position of the selected color on the curve. Use the contextual menu's Lock Selected Points toggle to unlock points and restore the option of horizontal movement.

Points are only added to enabled splines. To add points only on a specific channel, disable the other channels before making the selection.

Pre-Divide/Post-Multiply

Selecting this checkbox causes the image's pixel values to be divided by the Alpha values prior to the color correction, and then re-multiplied by the Alpha value after the correction. This helps when color correcting images that include a premultiplied Alpha channel.

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

OCIO CDL Transform [OCD]

The OCIO CDL Transform node

OCIO CDL Transform Node Introduction

Fusion supports the Open Color IO color management workflow by way of three OCIO nodes.

— The OCIO CDL Transform node allows you to create, save, load, and apply a Color Decision List (CDL) grade.
— The OCIO Color Space allows sophisticated color space conversions, based on an OCIO config file.
— The OCIO File Transform allows you to load and apply a variety of Lookup tables (LUTs).

Generally, the OCIO color pipeline is composed from a set of color transformations defined by OCIO-specific config files, commonly named with a ".ocio" extension. These config files allow you to share color settings within or between facilities. The path to the config file to be used is normally specified by a user-created environment variable called "OCIO," although some tools allow overriding this. If no other *.ocio config files are located, the DefaultConfig.ocio file in Fusion's LUTs directory is used.

For in-depth documentation of the format's internals, please refer to the official pages on opencolorio.org.

Inputs

The OCIO CDL Transform node includes two inputs: one for the main image and the other for an effect mask to limit the area where the CDL is applied.

— **Input:** This orange input is the only required connection. It connects a 2D image output for the CDL grade.
— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the CDL grade to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The OCIO CDL Transform node is often applied after a Gamut node converts the Loader to linear color in Fusion Studio.

A OCIO CDL Transform node applied to a Loader node
after a Gamut conversion to linear in Fusion Studio

Inspector
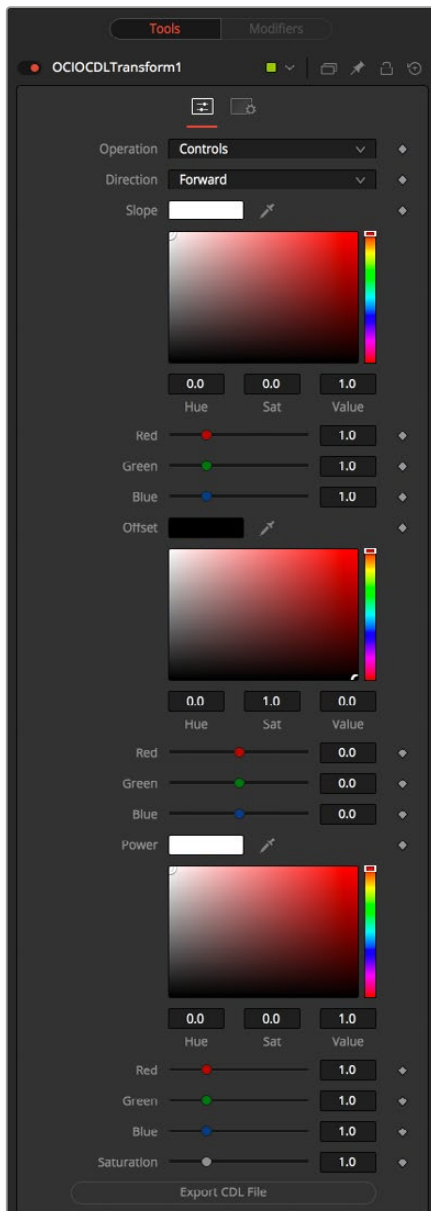
OCIO Transform controls

Controls Tab

The Controls tab for the OCIO CDL Transform contains primary color grading color correction controls in a format compatible with CDLs. You can make R, G, B adjustments based on the Slope, Offset, and Power. There is also overall Saturation control. You can also use the Controls tab to import and export the CDL compatible adjustments.

Operation

This menu switches between File and Controls. In File mode, standard ASC-CDL files can be loaded. In Controls mode, manual adjustments can be made to Slope, Offset, Power, and Saturation, and the CDL file can be saved.

> **NOTE:** Using DaVinci Resolve terminology, slope is similar to gain. It controls mids-to-high contrast. Offset is the overall offset of color balance and exposure. Power is very similar to contrast with a raised pivot, giving you control over shadow contrast.

Direction

Toggles between Forward and Reverse. Forward applies the corrections specified in the node, while Reverse tries to remove those corrections. Keep in mind that not every color correction can be undone.

Imagine that all slope values have been set to 0.0, resulting in a fully black image. Reversing that operation is not possible, neither mathematically nor visually.

Slope

Slope multiplies the color values; this is the same as Gain in the Brightness Contrast node.

Offset

Offset adds to the color values; this is the same as Brightness in the Brightness Contrast node.

Power

Applies a Gamma Curve. This is an inverse of the Gamma function of the Brightness Contrast node.

Saturation

Enhances or decreases the color saturation. This works the same as Saturation in the Brightness Contrast node.

Export File

Allows the user to export the settings as a CDL file.

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

OCIO Color Space [OCC]
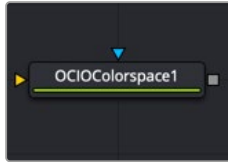
The OCIO Color Space node

OCIO Color Space Node Introduction

Fusion supports the Open Color IO color management workflow by way of three OCIO nodes.

— The OCIO CDL Transform node allows you to create, save, load, and apply a Color Decision List (CDL) grade.

— The OCIO Color Space allows sophisticated color space conversions, based on an OCIO config file.

— The OCIO File Transform allows you to load and apply a variety of Lookup tables (LUTs).

Generally, the OCIO color pipeline is composed from a set of color transformations defined by OCIO-specific config files, commonly named with a ".ocio" extension. These config files allow you to share color settings within or between facilities. The path to the config file to be used is normally specified by a user-created environment variable called "OCIO," though some tools allow overriding this. If no other *.ocio config files are located, the DefaultConfig.ocio file in Fusion's LUTs directory is used.

For in-depth documentation of the format's internals, please refer to the official pages on opencolorio.org.

Sample configs can be obtained from https://opencolorio.readthedocs.io/en/latest/configurations/_index.html#configurations

The functionality of the OCIO Color Space node is also available as a View LUT node from the View LUT menu.

Inputs

The OCIO Color Space node includes two inputs: one for the main image and the other for an effect mask to limit the area where the color space conversion is applied.

— **Input:** This orange input is the only required connection. It connects a 2D image for the color space conversion.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the color space conversion to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The OCIO Color Space node is typically placed directly after a MediaIn node in DaVinci Resolve or a Loader node in Fusion Studio. Another OCIO Color Space node is placed just before a Media Out node in DaVinci Resolve or a Saver node in Fusion Studio.

An OCIO Color Space node applied to a Loader node and a Saver node in Fusion Studio

Inspector
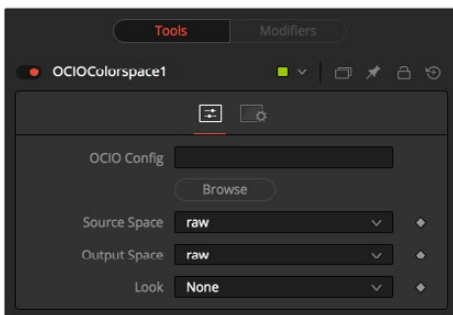
OCIO Color Space controls

Controls Tab

The Controls tab for the OCIO Color Space node allows you to convert an image from one color space to another based on an OCIO config file. By default, it uses the config file included with Fusion; however, the Controls tab does allow you to load your own config file as well.

OCIO Config

Displays a File > Open dialog to load the desired config file.

Source Space

Based on the config file, the available source color spaces are listed here.

The content of this list is based solely on the loaded profile and hence can vary immensely. If no other OCIO config file is loaded, the DefaultConfig.ocio file in Fusion's LUTs directory is used to populate this menu.

Output Space

Based on the config file, the available output color spaces are listed here.

The content of this list is based solely on the loaded profile and hence can vary immensely. If no other OCIO config file is loaded, the DefaultConfig.ocio file in Fusion's LUTs directory is used to populate this menu.

Look

Installed OCIO Color Transform Looks appear in this menu. If no looks are installed, this menu has only None listed as an option.

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

OCIO File Transform [OCF]

The OCIO File Transform node

OCIO File Transform Node Introduction

Fusion supports the Open Color IO color management workflow by way of three OCIO nodes.

— The OCIO CDL Transform node allows you to create, save, load,
and apply a Color Decision List (CDL) grade.

— The OCIO Color Space allows sophisticated color space conversions, based on an OCIO config file.

— The OCIO File Transform allows you to load and apply a variety of Lookup tables (LUTs).

Generally, the OCIO color pipeline is composed from a set of color transformations defined by OCIO-specific config files, commonly named with a ".ocio" extension. These config files allow you to share color settings within or between facilities. The path to the config file to be used is normally specified by a user-created environment variable called "OCIO," though some tools allow overriding this. If no other *.ocio config files are located, the DefaultConfig.ocio file in Fusion's LUTs directory is used.

For in-depth documentation of the format's internals, please refer to the official pages on opencolorio.org.

The functionality of the OCIO File Transform node is also available as a View LUT node from the View LUT menu.

Inputs

The OCIO File Transform node includes two inputs: one for the main image and the other for an effect mask to limit the area where the color space conversion is applied.

— **Input:** This orange input is the only required connection. It connects a 2D image for the LUT.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the applied LUT to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The OCIO File Transform node is often applied after a Gamut node converts the Loader to linear color in Fusion Studio.

An OCIO File Transform node applied to a Loader node
after a Gamut node conversion to linear color space

Inspector
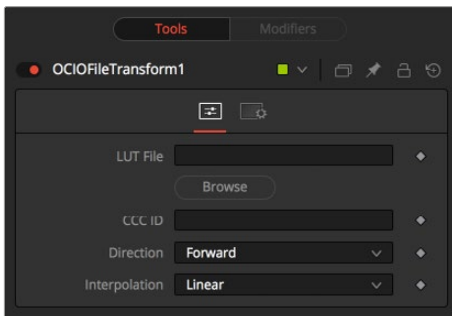
OCIO File Transform controls

Controls Tab

The Controls tab for the OCIO File Transform node includes options to import the LUT, invert the transform, and select the color interpolation method.

LUT File

Displays a File > Open dialog to load the desired LUT.

CCC ID

This is the ID key used to identify the specific file transform located within the ASC CDL color correction XML file.

Direction

Toggles between Forward and Reverse. Forward applies the corrections specified in the node, while Reverse tries to remove those corrections. Keep in mind that not every color correction can be undone. Imagine that all slope values have been set to 0.0, resulting in a fully black image. Reversing that operation is not possible, neither mathematically nor visually.

Interpolation

Allows the user to select the color interpolation to achieve the best quality/render time ratio. Nearest is the fastest interpolation, while Best is the slowest.

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Set Canvas Color [SCv]

The Set Canvas Color node

Set Canvas Color Node Introduction

Set Canvas Color is used to set the color of the area outside the domain of definition (DoD). This is the workspace area beyond the raster by default, which is invisible since outside the raster is not rendered. However, the DoD can be within the raster as well. This can occur when compositing images smaller than the raster, or with transforms. By default, the canvas color used is black/no Alpha (transparent). However, since some nodes may change an image's canvas color—for example, inverting a mask changes the mask's canvas from black to white—the Set Canvas Color allows you to control the color of the canvas to whatever you require.

The Set Canvas Color node sets the color of the workspace outside the domain of definition (DOD). For example, if you create a circular gradient, the DoD is a square around the circular gradient in the viewer. Everything outside the DoD is understood to be black and therefore does not have to be rendered. To change the area outside the DoD, attach the Set Canvas Color node after the background and change the color.

> **NOTE:** Position the mouse pointer in a black area outside the raster to view the RGB canvas color in the status bar at the bottom left of the Fusion window.

Inputs

The Set Canvas Color node includes two inputs: one for the main image and a second for a foreground.

— **Input:** This orange input is the only required connection. It accepts a 2D image that reveals the canvas color if the image's DoD is smaller than the raster.

— **Foreground:** The optional green foreground input allows the canvas color to be sampled from an image connected to this input.

Basic Node Setup

The Set Canvas Color node is placed after the image is transformed to reveal part of the raster outside the domain of definition.

A Set Canvas Color node applied to a transformed MediaIn 1 node

The Set Canvas Color node is often used for adjusting keys. In the example above, the Luma Keyer is extracting a key, and therefore assigns the area outside the DoD, which is black, as an opaque foreground. If the element is scaled down and composited, you do not see the background. To correct this, insert a SetBGColor before the keyed element is placed in the composite. For example, LumaKey > Set Canvas Color > Transform > Merge.

Inspector
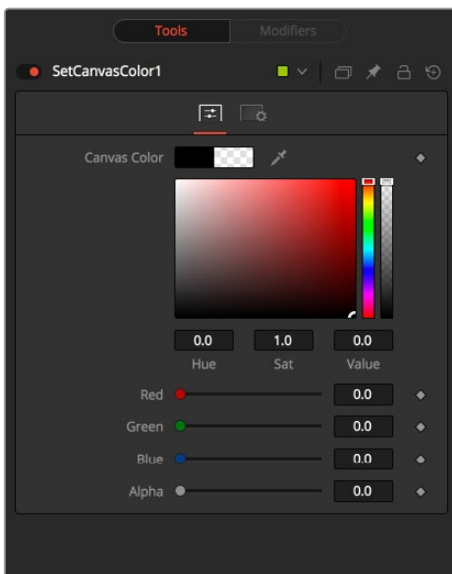
Set Canvas Color controls

Controls Tab

The Controls tab for the Set Canvas Color is used for simple color selection. When the green foreground is connected, the tab is empty.

Color Picker

Use these controls to adjust the Color and the Alpha value for the image's canvas. It defaults to black with zero Alpha.

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

White Balance [WB]

The White Balance node

White Balance Node Introduction

The White Balance node can be used to automatically remove color casts in the image caused by the incorrect setup of a camera or bad lighting conditions.

Correction can be done by selecting a color temperature or by choosing a neutral color from the original image that exhibits the color cast to be corrected.

> IMPORTANT When picking neutral colors using the Custom method, make sure you are picking from the source image, not the results of the White Balance node. This ensures that the image doesn't change while you are still picking, and that the White Balance node gets an accurate idea of the original colors it needs to correct.

Inputs

The White Balance node includes two inputs: one for the main image and the other for an effect mask to limit the area where the white balance is applied.

— **Input:** This orange input is the only required connection. It connects a 2D image for the white balance.
— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the white balance to only those pixels within the mask. An effect mask is applied to the tool after it is processed.

Basic Node Setup

The White Balance node, like many 2D image-processing nodes, receives a 2D image like the MediaIn1 shown below. The output continues the node tree by connecting to another 2D image-processing node or a Merge node.

A White Balance node applied to a MediaIn1 node

Inspector
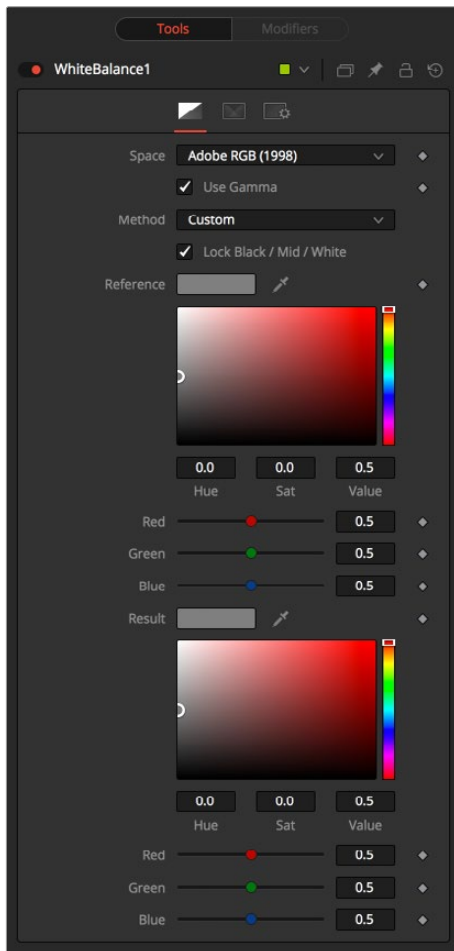
White Balance controls

Balance Tab

Space

Use this menu to select the color space of the source image, if it is known. This can make the correction more accurate since the node can take the natural gamma of the color space into account as part of the correction. If the color space that the image uses is unknown, leave this menu at its default value.

Method

The White Balance node can operate using one of two methods: a Custom method or a color Temperature method.

— **Custom:** The Custom method requires the selection of a pixel from the scene that should have been pure gray. The node uses this information to calculate the color correction required to convert the pixel so that it actually is gray. When the correction is applied without an effect mask connected and the LockBlack/Mid/White checkbox enabled, the node white balances the entire shot.

— **Temperature:** The color Temperature method requires that the actual color temperature of the shot be specified.

Lock Black/Mid/White

This checkbox locks the Black, Midtones, and White points together so that the entire image is affected equally. Unchecking the control provides individual controls for white balancing each range separately. This control affects both methods equally.

Black/Mid/White Reference

These controls appear only if the Custom method is selected. They are used to select a color from a pixel in the source image. The White Balance node color corrects the image so that the selected color is transformed to the color set in the Result Color Picker below. Generally, this is gray. A color that is supposed to be pure gray but is not truly gray for one reason or another should be selected.

If the Lock Black/Mid/White checkbox is deselected, different references can be selected for each color range.

For example, try to select a pixel for the black and white references that are not clipped in any of the color channels. In the high end, an example would be a pixel that is light pink with values of 255, 240, 240. The pixel is saturated/clipped in the red, although the color is not white. Similarly, a really dark blue-gray pixel might be 0, 2, 10. It is clipped in red as well, although it is not black.

Neither example would be a good choice as a reference pixel because there would not be enough headroom left for the White Balance node.

Black/Mid/White Result

These controls appear only if the Custom method is selected. They are used to select the color that the node uses to balance the reference color. This generally defaults to pure, midrange gray.

If the Lock Black/Mid/White checkbox is deselected, different results can be selected for each color range.

White Balance Temperature controls

Temperature Reference

When the Method menu is set to Temperature, the Temperature reference control is used to set the color temperature of the source image. If the Lock Black/ Mid/White checkbox is deselected, different references can be selected for each color range.

Temperature Result
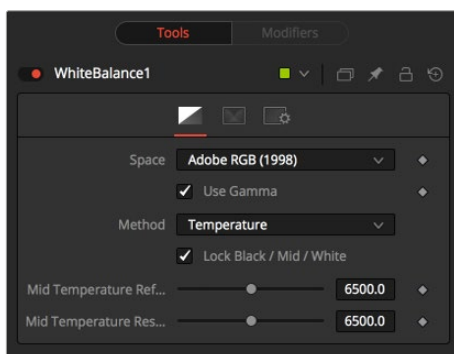
Use this control to set the target color temperature for the image. If the Lock Black/Mid/White checkbox is deselected, different results can be selected for each color range.

Use Gamma

This checkbox selects whether the node takes the gamma of the image into account when applying the correction, using the default gamma of the color space selected in the Space menu at the top of the tab.
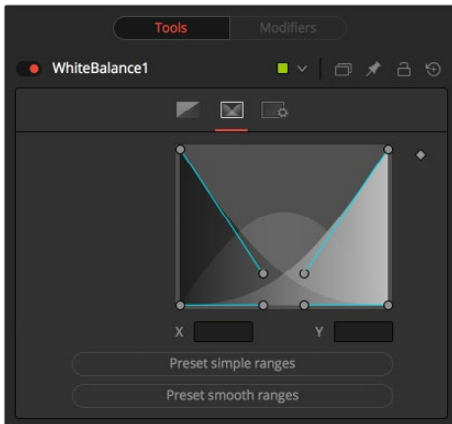
White Balance range tab

Ranges Tab

The Ranges tab can be used to customize the range of pixels in the image considered to be shadows, midtones, and highlights by the node.

Spline Display

The ranges are selected by manipulating the spline handles. There are four spline points, each with one Bézier handle. The two handles at the top represent the start of the shadow and highlight ranges, whereas the two at the bottom represent the end of the range. The Bézier handles are used to control the falloff.

The midtones range has no specific controls since its range is understood to be the space between the shadow and the highlight ranges.

The X and Y text controls below the Spline display can be used to enter precise positions for the selected Bézier point or handle.

Preset Simple/Smooth Ranges

These two buttons can be used to return the spline ranges to either Smooth (default) or Simple (linear) settings.

Settings Tab

The Settings tab in the Inspector is also duplicated in other Color nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

The Common Controls

Nodes that handle Color adjustment operations share several identical controls in the Inspector. This section describes controls that are common among color nodes.

Inspector

Color Settings Inspector

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Color category. The Settings controls are even found on third-party color type plugin tools. The controls are consistent and work the same way for each tool, although some tools do include one or two individual options that are also covered here.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this causes the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if processing of this node is scripted to trigger another task, but the value of the node is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the red button on a blur tool is deselected, the blur is first applied to the image, and then the red channel from the original input is copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option causes the RGB values of the masked image to be multiplied by the mask channel's values. This causes all pixels not included in the mask (i.e., set to 0) to become black/ transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels are used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID is used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the viewer. The image or sequence must have been rendered from a 3D software package with those channels included.

Clipping Mode

This option determines how edges are handled when performing domain of definition rendering. This is mostly important for nodes like Blur, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that would normally be outside the upstream DoD is treated as black/transparent.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off hardware-accelerated rendering using the graphics card in your computer. Enabled uses the hardware. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available.

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 causes Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one whole frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Comments

The Comments field is used to add notes to a tool. Click in the field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Composite Nodes

This chapter details the Dissolve and Merge nodes available in Fusion.

> The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.
>
> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Dissolve [Dx]
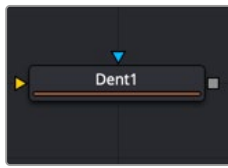
The Dissolve node

Dissolve Node Introduction

The Dissolve node is typically used to mix two images together, providing a gradual transition between two clips. A Background/Foreground slider controls the mix between the foreground and background images. Dissolves are commonly used to transition between one clip and another and are a very common effect in editing. However, you can also use the extreme left and right positions of the Background/Foreground slider to switch between inputs. Unlike other nodes in Fusion, the Dissolve node does not require you to connect an image to the background but lets you output either the background or foreground according to the setting of the Background/Foreground slider.

This quality makes it possible for you use the Dissolve node as an automatic layer switching tool when connected to background and foreground clips with different durations. Simply connect each clip to the background and foreground inputs, respectively, and set the Background/Foreground slider to the input of shorter duration, to determine which is "on top." After the last frame of that clip has ended, the Dissolve node automatically switches to the clip that's connected to the other input.

Besides the default dissolve, the Gradient Wipe setting of the Operation menu allows you to create arbitrary animated dissolve patterns based on the luminance of an image connected to the optional Gradient Wipe input. You can use this capability with images of geometric shapes or gradients of different kinds, movie clips of fire, water ripples, or rain, the Fast Noise node, or even particle systems you create within the Fusion page to create a variety of unique and creative transitions. Soft-edged effect masks may also be used to add to the possible effects.

Ultimately, animating the Background/Foreground control allows you to control the transition that's being used to switch from the foreground input to the background, or vice versa.

Inputs

The Dissolve node provides three image inputs, all of which are optional:

— **Background:** The first of two images you want to switch between or mix. Unlike most other nodes, it is unnecessary to connect the background input before connecting the foreground input.

— **Foreground:** The second of two images you want to switch between or mix. The Dissolve node works best when both foreground and background inputs are connected to images with the same resolution.

— **Gradient Map:** (Optional) The Gradient Map is required only when Gradient Wipe is selected.

Basic Node Setup

Dissolve nodes are typically connected in the following way, with two input images connected to the background and foreground inputs, and the output connected to the next node in the composition.

A typical dissolve node structure in Fusion

Resolution Handling

It is recommended to make sure that all images connected to the foreground, background, and gradient map inputs of the Dissolve node have the same resolution and the same pixel aspect. This is not required, however. But, the result if you mix resolutions depends on how you set the Background/ Foreground slider.

— If the input images are different sizes, but the Foreground/Background slider is set to full Foreground (all the way to the right) or full Background (all the way to the left), then the output resolution will be identical to the image resolution of the corresponding node input.

— If input images of different sizes are mixed by setting the Background/Foreground slider somewhere between, the output resolution will be set to the larger of the two input resolutions to make sure there's enough room to contain both images. In this case, you may experience undesirable resolution changes when the slider moves from full foreground or background to somewhere in between.

For example, if you try to dissolve between a 4K image (connected to the background) and an 8K image (connected to the foreground), the output of the Dissolve node will be 4K when the slider is set to full Background, but will suddenly jump to 8K when set to full Foreground or when mixed somewhere between the foreground and background.

Inspector
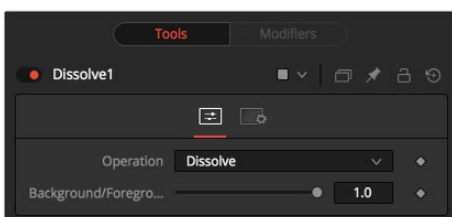
Dissolve controls

Controls Tab

These are the main controls that govern the Dissolve node's behavior.

— **Operation Pop-Up:** The Operation menu contains one of seven different methods for mixing the Foreground and Background inputs. The two images are mixed using the value of the Background/Foreground slider to determine the percentage each image contributes.

- **Dissolve:** The standard Dissolve mode is the equivalent of a cross dissolve: one clip fades out as another clip fades in.
- **Additive Dissolve:** Similar in look to a standard film dissolve, an Additive dissolve adds the second clip and then fades out the first one.
- **Erode:** The Erode method transitions between the two images by growing the darkest areas of the background image to reveal the foreground image. The effect appears similar to a filmstrip burning out.
- **Random Dissolve:** A randomly generated dot pattern is used to perform the mix of the images.
- **Random Noise Dissolve:** A moving random dot pattern is used to perform the mix of the images.
- **Gradient Wipe:** The dissolve is controlled by the luminance values of the image in the Gradient Map input. The edges of this dissolve can be softened. The density and the color of the border can be adjusted independently.
- **SMPTE Wipe:** The SMPTE wipe is similar to the basic effect wipes found on many video effects switchers. There is a horizontal wipe and a vertical wipe provided. The wipes can have soft edges and borders added. The density and the color of the border can be adjusted independently.
- **Background/Foreground Slider:** Defaults to Foreground. This control determines whether the output is the background image, the foreground image, or a mix between the two. The type of mix is determined by the Operation control. If one of the input images is not currently available, the other one will be output despite the setting of this slider.

Gradient/SMPTE Wipe Controls

The following controls appear only when Gradient Wipe or SMPTE Wipe are selected.

- **Wipe Style:** (SMPTE Wipe only) This drop-down list allows the selection of two wipe styles: Horizontal - Left to Right and Vertical - Top to Bottom. The direction of the wipes can be reversed by using the Invert Wipe checkbox.
- **Invert Wipe:** (SMPTE Wipe only) When checked, the direction of the wipe will be reversed.
- **Softness:** Use this control to soften the edge of the transition.
- **Border:** Select the Border to enable coloring of the transition's edge and to reveal the associated controls. The effect is to create a border around the transition edge.
- **Border Softness:** (Appears only when Border is turned on) The Border Softness slider controls the width and density of the border. Higher values will create a denser border, and lower values will create a thinner one.
- **Border Color:** (Appears only when Border is turned on) Use Border Color to select the color used in the border.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in both the Dissolve and Merge nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Merge [Mrg]
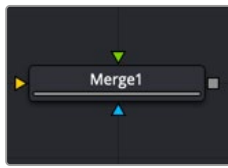
The Merge node

Merge Node Introduction

The Merge node combines two images based on the Alpha (opacity) channel associated with the one in front. This node takes two inputs: a background and a foreground image. The Operation mode determines which method is used to combine the foreground and background images, supporting the standard Over, In, Held Out, Atop, and XOr methods for compositing images. Meanwhile, an Apply Mode pop-up lets you use different composite modes, transfer modes, or blend modes (whichever is your preferred terminology) to combine the foreground against the background in different ways. This includes such standard modes as screen, dissolve, multiply, overlay, as well as many others.

The Merge node can perform both additive (premultiplied) and subtractive (non-premultiplied) compositing, depending on how your compositions and media are set up. However, you also have the flexibility of using the Additive/Subtractive slider to blend between additive and subtractive composite results, which has the bonus of providing solutions for problem edges in some cases.

Ordinarily, the foreground and background input connections determine the layer order of images composited with this node. However, you can also enable Z-Depth compositing if Z-channels are available in the input images. Z-merging compares the depth value of each pixel in each layer to determine which pixels should be in front and which should be behind.

Inputs

The Merge node provides three image inputs, all of which are optional:

— **Background:** The orange background input is for the first of two images you want to composite together. You should connect the background input before connecting the foreground input. If you connect an image to the background without connecting anything to the foreground input, the Merge node will output the background image.

— **Foreground:** The green foreground input is for the second of two images you want to composite together, which is typically a foreground subject that should be in front of the background. If you connect an image to the foreground input without connecting anything to the background input first, the Merge node won't output anything.

— **Effect Mask:** (Optional) The effect mask input lets you mask a limited area of the output image to be merged where the mask is white (where the foreground image shows in front of the background), letting the background image show through by itself where the mask is black.

Basic Node Setup

Merge nodes are typically connected in the following way, with two input images connected to the background and foreground inputs, and the output connected to the next node in the composition. In this example, the effect mask input is not used, as this is not typical.

A typical Merge node structure in DaVinci Resolve

Resolution Handling

While you can connect images of any resolution to the background and foreground inputs of the Merge node, the image that's connected to the background input determines the resolution of the output.

> **TIP:** If you want to change the resolution of the image connected to the background, you can use the Crop node to change the "canvas" resolution of the image without changing the size of the original image, or you can use the Resize node to change both the resolution and the size of the image.

Inspector
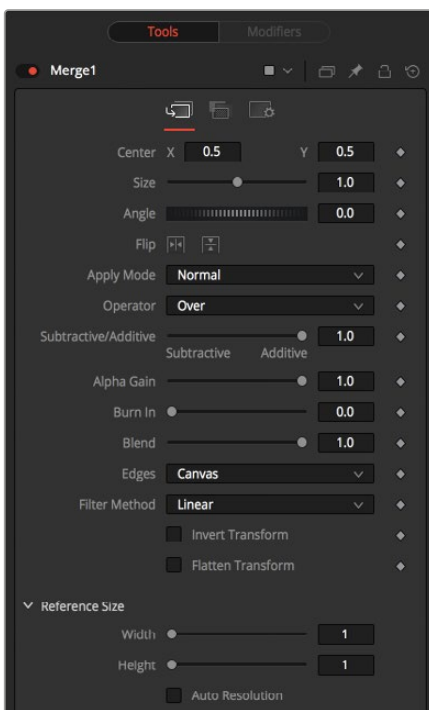
Merge node controls

Merge Tab

The Merge tab contains most of the controls necessary for customizing most merge operations.

Foreground Sizing Controls

These controls let you adjust the sizing of the image connected to the foreground input, making it unnecessary to use a separate Transform node to fit the foreground layer to match the background layer in simple compositions.

— **Center X and Y:** This control determines the position of the foreground image in the composite. The default is 0.5, 0.5, which centers the foreground image in the exact center of the background image. The value shown is always the actual position in normalized coordinates, multiplied by the reference size. See below for a description of the reference size controls.

— **Size:** Use this control to increase or decrease the size of the foreground image before it is composited over the background. The range of values for this slider is 0.0 to 5.0, but any value greater than 0 can be entered manually. A size of 1.0 gives a pixel-for-pixel composition, where a single pixel in the foreground is the same size as a single pixel in the background.

— **Angle:** Use this control to rotate the foreground image before it is combined with the background.

Compositing Mode and Adjustment Controls

The next six parameters control how the background and foreground input images are combined to create a single output image.

— **Apply Modes:** The Apply Mode setting determines the math used when blending or combining the foreground and background pixels.

  — **Normal:** The default Normal merge mode uses the foreground's Alpha channel as a mask to determine which pixels are transparent and which are not. When this is active, another menu shows possible operations, including Over, In, Held Out, Atop, and XOr.

  — **Screen:** Screen merges the images based on a multiplication of their color values. The Alpha channel is ignored, and layer order becomes irrelevant. The resulting color is always lighter. Screening with black leaves the color unchanged, whereas screening with white will always produce white. This effect creates a similar look to projecting several film frames onto the same surface. When this is active, another menu shows possible operations, including Over, In, Held Out, Atop, and XOr.

  — **Dissolve:** Dissolve mixes two image sequences together. It uses a calculated average of the two images to perform the mixture.

  — **Darken:** Darken looks at the color information in each channel and selects the background or foreground image's color value, whichever is darker, as the result color. Pixels lighter than the merged colors are replaced, and pixels darker than the merged color do not change.

  — **Multiply:** Multiplies the values of a color channel. This will give the appearance of darkening the image as the values are scaled from 0 to 1. White has a value of 1, so the result would be the same. Gray has a value of 0.5, so the result would be a darker image, or an image half as bright.

  — **Color Burn:** Color Burn uses the foreground's color values to darken the background image. This is similar to the photographic dark room technique of burning by increasing the exposure of an area of a print.

— **Linear Burn:** Linear Burn decreases the brightness of the base color based on the value of the blend color. The result is darker than Multiply but less saturated than Color Burn. Linear Burn also produces the most contrast in darker colors than any of the other blending modes in the Darker group.

— **Darker Color:** The Darker Color blending mode is very similar to Darken. This blending mode does not blend pixels. It only compares the base and blend colors, and it keeps the darkest of the two. The difference is that Darker Color looks at the composite of all the RGB channels, whereas Darken looks at each RGB channel individually to come up with a final blend.

— **Lighten:** Lighten looks at the color information in each channel and selects the background or foreground image's color values, whichever is lighter, as the result color value. Pixels darker than the merged color are replaced, and pixels lighter than the merged color do not change.

— **Color Dodge:** Color Dodge uses the foreground's color values to brighten the background image. This is similar to the photographic dark room technique of dodging by reducing the exposure of an area of a print.

— **Linear Dodge:** Linear Dodge (Add) produces similar but stronger results than Screen or Color Dodge. This blending mode looks at the color information in each channel and brightens the base color to reflect the blend color by increasing the brightness. Blending with black produces no change. Linear Dodge (Add) blends differently when Fill Opacity is adjusted, compared to when Opacity is adjusted.

— **Lighter Color:** Lighter Color is very similar to Lighten. This blending mode does not blend pixels. It only compares the base and blend colors, and it keeps the brightest of the two. The difference is that Lighter Color looks at the composite of all the RGB channels, whereas Lighten looks at each RGB channel to come up with a final blend.

— **Overlay:** Overlay multiplies or screens the color values of the foreground image, depending on the color values of the background image. Patterns or colors overlay the existing pixels while preserving the highlights and shadows of the color values of the background image. The background image is not replaced but is mixed with the foreground image to reflect the original lightness or darkness of the background image.

— **Soft Light:** Soft Light darkens or lightens the foreground image, depending on the color values of the background image. The effect is similar to shining a diffused spotlight on the image.

— **Hard Light:** Hard Light multiplies or screens the color values of the foreground image, depending on the color values of the background image. The effect is similar to shining a harsh spotlight on the image.

— **Vivid Light:** Vivid Light is an extreme version of Overlay and Soft Light. Anything darker than 50% gray is darkened, and anything lighter than 50% gray is lightened. Vivid Light is one of those blending modes where you may want to adjust the opacity, since 100% opacity is generally too strong. Vivid Light is the fifth blending mode of eight that give you different results when you reduce the fill compared to opacity.

— **Linear Light:** Linear Light uses a combination of the Linear Dodge blending on lighter pixels and a Linear Burn on darker pixels. Typically, the resulting colors are extreme, and you may want to use the Opacity or Fill sliders to adjust it. Linear Lights blends differently when Fill Opacity is adjusted, compared to when Opacity is adjusted.

— **Pin Light:** Pin Light is an extreme blending mode that performs a Darken and Lighten blending mode simultaneously. It can result in patches or blotches, and it completely removes all mid-tones.

— **Difference:** Difference looks at the color information in each channel and subtracts the foreground color values from the background color values or the background from the foreground, depending on which has the greater brightness value. Merging with white inverts the color. Merging with black produces no change.

— **Exclusion:** Exclusion creates an effect similar to, but lower in contrast than, the Difference mode. Merging with white inverts the base color values. Merging with black produces no change.

— **Hue:** Hue creates a result color with the luminance and saturation of the background color values and the hue of the foreground color values.

— **Saturation:** Saturation creates a result color with the luminance and hue of the base color and the saturation of the blend color.

— **Color:** Color creates a result color with the luminance of the background color value and the hue and saturation of the foreground. This preserves the gray levels in the image and is useful for coloring monochrome images.

— **Luminosity:** Luminosity creates a result color with the hue and saturation of the background color values and the luminance of the foreground color values. This mode creates an inverse effect from that of the Color mode.

— **Hypotenuse:** This blend mode is good for HDR images that have out of range colors above 1, It uses a square root of each color squared and added to blend the color.

  `Out = sqrt (Fc*Fc +Bc*Bc)`

— **Geometric:** This blend mode is good for HDR images that have out of range colors above 1, For values above zero the result is 2 times the foreground times the background color divided by the foreground plus background color.

  `Out = 2*Fc*Bc / (Fc+Bc)`

— **Operator Modes:** This menu is used to select the Operation mode of the merge. Changing the Operation mode changes how the foreground and background are combined to produce a result. This pop-up menu is visible only when the Merge node's Apply mode is set to either Normal or Screen.

For an excellent description of the math underlying the Operation modes, read Compositing Digital Images, Porter, Thomas, and T. Duff, ACM SIGGRAPH Computer Graphics proceedings, 1984, pages 253-259. Essentially, the math is as described below. Note that some modes not listed in the Operator drop-down menu (Under, In, Held In, Below) are easily obtained by swapping the foreground and background inputs (with Command-T or Ctrl-T) and choosing a corresponding mode. The formula used to combine pixels in the merge is always (fg * x) + (bg * y). The different operations determine exactly what x and y are, as shown in the description for each mode.

The Operator modes are as follows:

— **Over:** The Over mode adds the foreground layer to the background layer by replacing the pixels in the background with the pixels from the Z wherever the foreground's Alpha channel is greater than 1.

  `x = 1, y = 1-[foreground Alpha]`

— **In:** The In mode multiplies the Alpha channel of the background input against the pixels in the foreground. The color channels of the foreground input are ignored. Only pixels from the foreground are seen in the final output. This essentially clips the foreground using the mask from the background.

  `x = [background Alpha], y = 0`

— **Held Out:** Held Out is essentially the opposite of the In operation. The pixels in the foreground image are multiplied against the inverted Alpha channel of the background image. Accomplish exactly the same result using the In operation and a Matte Control node to invert the matte channel of the background image.

`x = 1–[background Alpha], y = 0`

— **Atop:** Atop places the foreground over the background only where the background has a matte.

`x = [background Alpha], y = 1–[foreground Alpha]`

— **XOr:** XOr combines the foreground with the background wherever either the foreground or the background has a matte, but never where both have a matte.

`x = 1–[background Alpha], y = 1–[foreground Alpha]`

— **Conjoint:** The Conjoint mode will make a decision based combination of Alpha channels of the foreground and background images; this is helpful in soft edge and motion blurred Alpha where Alpha is not solid.

`X= 1, Y= X+Y(1–af)/ab, if af>ab`

— **Disjoint:** The Disjoint mode will make a decision based combination of Alpha channels of the foreground and background images; this is helpful in combining layers as to not get out of range Alpha, and premultiplied edges get the correct Alpha combination.

`X= X+Y(1–af)/ab, Y= X+Y if af+ab<1`

— **Mask:** The Mask mode will output the background image multiplied by the foreground Alpha.

`X = X * af, Y =0`

— **Stencil:** The Stencil mode will output the background image multiplied by the inverse foreground Alpha.

`X = X * (1–af), Y =0`

— **Under:** The Under mode is the same operation as the Over mode but will swap foreground and background images in the operations.

`X = Y, Y =X *(1–af)`

— **Subtractive/Additive slider:** This slider controls whether Fusion performs an Additive merge, a Subtractive merge, or a blend of both. This slider defaults to Additive merging for most operations, assuming the input images are premultiplied (which is usually the case). If you don't understand the difference between Additive and Subtractive merging, here's a quick explanation.

— An Additive merge is necessary when the foreground image is premultiplied, meaning that the pixels in the color channels have been multiplied by the pixels in the Alpha channel. The result is that transparent pixels are always black, since any number multiplied by 0 always equals 0. This obscures the background (by multiplying with the inverse of the foreground Alpha), and then simply adds the pixels from the foreground.

— A Subtractive merge is necessary if the foreground image is not pre-multiplied. The compositing method is similar to an additive merge, but the foreground image is first multiplied by its own Alpha to eliminate any background pixels outside the Alpha area.

While the Additive/Subtractive option could easily have been a checkbox to select one mode or another, the Merge node lets you blend between the Additive and Subtractive versions of the merge operation—an operation that is occasionally useful for dealing with problem composites with edges that are calling attention to themselves as too bright or too dark.

For example, using Subtractive merging on a premultiplied image may lead to darker edges, whereas using Additive merging with a non-premultiplied image will cause any non-black area outside the foreground's Alpha to be added to the result, thereby lightening the edges. By blending between Additive and Subtractive, you can tweak the edge brightness to be just right for your situation.
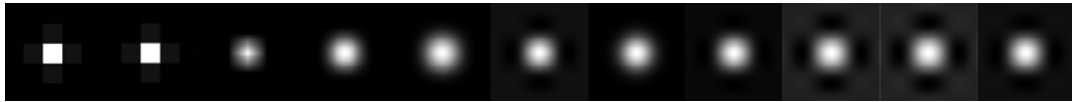
— **Alpha Gain slider:** Alpha Gain linearly scales the values of the foreground's Alpha channel. In Subtractive merges, this controls the density of the composite, similarly to Blend. In Additive merges, this effectively reduces the amount that the background is obscured, thus brightening the overall result. In an Additive merge with Alpha Gain set to 0.0, the foreground pixels are simply added to the background.

— **Burn In slider:** The Burn In control adjusts the amount of Alpha used to darken the background, without affecting the amount of foreground added in. At 0.0, the merge behaves like a straight Alpha blend, whereas at 1.0, the foreground is effectively added onto the background (after Alpha multiplication if in Subtractive mode). This gives the effect of the foreground image brightening the background image, as with Alpha Gain. For Additive merges, increasing the Burn In gives an identical result to decreasing Alpha Gain.

— **Blend slider:** This is a cloned instance of the Blend slider in the Common Controls tab. Changes made to this control are simultaneously made to the one in the common controls. The Blend slider mixes the result of the node with its input, blending back the effect at any value less than 1.0. In this case, it will blend the background with the merged result.

Additional Controls

The remaining controls let you fine-tune the results of the above settings.

— **Filter Method:** For input images that are being resized, this setting lets you choose the filter method used to interpolate image pixels when resizing clips. The default setting is Linear. Different settings work better for different kinds of resizing. Most of these filters are useful only when making an image larger. When shrinking images, it is common to use the Linear filter; however, the Catmull-Rom filter will apply some sharpening to the results and may be useful for preserving detail when scaling down an image.

   — **Nearest Neighbor:** This skips or duplicates pixels as needed. This produces the fastest but crudest results.

   — **Box:** This is a simple interpolation resize of the image.

   — **Linear:** This uses a simplistic filter, which produces relatively clean and fast results.

   — **Quadratic:** This filter produces a nominal result. It offers a good compromise between speed and quality.

   — **Cubic:** This produces better results with continuous-tone images. If the images have fine detail in them, the results may be blurrier than desired.

   — **Catmull-Rom:** This produces good results with continuous-tone images that are resized down. Produces sharp results with finely detailed images.

   — **Gaussian:** This is very similar in speed and quality to Bi-Cubic.

   — **Mitchell:** This is similar to Catmull-Rom but produces better results with finely detailed images. It is slower than Catmull-Rom.

   — **Lanczos:** This is very similar to Mitchell and Catmull-Rom but is a little cleaner and also slower.

   — **Sinc:** This is an advanced filter that produces very sharp, detailed results; however, it may produce visible "ringing" in some situations.

— **Bessel:** This is similar to the Sinc filter but may be slightly faster.

— **Window Method:** Some filters, such as Sinc and Bessel, require an infinite number of pixels to calculate exactly. To speed up this operation, a windowing function is used to approximate the filter and limit the number of pixels required. This control appears when a filter that requires windowing is selected.

— **Hanning:** This is a simple tapered window.

— **Hamming:** Hamming is a slightly tweaked version of Hanning.

— **Blackman:** A window with a more sharply tapered falloff.

— **Kaiser:** A more complex window, with results between Hamming and Blackman.

Resize Filters from left to right: Nearest Neighbor, Box, Linear, Quadratic, Cubic, Catmull-Rom, Gaussian, Mitchell, Lanczos, Sinc, and Bessel

— **Edges Buttons:** Four buttons let you choose how to handle the space around images that are smaller than the current DoD of the canvas as defined by the resolution of the background image.

— **Canvas:** The area outside the frame is set to the current color/opacity of the canvas. If you want to change this value, you can attach a Set Canvas Color node between the image connected to the foreground input and the foreground input itself, using Set Canvas Color to choose a color and/or transparency setting with which to fill the canvas.

— **Wrap:** Creates a "video wall" effect by duplicating the foreground image as a grid.

— **Duplicate:** Duplicates the outermost pixels along the edge of the foreground image, duplicating them to stretch up, down, left, and right from each side to reach the end of the DoD.

— **Mirror:** Similar to duplicate, except every other iteration of the foreground image is flipped and flopped to create a repeating pattern.

— **Invert Transform:** Select the Invert Transform control to invert any position, rotation, or scaling transformation. This option is useful when connecting the merge to the position of a tracker for match moving.

— **Flatten Transform:** The Flatten Transform option prevents this node from concatenating its transformation with subsequent nodes. The node may still concatenate transforms from its input, but it will not concatenate its transformation with the node at its output.

— **Reference Size:** The controls under Reference Size do not directly affect the image. Instead, they allow you to control how Fusion represents the position of the Merge node's center.

Normally, coordinates are represented as values between 0 and 1, where 1 is a distance equal to the full width or height of the image. This allows resolution independence, because the size of the image can be changed without having to change the value of the center.

One disadvantage to this approach is that it complicates making pixel-accurate adjustments to an image. To demonstrate, imagine an image that is 100 x 100 pixels in size. To move the center of the foreground element to the right by 5 pixels, we would change the X value of the merge center from 0.5, 0.5 to 0.55, 0.5. We know the change must be 0.05 because 5/100 = 0.05.

If you specify the dimensions of the background image in the Reference Size controls, this changes the way the center control values are displayed so that it shows the actual pixel positions in its X and Y fields.

Extending the example, set the width and height to 100 each and the center will now be shown as 50, 50, and we would move it 5 pixels toward the right by entering 55, 50.

Internally, the Merge node still stores this value as a number between 0 to 1 and, if the center control's value was to be queried via scripting or the center control was to be published for use by other nodes, the original normalized value would be retrieved. The change is only visible in the value shown for merge center in the node control.

— **Use Frame Format Settings:** Select this to force the merge to use the composition's current frame format settings to set the reference width and reference height values.

— **Width and Height:** Set these sliders to the width and height of the image to change the way that Fusion displays the values of the Merge node's center control.

Channels Tab
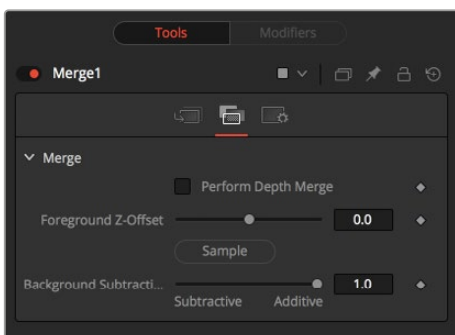
Merge node Channels tab

The Channels tab has controls that let the Merge node use Z-channels embedded within each image to define what's in front and what's behind during a Merge operation. The following controls let you customize the result.

— **Perform Depth Merge:** Off by default. When turned on, the Z-channel of both images will be used to determine the composite order. Alpha channels are still used to define transparency, but the values of the Z-Depth channels will determine the ordering of image elements, front to back. If a Z-channel is not available for either image, the setting of this checkbox will be ignored, and no depth compositing will take place. If Z-Depth channels are available, turning this checkbox off disables their use within this operation.

— **Foreground Z-Offset:** This slider sets an offset applied to the foreground image's Z value. Click the Pick button to pick a value from a displayed image's Z-channel, or enter a value using the slider or input boxes. Raising the value causes the foreground image's Z-channel to be offset further away along the Z-axis, whereas lowering the value causes the foreground to move closer.

— **Subtractive/Additive:** When Z-compositing, it is possible for image pixels from the background to be composited in the foreground of the output because the Z-buffer for that pixel is closer than the Z of the foreground pixel. This slider controls whether these pixels are merged in an Additive or a Subtractive mode, in exactly the same way as the comparable slider in the Merge tab.

When merged over a background of a different color, the original background will still be visible in the semitransparent areas. An Additive merge will maintain the transparencies of the image but will add their values to the background.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in both the Dissolve and Merge nodes. These common controls are described in the following "The Common Controls" section.
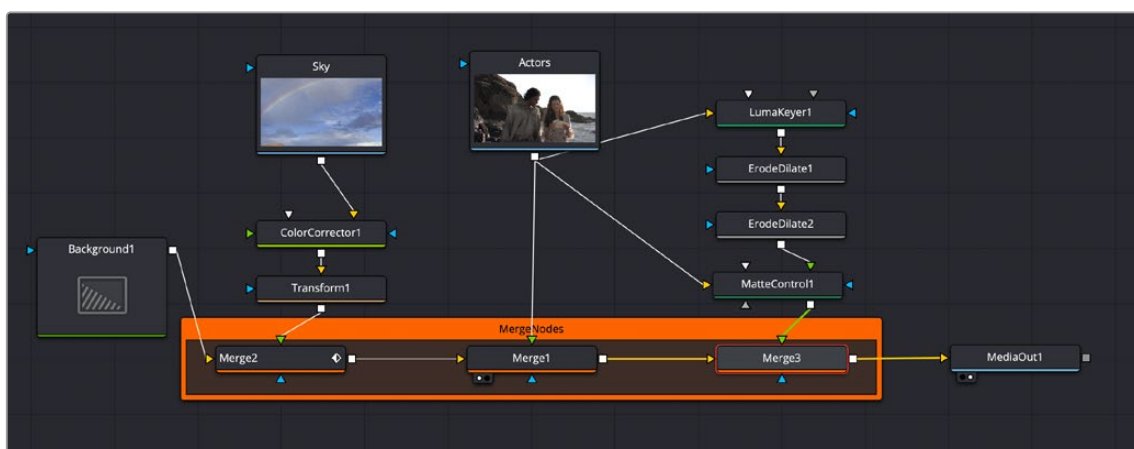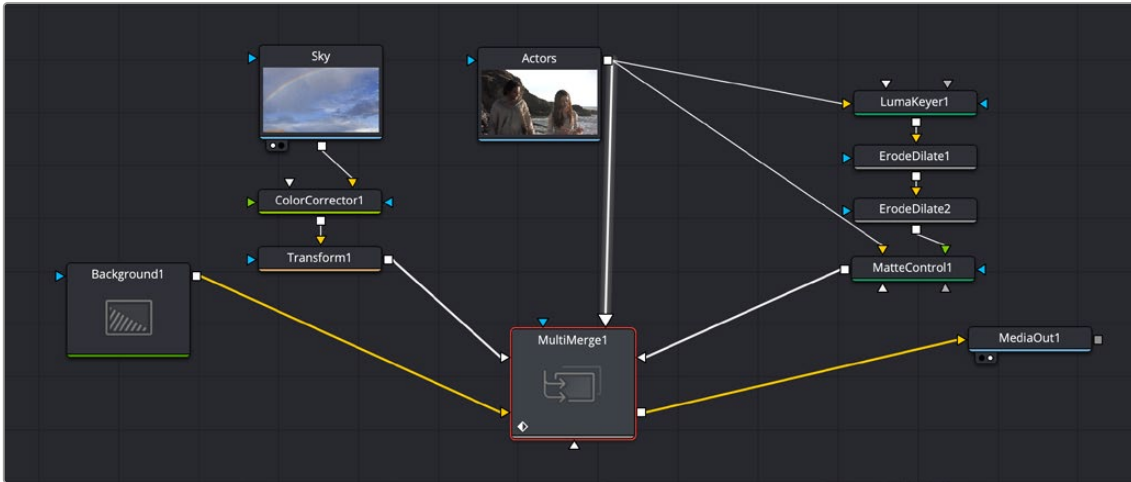
MultiMerge [MMrg]

The MultiMerge node

MultiMerge Node Introduction

The MultiMerge node allows you to combine several sources together in a layer-based structure. Unlike the standard Merge tool, which only accepts one foreground input, MultiMerge creates a new consecutive foreground input each time a new source is dragged to the tool.

These inputs are added to the Layer List in the Inspector and are displayed in a hierarchy from top to bottom as a stack. The layers are arranged so the layer closest to the foreground is at the top of the list, and the one closest to the background is at the bottom. Each layer has its own separate and independent Merge controls (see the Merge node above) that appear in the lower half of the Inspector, allowing you to individually adjust the position, size, composite modes, etc., for each source input.

As your composite increases in complexity, you can end up with a large amount of standard Merge nodes scattered throughout the node tree. MultiMerge can be used to combine these Merge nodes together, both as an organizational tool and allowing you to control the order of operations in a layer- based, rather than a node-based environment. You could conceivably composite and organize hundreds of separate layers using one MultiMerge node on your node tree.

A sky replacement composite requiring three separate Merge nodes (above) is replaced by one single Multi Merge node (bottom). The pipe of the currently selected layer in the Layer List has a slight glow applied (Actors node to MultiMerge node).
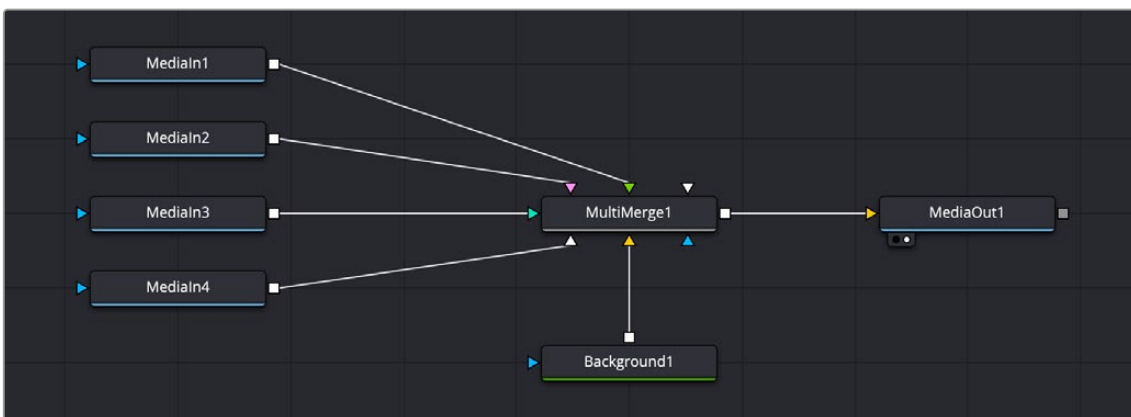
Inputs

The MultiMerge node provides these image inputs:

— **Background:** The orange background input is for the first of the images you want to composite together. You should connect the background input before connecting the foreground input.

— **Foreground:** The foreground inputs are for each subsequent image you want to composite together, which is typically a foreground subject that should be in front of the background. Connecting a pipe from a new source onto the MultiMerge node automatically creates a new foreground input on the node and a new layer in the Layer List. Foreground inputs are all colored white.

— **Effect Mask:** (Optional) The effect mask input lets you mask a limited area of the output image to be merged where the mask is white (where the foreground image shows in front of the background), letting the background image show through by itself where the mask is black.

Basic Node Setup

MultiMerge nodes are typically connected in the following way, with multiple input images connected to the foreground inputs, a background input, and the output connected to the next node in the composition. In this example, the effect mask input is not used, as this is not typical.
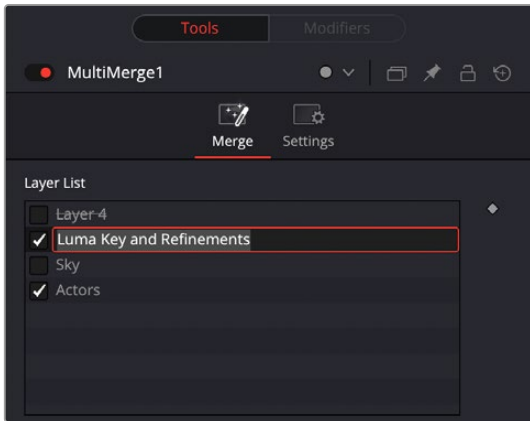

A typical MultiMerge node structure in DaVinci Resolve

Resolution Handling

While you can connect images of any resolution to the background and foreground inputs of the MultiMerge node, the image that's connected to the background input determines the resolution of the output.

Inspector

The Layer List of the MultiMerge node. The input to Layer 4 has been disconnected (strike through the name) and disabled (unchecked). The Luma Key layer is having its name changed, the Sky layer is disabled (unchecked), while the Actors layer is enabled (checked).

Layer List

For each foreground input connected to the MultiMerge node, a new layer is created sequentially in the Layer List. The Layer List is hierarchically sorted from top to bottom, with layers on top being above any layers below them in the image. You can customize this layer structure in several ways.

To Select a Layer:

Click on the layer in the Layer List to make that layer active. The active layer's Merge controls will appear at the bottom of the Inspector, and the pipe connecting to the layer's input on the MultiMerge node will glow slightly in the node tree.

You can multi-select layers by holding the Shift key down while clicking to select a range of layers, or the Command key to select multiple non-adjacent layers.

Right-clicking on any layer opens up a contextual menu with the following options:

— **Go To Source:** Selects the tool that is connected to the layer's input on the MultiMerge node and opens the tool's parameters in the Inspector.

— **Split Here:** Automatically creates another connected MultiMerge node and splits the existing layers between the two. The layer that was selected for the split, and all layers above it, will be spun off into a new MultiMerge node. The original MultiMerge node will be connected to the background input of the new MultiMerge node.

— **Rename Layer X:** Opens up a dialog box allowing you to rename the selected layer.

— **Rename Layer to Match Source:** Automatically changes the name of the layer to the name of the tool it's connected to.

— **Set Layer X to Default:** This resets the layer's merge settings below, back to the default.
— **Delete Layer:** This deletes the layer entirely from the Layer List and disconnects the pipe from the original tool. It does not delete the original tool.

To Move a Layer up or down the List:

Click and drag on the Layer name and move it up or down the list. A grey line will highlight where in the list the layer will now be placed. Let go of the mouse button to make the change.

To Rename a Layer:

By default each layer is named based in the input tool's name. To change the Layer's name to something more meaningful, double-click on the Layer's name and type a new name in the text field. You can also right-click on a Layer's name and select Rename Layer from the contextual menu.

To Disable/Enable a Layer:

Checking or unchecking the box next to the Layer's name will enable or disable that layer respectively. Disabling the layer turns off that input's contribution to the overall composite in the MultiMerge but does not delete it.

To Replace a Layer:

By design, if you disconnect the pipe from a the MultiMerge's input, the layer will still remain in its place in the Layer List but with a strike-through its name to let you know that nothing is currently connected. This allows you to quickly iterate and audition several input sources (clips, graphics, etc.) to the same layer without having to constantly rearrange the order of your Layer List.

Keyframing the Layer List:

The Layer List itself is keyframable, using the standard keyframe controls to the right of the list. Parameters that can be keyframed are layer order, and layer enable and disable.

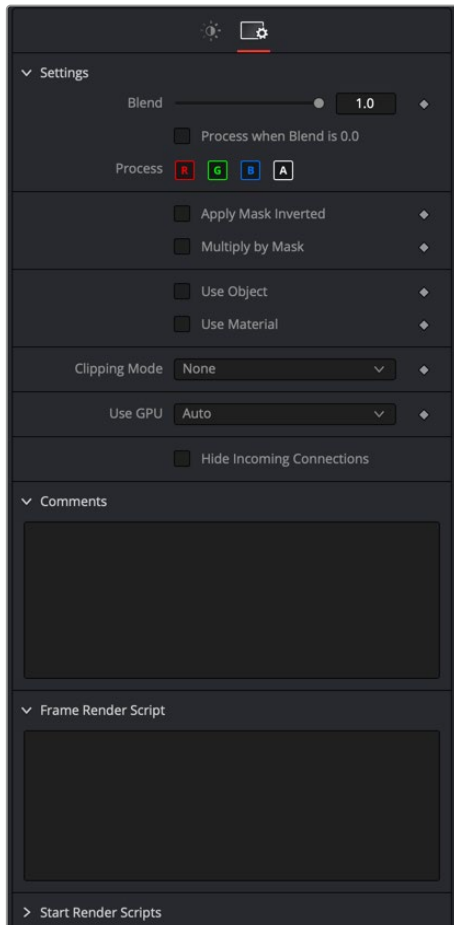## Merge Controls

Each layer has a separate and independent set of Merge controls that appear here when the layer is selected. For detailed information on how the Merge controls work, see the previous section in this chapter titled **Merge [Mrg]**.

The Common Controls

The Merge, MultiMerge, and Dissolve nodes share several identical controls in the Inspector. This section describes controls that are common among those two nodes.

Inspector

Color Setting Inspector

Settings Tab

The Settings tab in the Inspector can be found on both tools in the Composite category. The Settings controls are even found on third-party color type plugin tools. The controls are consistent and work the same way for each tool, although some tools do include one or two individual options that are also covered here.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this will cause the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if processing of this node is scripted to trigger another task, but the value of the node is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the red button on a blur tool is deselected, the blur will first be applied to the image, then the red channel from the original input will be copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this will generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option will cause the RGB values of the masked image to be multiplied by the mask channel's values. This will cause all pixels of the image not in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels will be used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID will be used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the view. The image or sequence must have been rendered from a 3D software package with those channels included.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off hardware-accelerated rendering using the graphics card in your computer. Enabled uses the hardware. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 will cause Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one whole frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Comments

The Comments field is used to add notes to a tool. Click in the field and type the text. When a note is added to a tool, a small red square appears in the lower left corner of the node when the full tile is displayed or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the node editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Deep Pixel Nodes

This chapter details Deep Pixel nodes found in Fusion. Deep Pixel nodes are capable of handling AOVs (Arbitrary Output Variables) from 3D-rendered files.

> The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.
>
> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Ambient Occlusion [SSAO]

The Ambient Occlusion node

Ambient Occlusion Node Introduction

Ambient Occlusion (AO) is the lighting caused when a scene is surrounded by a uniform diffuse spherical light source. Think of the scene as being surrounded by a humongous sphere that uniformly emits light from its surface. AO captures the low frequency lighting. It does not capture sharp shadows or Diffuse or Specular lighting. So, AO is usually combined with Diffuse and Specular lighting to create a full lighting solution.

The Ambient Occlusion node generates global lighting effects in 3D-rendered scenes as a post effect. It quickly approximates computationally expensive ray-traced global illumination. Being a post effect, it exposes similar aliasing issues like the Shader, Texture, and Volume Fog nodes. Hence, artifacts may appear in certain situations.

Usage

The AO node rarely works out of the box, and requires some tweaking. The setup process involves adjusting the Kernel Radius and Number Of Samples to get the desired affect.

The Kernel Radius depends on the natural "scale" of the scene. Initially, there might appear to be no AO at all. In most cases, the Kernel Radius is too small or too big, and working values must be found.

Inputs

There are three inputs on the AO node. The standard effect mask is used to limit the AO effect. The Input and Camera connections are required. If either of these is not supplied, the node does not render an image on output.

— **Input:** This orange input accepts a 2D RGBA image, Z-Depth, and Normals.
— **Camera:** The green camera input can take either a 3D Scene or a 3D Camera that rendered the 2D image.
— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Ambient Occlusion to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.
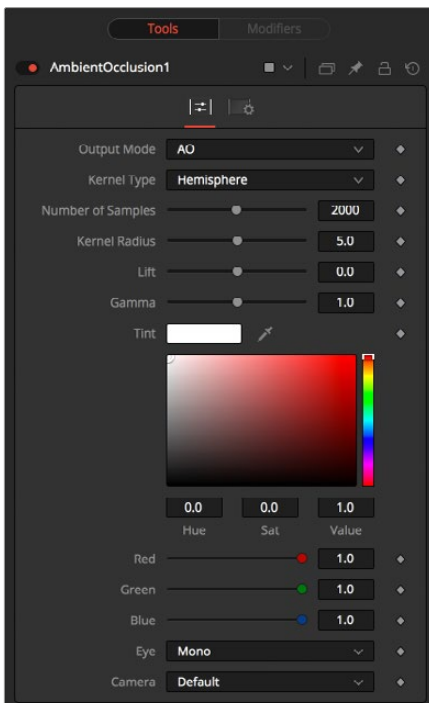
Basic Node Setup

The Ambient Occlusion node is typically placed after a Renderer 3D node. The Renderer 3D node must have Z-Depth and Normals enabled in its output channels. The Camera3D that is rendered by the Renderer3D node is then connected to the camera input on the AO node.
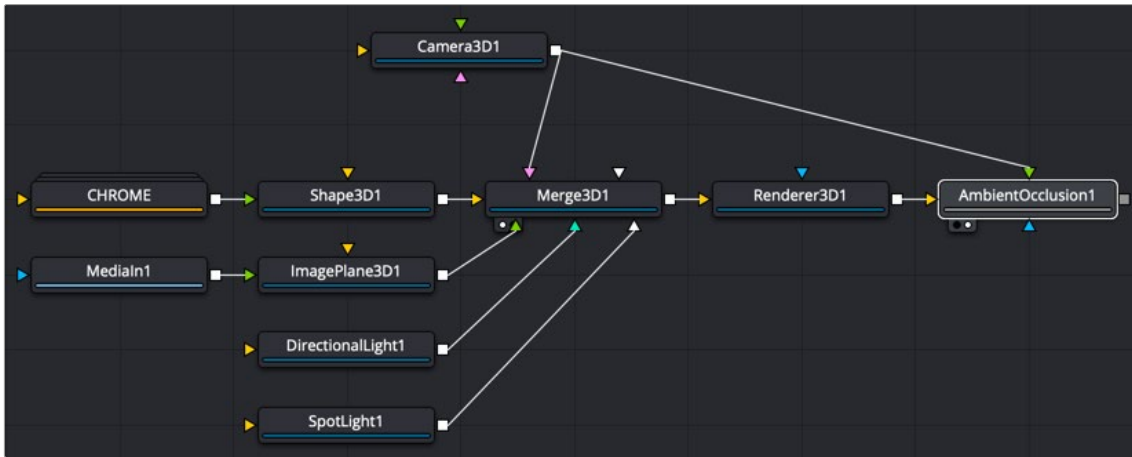
Ambient occlusion as a 2D post process

Inspector

Ambient Occlusion controls

Controls Tab

The controls tab includes all the main controls for compositing with AO. It controls the quality and appearance of the effect.

Output Mode

— **Color:** Using the Color menu option combines the incoming image with Ambient Occlusion applied.

— **AO:** This option outputs the pure Ambient Occlusion as a grayscale image. White corresponds to regions in the image that should be bright, while black correspond to regions that should be darker. This allows you to create a lighting equation by combining separate ambient/diffuse/specular passes. Having the AO as a separate buffer allows creative freedom to combine the passes in various ways.

Kernal Type

To determine the AO, rays are cast from a point on the surface being shaded, outward to a large enclosed sphere.

The AO factor is determined by the unoccluded rays that reach the sphere.

— **Hemisphere:** Rays are cast toward a hemisphere oriented to the surfaces normal. This option is more realistic than Sphere and should be used unless there is a good reason otherwise. Flat surfaces receive 100% ambient intensity, while other parts are darkened.

— **Sphere:** Rays are cast toward a sphere centered about the point being shaded. This option is provided to produce a stylistic effect. Flat surfaces receive 50% ambient intensity, while other parts are made darker or brighter.

Number of Samples

Increase the samples until artifacts in the AO pass disappear. Higher values can generate better results but also increase render time.

Kernel Radius

The Kernel Radius controls the size of the filter kernel in 3D space. For each pixel, it controls how far one searches in 3D space for occluders. The Filter Kernel should be adjusted manually for each individual scene.

If made too small, nearby occluders can be missed. If made too large, the quality of the AO decreases and the samples must be increased dramatically to get the quality back.

This value is dependent on the scene Z-depth. That means with huge Z values in the scene, the kernel size must be large as well. With tiny Z values, a small kernel size like 0.1 should be sufficient.

Lift/Gamma/Tint

You can use the lift, gamma, and tint controls to adjust the AO for artistic effects.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Deep Pixel nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

> **TIP:** Combining multiple AO passes with different kernel radii can produce better effects.

Kernel Radius 0.01   Kernel Radius 0.01   Kernel Radius 0.01   Kernel Radius 0.01
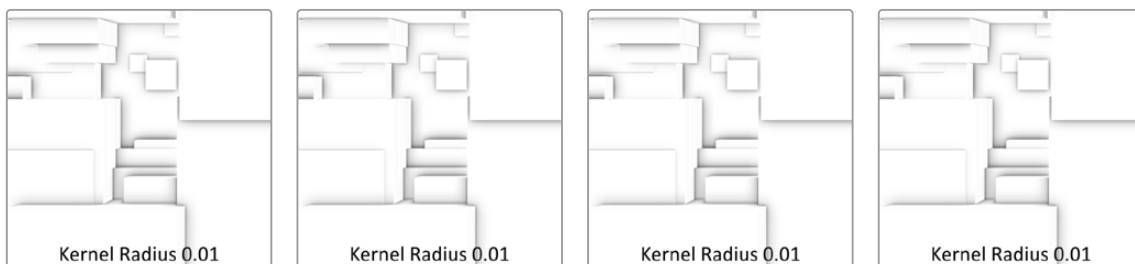
AO Tips and Limitations

Transparency/Translucency: AO is designed to work with opaque objects. There are known limitations with transparent receivers and those with transparent occluders. You can work around some of these limitations by splitting out the transparent/translucent objects into separate scenes and only computing AO on the opaque objects.

Particles: Because of the transparency/translucency limitations, do not use AO on particles, unless the particles are solid opaque geometry. Anti-aliased edges are another form of transparency, so they also cause problems with AO.

Supersampling: To render anti-aliasing with Ambient Occlusion, enable HiQ for the Z and Normals pass in the Renderer 3D.

Viewer Dependence: AO methods work in viewer space, and the results are viewer dependent. This means the amount of darkening can vary depending on the view location, when in reality it should be constant. If at a point on an object the AO is 0.5, moving the camera could change it to 0.4.

Baking of AO: The OpenGL UV renderer can be used to bake AO into the textures on models.

Depth Blur [DBl]

The Depth Blur node

Depth Blur Node Introduction

The Depth Blur node is primarily used to create focal length or depth-of-field effects. It blurs 3D-rendered images based on included Z-channel values, and can also be used for general per-pixel blurring effects through the Blur Channel controls.

Inputs

The Depth Blur node includes three inputs: one for the main image, one for a blur image, and another for an effect mask to limit the area where the depth blur is applied.

— **Input:** This orange input is the only required connection. It accepts a 2D image that includes a Z channel. The Z channel is used to determine the blur amount in different regions of the image.

— **Blur Image:** If the Blur Image input is connected, channels from the image are used to control the blur. This allows general 2D per-pixel blurring effects.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the depth blur to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Depth Blur node receives the image containing the Z channel. Below, the Z depth channel is provided in a separate image file and combined with the RGB (beauty) image using a Channels Booleans tool. Channel Booleans sets the Z buffer channel to copy into the luminance foreground.

Depth Blur is applied to the beauty image based on a Z depth channel

Inspector
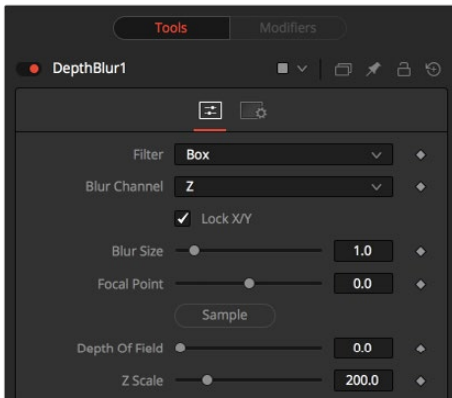
Depth Blur controls

Controls Tab

The Controls tab includes parameters for adjusting the amount of blur applied and the depth of the blurred area. It also includes options for selecting channels other than the Z channel for the blur map.

Filter

This menu selects the filter used for the blur.

— **Box:** This applies a basic depth-based box blur effect to the image.
— **Soften:** This applies a depth-based general softening filter effect.
— **Super Soften:** This applies a depth-based high-quality softening filter effect.

Blur Channel

Select one of these options to determine the channel used to control the level of blur applied to each pixel. The channel from the main image input is used, unless an image is connected to the node's green Blur Image input.

Lock X/Y

When toggled on, this control locks the X and Y Blur sliders together for symmetrical blurring.

Blur Size

This slider is used to set the strength of the horizontal and vertical blurring.

Focal Point

This control is visible only when the Blur channel menu is set to use the Z channel.

Use this control to select the distance of the simulated point of focus. Lowering the value causes the Focal Point to be closer to the camera; raising the value causes the Focal Point to be farther away.

Depth of Field

This control is used to determine the depth of the area in focus. The focal point is positioned in the middle of the region, and all pixels with a Z-value within the region stay in focus. For example, if the focal point were selected from the image and set to a value of 300, and the depth of field is set to 200, any pixel with a Z-value between 200 and 400 would remain in focus.

Z Scale

Scales the Z-buffer value by the selected amount. Raising the value causes the distances in the Z-channel to expand. Lowering the value causes them to contract. This is useful for exaggerating the depth effect. It can also be used to soften the boundaries of the blur. Some images with small depth values may require the Z-scale to be set quite low, below 1.0.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Deep Pixel nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Fog [Fog]

The Fog node

Fog Node Introduction

The Fog node is used to create simulated fog effects on 3D-rendered images that contain a valid Z-buffer channel. The fog can be placed in front of or behind various elements of a rendered image based on the selected Z-channel planes.

Inputs

The Fog node includes three inputs: one for the main image with a Z channel, one for a blur image, and another for an effect mask to limit the area where the depth blur is applied.

— **Input:** This orange input is the only required connection. It accepts a 2D image that includes a Z channel. The Z channel is used to determine the fog amount in different regions of the image.

— **Blur Image:** The green second image input connects an image that is used as the source of the fog. If no image is provided, the fog consists of a single color. Generally, a noise map of some sort is connected here.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the fog to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.
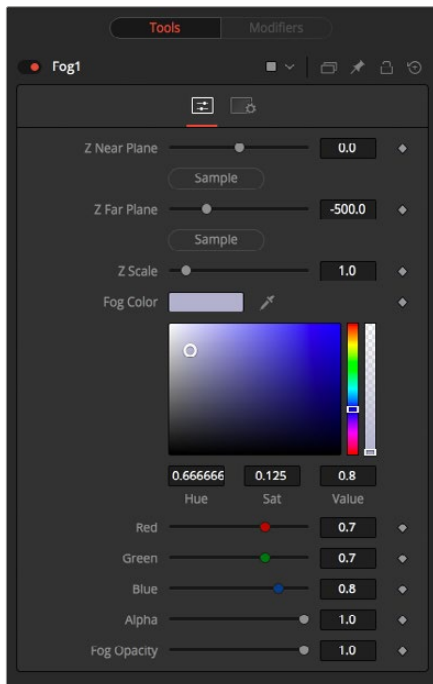
Basic Node Setup

The Fog node is typically placed after a Renderer 3D node. The Renderer 3D node must have Z-Depth enabled in its output channels.
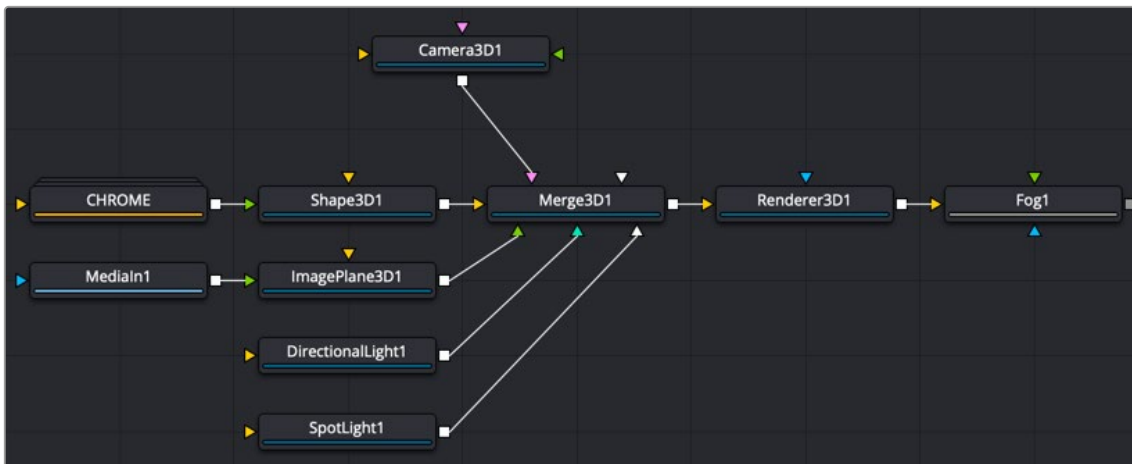
A Fog node added as a 2D post process after the Renderer 3D node

Inspector

Fog controls

Controls Tab

The Controls tab includes parameters for adjusting the density and color of the fog.

Z-Buffer Near Plane and Far Plane

These controls are used to select the extents of the fog within the scene. To pick a value, drag the Pick button to an area on the image being viewed where the plane is to be located.

The Near Plane is used to select the depth where the fog thins out to nothing. The Far Plane is used to select the depth at which the fog becomes opaque.

Z Depth Scale

This option scales the Z-buffer values by the selected amount. Raising the value causes the distances in the Z-channel to expand, whereas lowering the value causes the distances to contract. This is useful for exaggerating the fog effect.

Fog Color

This option displays and controls the current fog color. Alpha adjusts the fog's transparency value.

Fog Opacity

Use this control to adjust the opacity on all channels of the fog.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Deep Pixel nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Shader [Shd]

The Shader node

Shader Node Introduction

The Shader node can control the lighting, reflection mapping, and 3D shading of elements in a rendered image. The node relies on the presence of the normal map channel in a rendered image. If this channel is not present, this node has no effect.
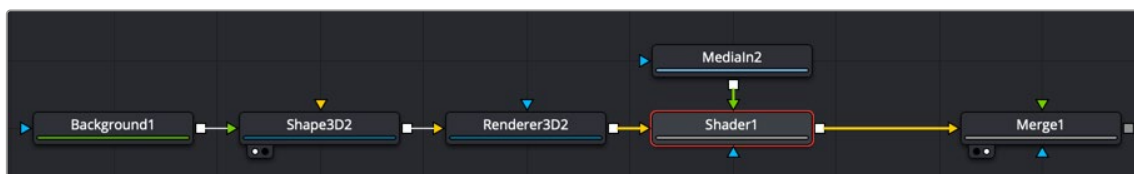
Inputs

The Shader node includes three inputs: one for the main image with normal map channels, one for a reflection map, and another for an effect mask to limit the area where the depth blur is applied.

— **Input:** This orange input is the only required connection. It accepts a 2D image that includes a normals channel.

— **Reflection Map Image:** The green reflection map image input projects an image onto all elements in the scene or to elements selected by the Object and Material ID channels in the Common Controls. Reflection maps work best as 32-bit floating point, equirectangular formatted images

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the shader to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.
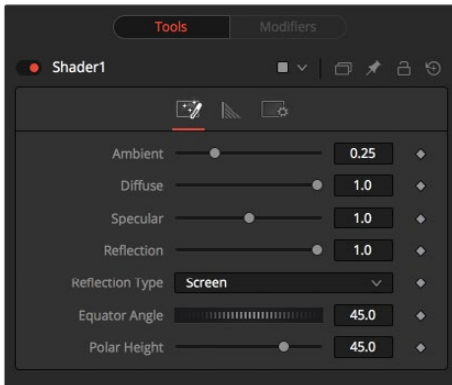
Basic Node Setup

The Shader node is inserted after a 2D image that contains a Normals channel. Below, a Renderer 3D is used to add a Normals channel to an image. The Shader node uses the normals for refining the surface appearance with a reflection map connected.

The Shader node using normals from a Renderer 3D and a reflection input

Inspector

Shader controls

Controls Tab

The Controls tab for the Shader node includes parameters for adjusting the overall surface reaction to light sources. You can modify the ambient, diffuse, specular, and reflection properties of the image connected to the orange image input.

Light Tab

The Controls tab includes parameters for basic lighting brightness and reflections.

Ambient

Ambient controls the Ambient color present in the scene or the selected object. This is a base level of light added to all pixels, even in completely shadowed areas.

Diffuse

This option controls the Diffuse color present in the scene or for the selected object. This is the normal color of the object, reflected equally in all directions.

Specular

This option controls the Specular color present in the scene or for the selected object. This is the color of the glossy highlights reflected toward the eye from a light source.

Reflection

This option controls the Reflection contribution in the scene or for the selected object. High levels make objects appear mirrored, while low levels overlay subtle reflections giving a polished effect. It has no effect if no reflection map is connected.

Reflection Type

This menu determines the type of reflection mapping used to project the image in the second input.

— **Screen:** Screen causes the reflection map to appear as if it were projected on to a screen behind the point of view.

— **Spherical:** Spherical causes the reflection map to appear as if it were projected on to a huge sphere around the whole scene.

— **Refraction:** Refraction causes the reflection map to appear as if it were refracting or distorting according to the geometry in the scene.

Equator Angle

Equator Angle controls the left to right angle of the light generated and mapped by the Shader node for the scene or the selected object.

Polar Height

Polar Height controls the top to bottom angle of the light generated and mapped by the Shader node for the scene or the selected object.
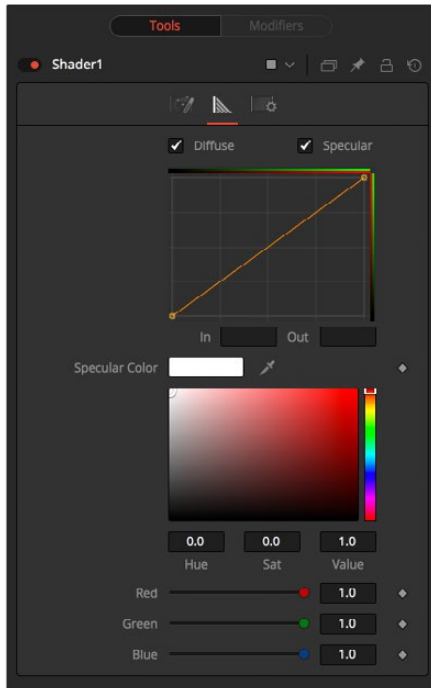
Shader tab controls

Shader Tab

The Shader tab is used to adjust the falloff of the Diffuse and Specular light and the tint color of the specular highlight.

Diffuse and Specular

When enabled, these checkboxes allow you to edit the Diffuse and/or Specular Shader curves in the Shader spline window.

In and Out

These options are used to display and edit point values on the spline.

Specular Color

Use the Diffuse curve to manipulate the diffuse shading and the Specular curve to affect the specular shading. Drag a box over several points to group-select them. Right-clicking displays a menu with options for adjusting the spline curves.
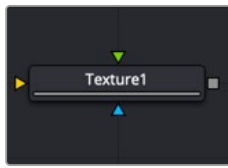
Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Deep Pixel nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Texture [Txr]

The Texture node

Texture Node Introduction

The Texture node controls the texture mapping of elements in a rendered image. The Texture node relies on the presence of U and V Map channels in a 3D-rendered image connected to the main Image input. If these channels are not present, this node has no effect.

> **NOTE:** Background pixels may have U and V values of 0.0, which set those pixels to the color of the texture's corner pixel. To restrict texturing to specific objects, use an effect mask based on the Alpha of the object or its Object or Material ID channel.

Inputs

The Texture node includes three inputs: one for the main image with UV map channels, one for a texture map image, and another for an effect mask to limit the area where the replace texture is applied.

— **Input:** This orange input accepts a 2D image that includes UV channels. If the UV channels are not in the images, this node has no effect.
— **Texture:** The green texture map input provides the texture that is wrapped around objects, replacing the current texture.
— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the texture to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Texture node is inserted after a 2D image that contains a Texture UV channel. Below, a Renderer 3D is used to add texture coordinates to 3D text. The Texture node can then be used to manipulate those coordinates using a new texture connected to the green texture input.
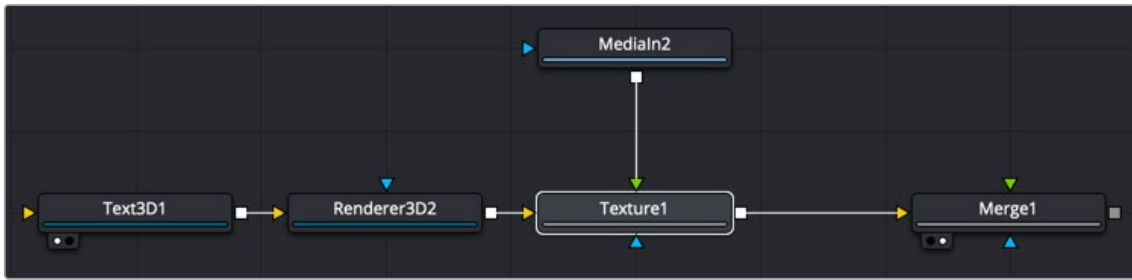
A Texture node used to manipulate the texture coordinates and add texture to a Text 3D node

Inspector
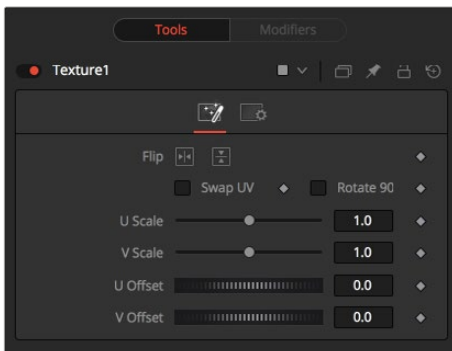
Texture controls

Texture Tab

The Texture tab controls allow you to flip, swap, scale, and offset the UV texture image connected to the texture input.

Flip Horizontal and Vertical

Use these two buttons to flip the texture image horizontally and/or vertically.

Swap UV

When this checkbox is selected, the U and V channels of the source image are swapped.

Rotate 90

The texture map image is rotated 90 degrees when this checkbox is enabled.

U and V Scale

These controls change the scaling of the U and V coordinates used to map the texture. Changing these values effectively enlarges and shrinks the texture map as it is applied.

U and V Offset

Adjust these controls to offset the U and V coordinates. Changing the values causes the texture to appear to move along the geometry of the object.
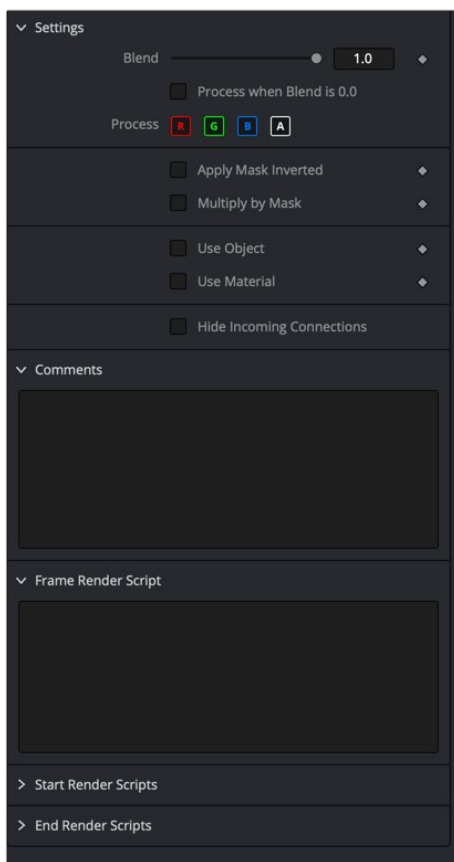
Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Deep Pixel nodes. These common controls are described in the following "The Common Controls" section.

The Common Controls

Nodes that handle Deep Pixel compositing operations share several identical controls in the Inspector. This section describes controls that are common among Deep Pixel nodes.

Inspector

Common Settings tab in Deep Pixel Nodes

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Deep Pixel category. The Settings controls are even found on third-party Deep Pixel-type plugin tools. The controls are consistent and work the same way for each tool although some tools do include one or two individual options that are also covered here.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this causes the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if processing of this node is scripted to trigger another task, but the value of the node is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the red button on a Blur tool is deselected, the blur is first applied to the image, and then the red channel from the original input is copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option causes the RGB values of the masked image to be multiplied by the mask channel's values. This causes all pixels of the image not in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels are used if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option is disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID is used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the viewer. The image or sequence must have been rendered from a 3D software package with those channels included.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off hardware-accelerated rendering using the graphics card in your computer. Enabled uses the hardware. Auto uses a capable GPU if one is available, but falls back to software rendering when a capable GPU is not available.

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 causes Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one whole frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Comments

The Comments field is used to add notes to a tool. Click in the field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Effect Nodes

This chapter details the Effect nodes in Fusion.

The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Duplicate [Dup]

The Duplicate node

Duplicate Node Introduction

Similar to the Duplicate 3D node, the Duplicate node can be used to quickly duplicate any 2D image, applying a successive transformation to each, and creating repeating patterns and complex arrays of objects. The options in the Jitter tab allow for non-uniform transformations, such as random positioning or sizes.

Inputs

The two inputs on the Duplicate node are used to connect a 2D image and an effect mask, which can be used to limit the area where duplicated objects appear.

— **Input:** The orange input is used for the primary 2D image that is duplicated.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the duplicated objects to appear only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.
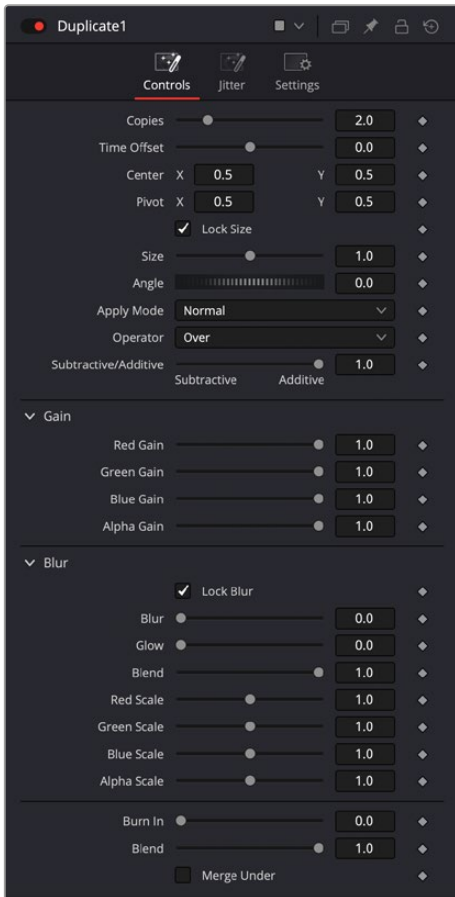
Basic Node Setup

The Duplicate node can be used in a variety of different ways and with a variety of different inputs. Below, to create motion graphics, a masked Background node creates a circular shape that is duplicated in the Duplicate node.

A Duplicate node used to create a repeating circular object

Inspector

Duplicate controls

Controls Tab

The Controls tab includes all the parameters you can use to create, offset, and scale copies of the object connected to the input on the node.

Copies

Use this slider to set the number of copies made. Each copy is a copy of the last copy. So, when set to 5, the parent is copied, then the copy is copied, then the copy of the copy is copied, and so on. This allows for some interesting effects when transformations are applied to each copy using the following controls.

Time Offset

Use the Time Offset slider to offset any animations that are applied to the original image by a set amount per copy. For example, set the value to -1.0 and use a square set to rotate on the Y-axis as the source. The first copy shows the animation from a frame earlier. The second copy shows animation from a frame before that, and so forth. This can be used with great effect on textured planes, for example, where successive frames of a clip can be shown.

Center

The X and Y Center controls set the offset position applied to each copy. An X offset of 1 would offset each copy 1 unit along the X-axis from the last copy.
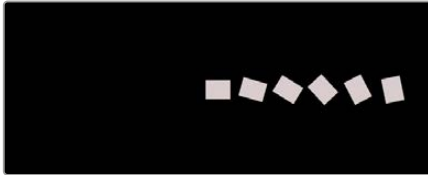
Pivot

The Pivot controls determine the position of the pivot point used when changing the size, position, or angle of each copy. The pivot does not move with the original object or the duplicated array. To have the pivot follow the army, you must modify the pivot controls.
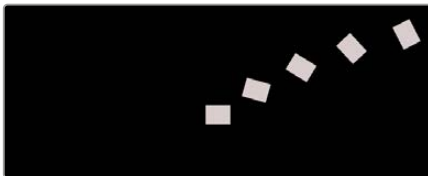
Size

The Size control determines how much scaling to apply to each copy.

Angle

The Angle control sets the amount of Z rotation applied to each copy. The angle adjustment is linear based on the location of the pivot point.

Angle adjustment with centered pivot

Angle adjustment with offset pivot

Apply Mode

The Apply Mode setting determines the math used when blending or combining duplicated objects that overlap.

— **Normal:** The default mode uses the foreground object's Alpha channel as a mask to determine which pixels are transparent and which are not. When this is active, another menu shows possible operations, including Over, In, Held Out, Atop, and XOr.

— **Screen:** Screen blends the objects based on a multiplication of their color values. The Alpha channel is ignored, and layer order becomes irrelevant. The resulting color is always lighter. Screening with black leaves the color unchanged, whereas screening with white always produces white. This effect creates a similar look to projecting several film frames onto the same surface. When this is active, another menu shows possible operations, including Over, In, Held Out, Atop, and XOr.

— **Dissolve:** Dissolve mixes overlapping objects. It uses a calculated average of the objects to perform the mixture.

— **Multiply:** Multiplies the values of a color channel. This gives the appearance of darkening the object as the values are scaled from 0 to 1. White has a value of 1, so the result would be the same. Gray has a value of 0.5, so the result would be a darker object or, in other words, an object half as bright.

— **Overlay:** Overlay multiplies or screens the color values of the foreground object, depending on the color values of the object behind. Patterns or colors overlay the existing pixels while preserving the highlights and shadows of the color values of the objects behind the foreground objects. Objects behind other objects are not replaced but mixed with the front objects to reflect the original lightness or darkness of the objects behind.

— **Soft Light:** Soft Light darkens or lightens the foreground object, depending on the color values of the objects behind them. The effect is similar to shining a diffused spotlight on the image.

— **Hard Light:** Hard Light multiplies or screens the color values of the foreground object, depending on the color values of the objects behind them. The effect is similar to shining a harsh spotlight on the image.

— **Color Dodge:** Color Dodge uses the foreground object's color values to brighten the objects behind them. This is similar to the photographic practice of dodging by reducing the exposure of an area of a print.

— **Color Burn:** Color Burn uses the foreground object's color values to darken the objects behind them. This is similar to the photographic practice of burning by increasing the exposure of an area of a print.

— **Darken:** Darken looks at the color information in each channel and selects the object's foreground or background's color value, whichever is darker, as the result color. Pixels lighter than the blended colors are replaced, and pixels darker than the blended color do not change.

— **Lighten:** Lighten looks at the color information in each channel and selects the object's foreground or background's color values, whichever is lighter, as the result color value. Pixels darker than the blended color are replaced, and pixels lighter than the blended color do not change.

— **Difference:** Difference looks at the color information in each channel and subtracts the foreground object's color values from the background object's color values or the behind object's values from the foreground object's values, depending on which has the higher brightness value. Blending with white inverts the color. Blending with black produces no change.

— **Exclusion:** Exclusion creates an effect similar to but lower in contrast than the Difference mode. Blending with white inverts the base color values. Blending with black produces no change.

— **Hue:** Hue creates a result color with the luminance and saturation of the background objects color values and the hue of the foreground object's color values.

— **Saturation:** Saturation creates a result color with the luminance and hue of the base color and the saturation of the blend color.

— **Color:** Color creates a result color with the luminance of the background object's color value and the hue and saturation of the objects in the foreground. This preserves the gray levels in the image and is useful for colorizing monochrome objects.

— **Luminosity:** Luminosity creates a color using the hue and saturation of the background object and the luminance of the foreground object. This mode creates an inverse effect from that of the Color mode.

Operator

This menu is used to select the Operation mode used when the duplicate objects overlap. Changing the Operation mode changes how the overlapping objects are combined. This drop-down menu is visible only when the Apply mode is set to Normal.

The formula used to combine pixels in the Duplicate node is always (fg object * x) + (bg object * y). The different operations determine what x and y are, as shown in the description for each mode.

The Operator Modes are as follows:

— **Over:** The Over mode adds the foreground object to the background object by replacing the pixels in the background with the pixels from the Z wherever the foreground object's Alpha channel is greater than 1.

`x = 1, y = 1 – [foreground object Alpha]`

— **In:** The In mode multiplies the Alpha channel of the background object against the pixels in the foreground object. The color channels of the foreground object are ignored. Only pixels from the foreground object are seen in the final output. This essentially clips the foreground object using the mask from the background object.

`x = [background Alpha], y = 0`

— **Held Out:** Held Out is essentially the opposite of the In operation. The pixels in the foreground object are multiplied against the inverted Alpha channel of the background object.

`x = 1 – [background Alpha], y = 0`

— **Atop:** Atop places the foreground object over the background object only where the background object has a matte.

`x = [background Alpha], y = 1 – [foreground Alpha]`

— **XOr:** XOr combines the foreground object with the background object wherever either the foreground or the background have a matte, but never where both have a matte.

`x = 1 – [background Alpha], y = 1–[foreground Alpha]`

Subtractive/Additive

This slider controls whether Fusion performs an Additive composite, a Subtractive composite, or a blend of both when the duplicate objects overlap. This slider defaults to Additive assuming the input image's Alpha channel is premultiplied (which is usually the case). If you don't understand the difference between Additive and Subtractive compositing, here's a quick explanation.

An Additive blend operation is necessary when the foreground image is premultiplied, meaning that the pixels in the color channels have been multiplied by the pixels in the Alpha channel. The result is that transparent pixels are always black since any number multiplied by 0 always equals 0. This obscures the background (by multiplying with the inverse of the foreground Alpha), and then adds the pixels from the foreground.

A Subtractive blend operation is necessary if the foreground image is not premultiplied. The compositing method is similar to an additive composite, but the foreground image is first multiplied by its Alpha, to eliminate any background pixels outside the Alpha area.

While the Additive/Subtractive option is often an either/or mode in most other applications, the Duplicate node lets you blend between the Additive and Subtractive versions of the compositing operation. This can be useful for dealing with problem composites with bright or dark edges.

For example, using Subtractive merging on a premultiplied image may result in darker edges, whereas using Additive merging with a non-premultiplied image causes any non-black area outside the foreground's Alpha to be added to the result, thereby lightening the edges. By blending between Additive and Subtractive, you can tweak the edge brightness to be just right for your situation.

Gain

The Gain RGB controls multiply the values of the image channel linearly. All pixels are multiplied by the same factor, but the effect is larger on bright pixels and smaller on dark pixels. Black pixels are not changed since multiplying any number times 0 always equals 0.

Alpha Gain linearly scales the Alpha channel values of objects in front. This effectively reduces the amount that the objects in the background are obscured, thus brightening the overall result. When the Subtractive/Additive slider is set to Additive with Alpha Gain set to 0.0, the foreground pixels are simply added to the background.

When Subtractive/Additive slider is set to Subtractive, this controls the density of the composite, similarly to Blend.

All Gain values will compound based on the number of duplications.

Blur

Adds a blurring effect to the duplicated layers.

— **Lock Blur:** Locks the X and Y Blur sliders together for symmetrical blurring. This is enabled by default. When the Lock Blur control is deselected, independent control over each axis is provided.
— **Blur:** Sets the amount of blur applied to the duplicated layers in the tool. The Blur amount will not compound based on the number of duplications.
— **Glow:** Adds a glow effect to the blur of the duplicated layers.
— **Blend:** The Blend slider determines the percentage of the affected image that is mixed with original image. It blends in more of the original image as the value gets closer to 0.
— **RGBA Scale:** Allows adjusting the strength of the individual Red, Green, Blue, and Alpha channels to the blur of the duplicated layers.

Burn In

The Burn In control adjusts the amount of Alpha used to darken the objects that fall behind other objects, without affecting the amount of foreground objects added. At 0.0, the blending behaves like a straight Alpha blend, in contrast to a setting of 1.0 where the objects in the front are effectively added on to the objects in the back (after Alpha multiplication if in Subtractive mode). This gives the effect of the foreground objects brightening the objects in the back, as with Alpha Gain. In fact, for Additive blends, increasing the Burn In gives an identical result to decreasing Alpha Gain.

Blend

This blend control is different from the Blend slider in the Common Settings tab. Changes made to this control apply the blend between objects. The Blend slider fades the results of the last object first, the penultimate after that, and so on. The blending is divided between 0 and 1, with 1 being all objects are fully opaque and 0 being only the original object showing.

Merge Under

This checkbox reverses the layer order of the duplicated elements, making the last copy the bottommost layer and the first copy the topmost layer.

Jitter Tab

The options in the Jitter tab allow you to randomize the position, rotation, size, and color of all the copies created in the Controls tab.
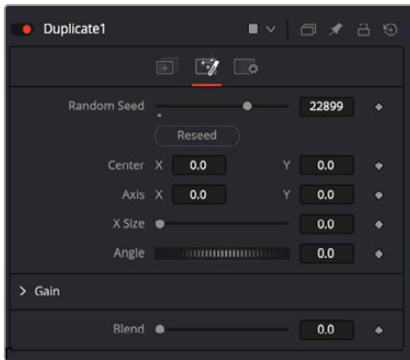
Duplicate Jitter tab

Random Seed

The Random Seed slider and Reseed button are used to generate a random starting point for the amount of jitter applied to the duplicated objects. Two Duplicate nodes with identical settings but different random seeds produce two completely different results.

Center X and Y

Use these two controls to adjust the amount of variation in the X and Y position of the duplicated objects.

Axis X and Y

Use these two controls to adjust the amount of variation in the rotational pivot center of the duplicated objects. This affects only the additional jitter rotation, not the rotation produced by the Rotation settings in the Controls tab.

X Size

Use this control to adjust the amount of variation in the Scale of the duplicated objects.

Angle

Use this dial to adjust the amount of variation in the Z rotation of the duplicated objects.

Gain

The Gain RGBA controls randomly multiply the values of the image channel linearly.

Blend

Changes made to this control randomize the blend between objects.

Common Controls

Settings Tab

The Settings tab controls are common to all Effect nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Highlight [HIL]

The Highlight node

Highlight Node Introduction

The Highlight filter creates star-shaped highlights or glints in bright regions of the image, similar to a lens star filter effect.

Inputs

There are three Inputs on the Highlight node: one for the image, one for the effects mask, and another for a highlight mask.

— **Input:** The orange input is used for the primary 2D image that gets the highlight applied.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input restricts the highlight to be within the pixels of the mask. An effects mask is applied to the tool after the tool is processed.

— **Highlight Mask:** The Highlight node supports pre-masking using the white highlight mask input. The image is filtered before the highlight is applied. The highlight is then merged back over the original image. Unlike regular effect masks, it does not crop off highlights from source pixels when the highlight extends past the edges of the mask.

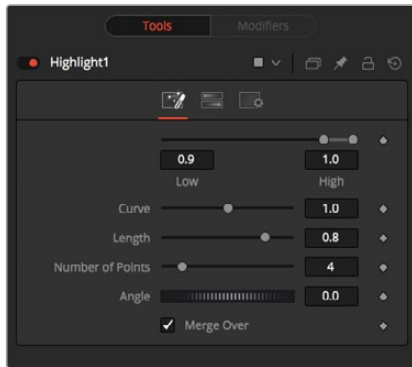Highlight masks are identical to effects masks in every other respect.

Basic Node Setup

The Highlight node below is used to create glint-type highlights on an incoming image. The highlight mask is used to limit the area where the effect is applied.

A Highlight node applied to an image, with a highlight mask limiting the area of the effect

Inspector

Highlight controls

Controls Tab

The Controls tab includes parameters for the highlight style except for color, which is handled in the Color Scale tab.

Low and High

This range control designates the range of Luminance values in the image that generates highlights. Values less than the Low value do not receive highlights. Values above the High value receive the full highlight effect.

Curve

The Curve value changes the drop-off over the length of the highlight. Higher values cause the brightness of the flares to drop off closer to the center of the highlight, whereas lower values drop off farther from the center.

Length

This designates the length of the flares from the highlight.
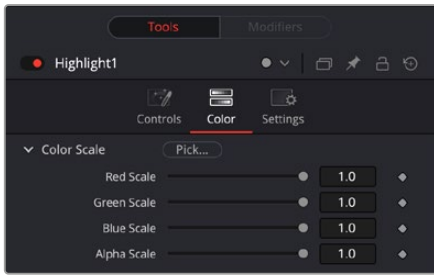
Number of Points

This determines the number of flares emanating from the highlight.

Angle

Use this control to rotate the highlights.

Merge Over

When enabled, the effect is overlaid on the original image. When disabled, the output is the highlights only. This is useful for downstream color correction of the highlights.

Highlight Color Scale controls

Color Scale Tab

The Color Scale tab controls the color of the highlight.

By click and holding on the Pick button, then dragging the pointer over the viewer, you can select a specific color from the image.

Red, Green, and Blue Scale

Moving the sliders of one or all of these channels down changes the falloff color of the highlight.
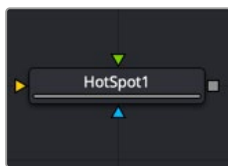
Alpha Scale

Moving the Alpha slider down makes highlight falloff more transparent.

Common Controls

Setting Tab

The Settings tab controls are common to all Effect nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Hot Spot [Hot]

The Hot Spot node

Hot Spot Node Introduction

The Hot Spot node is used to create lens flare, spotlight, and burn/dodge effects of various types.

In the real world, lens flares occur when extremely bright light sources in the scene by the reflections are reflected off elements inside the lens of the camera. One might see lens flares in a shot when viewing a strong light source through a camera lens, like the sun or another bright star.

Inputs

There are three inputs on the Hot Spot node: one for the image, one for the effects mask, and another for an Occlusion image.

— **Input:** The required orange input is used for the primary 2D image that gets the hot spot applied.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input restricts the hot spot to be within the pixels of the mask. An effects mask is applied to the tool after the tool is processed.

— **Occlusion:** The green Occlusion input accepts an image to provide the occlusion matte. The matte is used to block the hot spot, causing it to "wink." The white pixels in the image occlude the hot spot. Gray pixels partially suppress the hot spot.

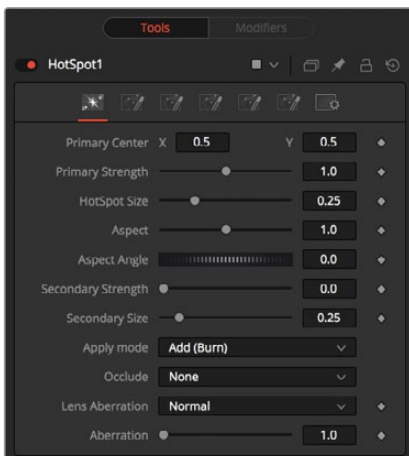# Basic Node Setup

The Hot Spot node is not a stand-alone generator, so it must have an image input that gets the hot spot applied.
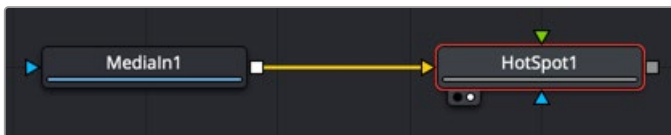
Hot Spot node applied to an image

Inspector

Hot Spot controls

Hot Spot Tab

The Hotspot tab is used to control the primary and secondary hot spots. You can adjust their position, size, strength, angle, and apply mode.

Primary Center X and Y

This is the position of the primary hot spot within the scene. Secondary lens elements and reflections are positioned relative to the position of the primary hot spot.

Primary Strength

This control determines the brightness of the primary hot spot.

Hot Spot Size

This control determines the diameter of the primary hot spot. A value of 1.0 represents a circle the full width of the image.

Aspect

This controls the aspect of the spot. A value of 1.0 produces a perfectly circular hot spot. Values above 1.0 elongate the circle horizontally, and values below 1.0 elongate the circle vertically.

Aspect Angle

This control can be used to rotate the primary hot spot.

Secondary Strength

This control determines the strength, which is to say the brightness, of the secondary hot spot. The secondary hot spot is a reflection of the primary hot spot. It is always positioned on the opposite side of the image from the primary hot spot.

Secondary Size

This determines the size of the secondary hot spot.

Apply Mode

This control determines how the hot spot affects the underlying image.

— **Add (Burn):** This causes the spots created to brighten the image.
— **Subtract (Dodge):** This causes the spots created to dim the image.
— **Multiply (Spotlight):** This causes the spots created to isolate a portion of the image with light and to darken the remainder of the image.

Occlude

This menu is used to select which channel of the image connected to the Hot Spot node's Occlusion input is used to provide the occlusion matte. Occlusion can be controlled from Alpha or R, G, or B channels of any image connected to the Occlusion input on the node's tile.

Lens Aberration

Aberration changes the shape and behavior of the primary and secondary hot spots.

— **In and Out Modes:** Elongates the shape of the hot spot into a flare. The hot spot stretches toward the center when set to In mode and stretches toward the corners when set to Out mode.
— **Flare In and Flare Out Modes:** This option is a lens distortion effect that is controlled by the movement of the lens effect. Flare In causes the effect to become more severe, the closer the hot spot gets to the center. Flare Out causes the effect to increase as the hot spot gets closer to the edges of the image.
— **Lens:** This mode emulates a round, ringed lens effect.

Aberration

The Aberration slider controls the overall strength of the lens aberration effect.
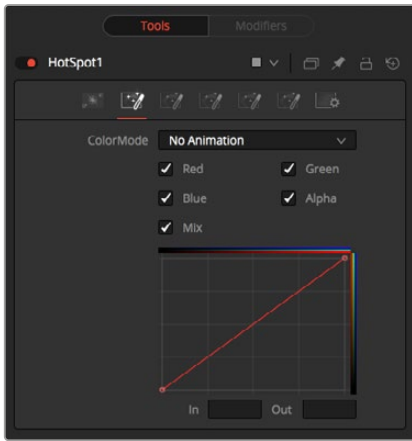
Hot Spot color controls

Color Tab

The Color tab is used to modify the color of the primary and secondary hot spots.

Color Mode

This menu allows you to choose between animated or static color modifications using the small curves editor in the Inspector.

— **None:** The default None setting retains a static curve adjustment for the entire range.
— **Animated Points:** This setting allows the color curves in the spline area to be animated over time. Once this option is selected, moving to the desired frame and making a change in the Spline Editor sets a keyframe.
— **Dissolve mode:** Dissolve mode is mostly obsolete and is included for compatibility reasons only.

Color Channel and Mix

When selected, these checkboxes enable the editing of the chosen splines in the small Inspector Spline Editor. The Mix checkbox enables the Mix Spline, which is used to determine the influence of the controls that the Radial tab has along the radius of the hot spot.

Red, Green, Blue, and Alpha Splines

The Spline Window shows the curves for the individual channels. It is a miniature Spline Editor. The Red, Green, Blue, and Alpha splines are used to adjust the color of the spotlight along the radius of the hot spot.

The vertical axis represents the intensity or strength of the color channel. The horizontal axis represents the hot spot position along the radius, from the left outside edge to the inside right edge.

The default curve indicates that the red, green, blue, and Alpha channels all have a linear falloff.

Mix Spline

The Mix spline is used to determine the influence that the Radial controls have along the radius of the hot spot. The horizontal axis represents the position along the circle's circumference, with 0 being 0 degrees and 1.0 being 360 degrees. The vertical axis represents the amount of the radial hot spot to blend with the color hot spot. A value of 0 is all radial hot spot, while a value of 1.0 is all color hot spot.

> **NOTE:** Right-clicking in the LUT displays a contextual menu with options related to modifying spline curves.

For more information on the LUT Editor, *see Chapter 69, "Using Viewers."* in the DaVinci Resolve Reference Manual, or Chapter 7 in the Fusion Reference Manual.
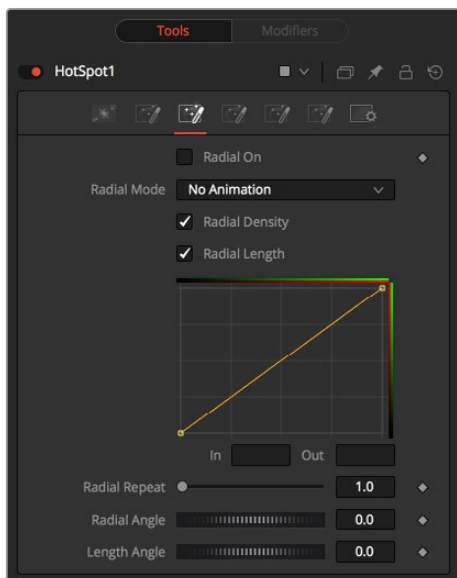
Hot Spot Radial tab

Radial Tab

Radial On

This control enables the Radial splines. Otherwise, the radial matte created by the splines is not applied to the hot spot, and the Mix spline in the color controls does not affect the hot spot.

Radial Mode

Similar to the Color mode menu, this menu allows you to choose between animated or static radial hot spot modifications using the small curves editor in the Inspector.

— **No Animation:** The default setting retains a static curve adjustment for the entire range.
— **Animated Points:** This setting allows the radial curves in the spline area to be animated over time. Once this option is selected, moving to the desired frame and making a change in the Spline Editor sets a keyframe.

The Interpolated Values option is mostly obsolete and is included for compatibility reasons only.

Radial Length and Radial Density Splines

The Spline window shows curves for the Length and Density of the hot spot. It is a miniature Spline Editor. The key to these splines is realizing that the horizontal axis in Inspector's Spline Editor represents a position around the circumference of the hot spot. A value of 0.0 is 0 degrees, and 1.0 is 360 degrees. With that in mind, the length determines the radius of light making up the hot spot along the circumference. The density represents how bright the light is along the circumference.

Radial Repeat

This control repeats the effect of the radial splines by x number of times. For example, a repeat of 2.0 causes the spline to take effect between 0 and 180 degrees instead of 0 and 360, repeating the spline between 180 and 360.

Length Angle

This control rotates the effect of the Radial Length spline around the circumference of the hot spot.

Density Angle

This control rotates the effect of the Radial Density spline around the circumference of the hot spot.
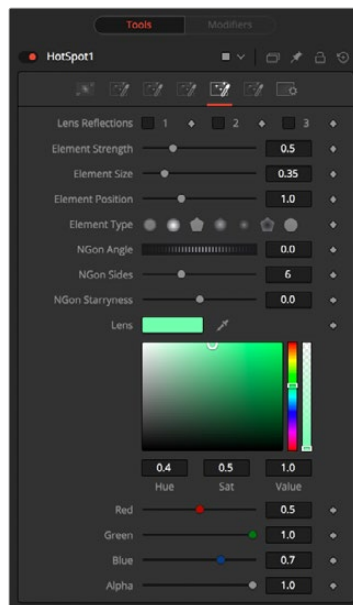
> **NOTE:** Right-clicking in the spline area displays a contextual menu containing options related to modifying spline curves.

A complete description of LUT Editor controls and options can be found in Chapter 45, "LUT Nodes," in the Fusion Reference Manual or Chapter 105 in the DaVinci Resolve Reference Manual.

L1, L2, and L3 Tab
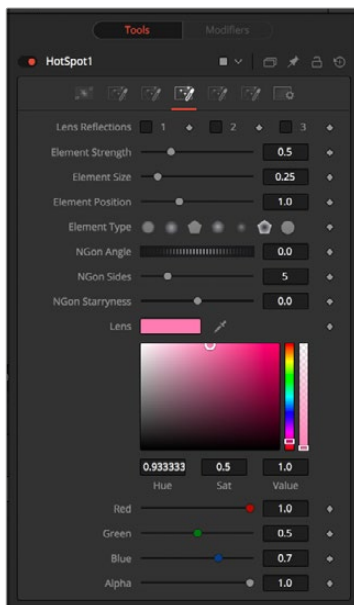
L1 tab   L2 tab   L3 tab
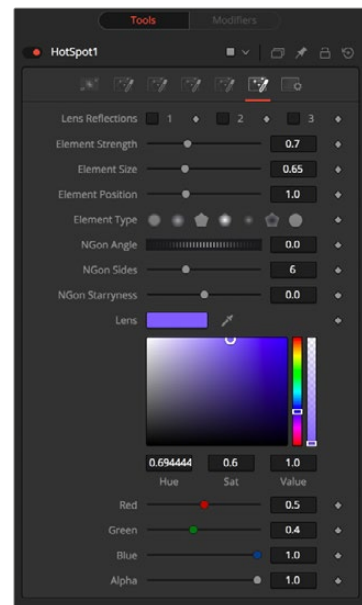
Lens Reflect Tabs

The three Lens Reflect tabs are used to enable and design additional lens flare elements beyond the primary and secondary hot spots.

Lens Reflect 1-3

Each of these three checkboxes enables a pair of lens reflection elements that you can modify using the controls in this tab. The parameters affect all the enabled Lens reflection elements in this tab.

Element Strength

This determines the brightness of element reflections.

Element Size

This determines the size of element reflections.

Element Position

This determines the distance of element reflections from the axis. The axis is calculated as a line between the hot spot position and the center of the image.

Element Type

Use this group of buttons to choose the shape and density of the element reflections. The presets available are described below.

— **Circular:** This creates slightly soft-edged circular shaped reflections.
— **Soft Circular:** This creates very soft-edged circular shaped reflections.
— **Circle:** This creates a hard-edged circle shape.
— **NGon Solid:** This creates a filled polygon with a variable number of sides.
— **NGon Star:** This creates a very soft-edged star shape with a variable number of sides.
— **NGon Shaded Out:** This creates soft-edged circular shapes.
— **NGon Shaded In:** This creates a polygon with a variable number of sides, which has a very soft reversed (dark center, bright radius) circle.
— **NGon Angle:** This control is used to determine the angle of the NGon shapes.
— **NGon Sides:** This control is used to determine the number of sides used when the Element Type is set to Ngon Star, Ngon Shaded Out, and Ngon Shaded In.
— **NGon Starriness:** This control is used to bend polygons into star shapes. The higher the value, the more star-like the shape.

Lens Color Controls

These controls determine the color of the lens that affects the colors of the reflections. To choose a lens color, pick one from a displayed image or enter RGBA values using the sliders or input boxes.

Common Controls

Settings Tab

The Settings tab controls are common to all Effect nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Pseudo Color [PsCl]

The Pseudo Color node

Pseudo Color Node Introduction

The Pseudo Color node provides the ability to produce variations of an image's color based on waveforms generated by the node's controls. Static or animated variances of the original image can be produced.

Inputs

There are two Inputs on the Pseudo Color node: one for an image and one for an effects mask.

— **Input:** The orange input is used for the primary 2D image that gets its color modified.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input restricts the pseudo color to be within the pixels of the mask. An effects mask is applied to the tool after the tool is processed.
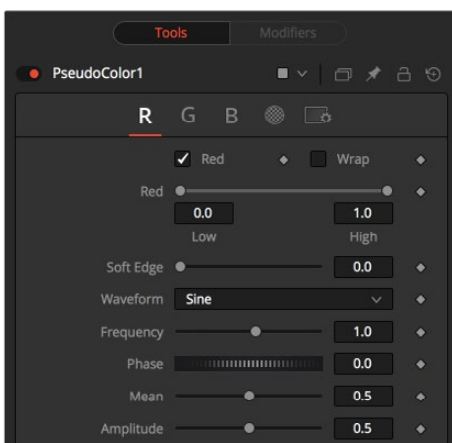
Basic Node Setup

The Pseudo Color node is not a stand-alone generator, so it must have an image input that it uses to generate variations in colors.
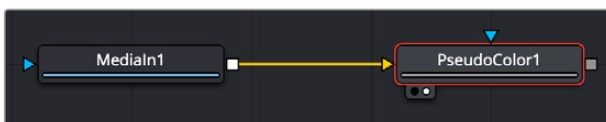
Pseudo Color node applied to an image

Inspector

Pseudo Color RGBA controls

Red/Green/Blue/Alpha Tabs

The node's controls are separated into four identical tabs, one for each of the RGBA color channels.

Color Checkbox

When enabled, the Pseudo Color node affects this color channel.

Wrap

When enabled, waveform values that exceed allowable parameter values are wrapped to the opposite extreme.

High and Low

High and Low determine the range to be affected by the node in a specific color channel.

Soft Edge

This slider determines the soft edge of color transition.

Waveform

This selects the type of waveform to be created by the generator. Four waveforms are available: Sine, Triangle, Sawtooth, and Square.

Frequency

This controls the frequency of the waveform selected. Higher values increase the number of occurrences of the variances.

Phase

This modifies the Phase of the waveform. Animating this control produces color cycling effects.

Mean

This determines the level of the waveform selected. Higher values increase the overall brightness of the channel until the allowed maximum is reached.

Amplitude

Amplitude increases or decreases the overall power of the waveform.

Common Controls

Settings Tab

The Settings tab controls are common to all Effect nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Rays [CIR]

The Rays node

Rays Node Introduction

Rays is a modified zoom blur effect that radiates through an object from a specified point.

Inputs

There are two inputs on the Rays node: one for the image and one for the effects mask.

— **Input:** The orange input is used for the primary 2D image that gets the rays applied to it.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input restricts the rays to be within the pixels of the mask. An effects mask is applied to the tool after the tool is processed.
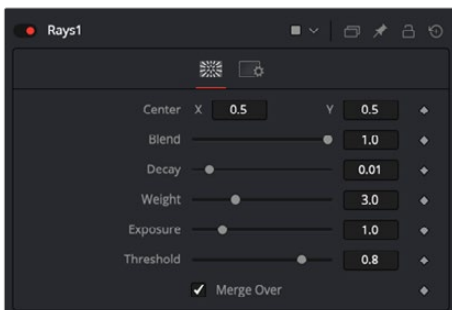
Basic Node Setup

The Rays node works best when the image or graphic connected to the orange input includes an Alpha channel from which the rays emit.

Rays node set up to emit from a line of text

Inspector

Rays node controls

Controls Tab

The Controls tab contains all the primary controls necessary for customizing the rays.

Center X and Y

This coordinate control and related viewer crosshair set the center point for the light source.

Blend

Sets the percentage of the original image that's blended with the light rays.

Decay

Sets the length of the light rays.

Weight

Sets the falloff of the light rays.

Exposure

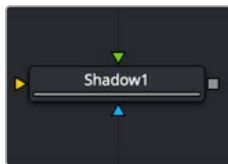Sets the intensity level of the light rays.

Threshold

Sets the luminance limit at which the light rays are produced.

Common Controls

Settings Tab

The Settings tab controls are common to all Effect nodes, so their descriptions can be found in the "The Common Controls" section at the end of this chapter.

Shadow [Sh]

The Shadow node

Shadow Node Introduction

Shadow is a versatile node used in the creation of a drop shadow, based on the Alpha channel in an image. Optionally, a second image can be used as a depth matte to distort the shadow based on the varying depth in a background image.

Input

The three inputs on the Shadow node are used to connect a 2D image that causes the shadow. A depth map input and an effect mask can be used to limit the area where trails appear. Typically, the output of the shadow is then merged over the actual background in the composite.

— **Input:** The orange input is used for the primary 2D image with Alpha channel that is the source of the shadow.

— **Depth:** The green Depth map input takes a 2D image as its input and extracts a depth matte from a selected channel. The light Position and Distance controls can then be used to modify the appearance of the shadow based on depth.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the area where the shadow appears. An effects mask is applied to the tool after the tool is processed.

> **NOTE:** The Shadow node is designed to create simple 2D drop shadows. Use a Spot Light node and an Image Plane 3D node for full 3D shadow casting.
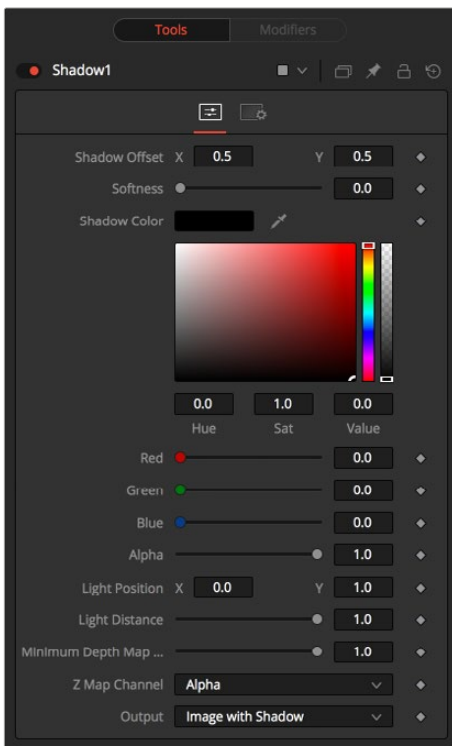
Basic Node Setup

Below, the Shadow node uses the output of an image with Alpha and connects to the foreground of a Merge. The shadow is shown over the background input to the Merge.
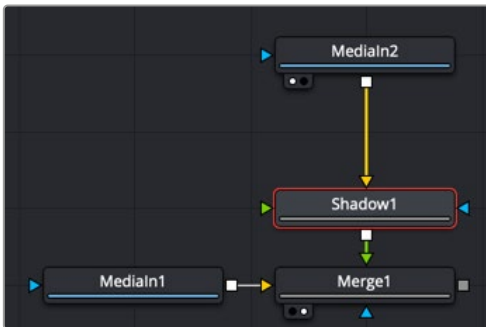
A Shadow node is generated from the MediaIn2 and shown over MediaIn1

Inspector

Shadow node controls

Controls Tab

The Controls tab contains all the primary controls necessary for customizing the shadow appearance.

Shadow Offset

This control sets the X and Y position of the shadow. When the Shadow node is selected, you can also adjust the position of the Shadow Offset using the crosshair in the viewer.

Softness

Softness controls how blurry the shadow's edges appear.

Shadow Color

Use this control to select the color of the shadow. The most realistic shadows are usually not totally black and razor sharp.

Light Position

This control sets the position of the light relative to the shadow-casting object. The Light Position is only taken into consideration when the Light Distance slider is not set to infinity (1.0).

Light Distance

This slider varies the apparent distance of the light between infinity (1.0) and zero distance from the shadow-casting object. The advantage of setting the Light Distance is that the resulting shadow is more realistic-looking, with the further parts of the shadow being longer than those that are closer.

Minimum Depth Map Light Distance

This control is active when an image is connected to the shadow's Depth Map input. The slider is used to control the amount that the depth map contributes to the Light Distance. Dark areas of a depth map make the shadow deeper. White areas bring it closer to the camera.

Z Map Channel

This menu is used to select which color channel of the image connected to the node's Depth Map input is used to create the shadow's depth map. Selections exist for the RGB and A, Luminance, and Z-buffer channels.

Output

This menu determines if the output image contains the image with shadow applied or the shadow only.
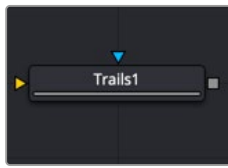
The shadow only method is useful when color correction, perspective, or other effects need to be applied to the resulting shadow before it is merged back with the object.

Common Controls

Settings Tab

The Settings tab controls are common to all Effect nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Trails [Trls]

The Trails node

Trails Node Introduction

The Trails node is used to create a ghost-like after-trail of the image. This creates an interesting effect when applied to moving images with an Alpha channel. Unlike a directional blur, only the preceding motion of an image is displayed as part of the effect. Since the trail effect is based on an image buffer, it requires you to play or activate the pre-roll for some number of frames before you see the effect.

Input

The two inputs on the Trails node are used to connect a 2D image and an effect mask that can be used to limit the area where trails appear.

— **Input:** The orange input is used for the primary 2D image that receives the trails applied.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the area where the trails effect appears. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The output of an animated Text node is connected to the input of the Trails node. Trails are generated based on the motion of the text. The Reset button must be pressed in the Inspector between each preview, or the trails will accumulate.
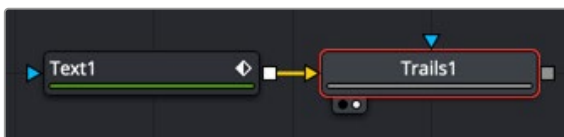
A Trails node generates trails for
the animation in the Text node

Inspector

Trails node controls

Controls Tab

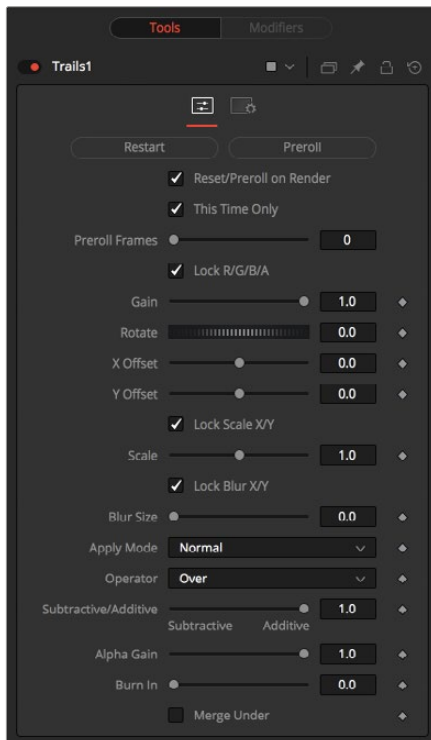The Controls tab contains all the primary controls necessary for customizing the trails.

Restart

This control clears the image buffer and displays a clean frame, without any of the ghosting effects.

Preroll

This makes the Trails node pre-render the effect by the number of frames on the slider.

Reset/Preroll on Render

When this checkbox is enabled, the Trails node resets itself when a preview or final render is initiated. It pre-rolls the designated number of frames.

This Time Only

Selecting this checkbox makes the pre-roll use this current frame only and not the previous frames.

Preroll Frames

This determines the number of frames to pre-roll.

Lock RGBA

When selected, this checkbox allows the Gain of the color channels to be controlled independently. This allows for tinting of the Trails effect.

Gain

The Gain control affects the overall intensity and brightness of the image in the buffer. Lower values in this parameter create a much shorter, fainter trail, whereas higher values create a longer, more solid trail.

Rotate

The Rotate control rotates the image in the buffer before the current frame is merged into the effect. The offset is compounded between each element of the trail. This is different than each element of the trail rotating on its pivot point. The pivot remains over the original object.

Offset X/Y

These controls offset the image in the buffer before the current frame is merged into the effect. Control is given over each axis independently. The offset is compounded between each element of the trail.

Lock Scale X/Y

When selected, this checkbox allows the X- and Y-axis scaling of the image buffer to be manipulated separately for each axis.

Scale

The Scale control resizes the image in the buffer before the current frame is merged into the effect. The size is compounded between each element of the trail.

Lock Blur X/Y

When selected, this checkbox allows the blurring of the image buffer to be controlled separately for each axis.

Blur Size

The Blur Size control applies a blur to the trails in the buffer before the current frame is merged into the effect. The blur is compounded between each element of the trail.

Apply Mode

The Apply Mode setting determines the math used when blending or combining the trailing objects that overlap.

— **Normal:** The default mode uses the foreground object's Alpha channel as a mask to determine which pixels are transparent and which are not. When this is active, another menu shows possible operations, including Over, In, Held Out, Atop, and XOr.

— **Screen:** Screen blends the objects based on a multiplication of their color values. The Alpha channel is ignored, and layer order becomes irrelevant. The resulting color is always lighter. Screening with black leaves the color unchanged, whereas screening with white always produces white. This effect creates a similar look to projecting several film frames onto the same surface. When this is active, another menu shows possible operations, including Over, In, Held Out, Atop, and XOr.

— **Dissolve:** Dissolve mixes overlapping objects. It uses a calculated average of the objects to perform the mixture.

— **Multiply:** Multiplies the values of a color channel. This gives the appearance of darkening the object as the values are scaled from 0 to 1. White has a value of 1, so the result would be the same. Gray has a value of 0.5, so the result would be a darker object or, in other words, an object half as bright.

— **Overlay:** Overlay multiplies or screens the color values of the foreground object, depending on the color values of the background object. Patterns or colors overlay the existing pixels while preserving the highlights and shadows of the color values of the objects behind the foreground objects. The objects behind the foreground objects are not replaced but mixed with the foreground objects to reflect the original lightness or darkness of the background objects.

— **Soft Light:** Soft Light darkens or lightens the foreground object, depending on the color values of the objects behind them. The effect is similar to shining a diffused spotlight on the image.

— **Hard Light:** Hard Light multiplies or screens the color values of the foreground object, depending on the color values of the objects behind them. The effect is similar to shining a harsh spotlight on the image.

— **Color Dodge:** Color Dodge uses the foreground object's color values to brighten the objects behind them. This is similar to the photographic practice of dodging by reducing the exposure of an area of a print.

— **Color Burn:** Color Burn uses the foreground object's color values to darken the objects behind them. This is similar to the photographic practice of burning by increasing the exposure of an area of a print.

— **Darken:** Darken looks at the color information in each channel and selects the color value from the object in front or behind, whichever is darker. Pixels lighter than the blended colors are replaced, and pixels darker than the blended color do not change.

— **Lighten:** Lighten looks at the color information in each channel and selects the color value from the object in front or behind, whichever is lighter. Pixels darker than the blended color are replaced, and pixels lighter than the blended color do not change.

— **Difference:** Difference looks at the color information in each channel and subtracts the foreground object's color values from the background object's color values or vice versa, depending on which has the higher brightness value. Blending with white inverts the color. Blending with black produces no change.

— **Exclusion:** Exclusion creates an effect similar to but lower in contrast than the Difference mode. Blending with white inverts the base color values. Blending with black produces no change.

— **Hue:** Hue creates color with the luminance and saturation of the background object's color and the hue of the foreground object's color.

— **Saturation:** Saturation creates color with the luminance and hue of the base color and the saturation of the blend color.

— **Color:** Color creates color with the luminance of the background object's color and the hue and saturation of the object in front. This preserves the gray levels in the image and is useful for colorizing monochrome objects.

— **Luminosity:** Luminosity creates color with the hue and saturation of the background object's color and the luminance of the foreground object's color. This mode creates an inverse effect from that of the Color mode.

Operator

This menu is used to select the Operation mode used when the trailing objects overlap. Changing the Operation mode changes how the overlapping objects are combined to produce a result. This drop-down menu is visible only when the Apply mode is set to Normal.

The formula used to combine pixels in the trails node is always (fg object * x) + (bg object * y). The different operations determine what x and y are, as shown in the description for each mode.

The Operator Modes are as follows:

— **Over:** The Over mode adds the foreground object to the background object by replacing the pixels in the background with the pixels from the Z wherever the foreground object's Alpha channel is greater than 1.

  `x = 1, y = 1 – [foreground object Alpha]`

— **In:** The In mode multiplies the Alpha channel of the background object against the pixels in the foreground object. The color channels of the foreground object are ignored. Only pixels from the foreground object are seen in the final output. This essentially clips the foreground object using the mask from the background object.

  `x = [background Alpha], y = 0`

— **Held Out:** Held Out is essentially the opposite of the In operation. The pixels in the foreground object are multiplied against the inverted Alpha channel of the background object.

  `x = 1 – [background Alpha], y = 0`

— **Atop:** Atop places the foreground object over the background object only where the background object has a matte.

  `x = [background Alpha], y = 1 – [foreground Alpha]`

— **XOr:** XOr combines the foreground object with the background object wherever either the foreground or the background have a matte, but never where both have a matte.

  `x = 1 – [background Alpha], y = 1 – [foreground Alpha]`

Subtractive/Additive

This slider controls whether Fusion performs an Additive composite, a Subtractive composite, or a blend of both when the trailing objects overlap. This slider defaults to Additive assuming the input image's Alpha channel is premultiplied (which is usually the case). If you don't understand the difference between Additive and Subtractive compositing, below is a quick explanation.

> **NOTE:** An Additive blend operation is necessary when the foreground image is premultiplied, meaning that the pixels in the color channels have been multiplied by the pixels in the Alpha channel. The result is that transparent pixels are always black since any number multiplied by 0 always equals 0. This obscures the background (by multiplying with the inverse of the foreground Alpha), and then adds the pixels from the foreground.
>
> A Subtractive blend operation is necessary if the foreground image is not premultiplied. The compositing method is similar to an additive composite, but the foreground image is first multiplied by its Alpha, to eliminate any background pixels outside the Alpha area.
>
> Although the Additive/Subtractive option is often an either/or checkbox in other software, the Trails node lets you blend between the Additive and Subtractive versions of the compositing operation. This can be useful when dealing with problem edges that are too bright or too dark.
>
> For example, using Subtractive merging on a premultiplied image may result in darker edges, whereas using Additive merging with a non-premultiplied image causes any non-black area outside the foreground's Alpha to be added to the result, thereby lightening the edges. By blending between Additive and Subtractive, you can tweak the edge brightness to be just right for your situation.

Alpha Gain

Alpha Gain linearly scales the Alpha channel values of the trailing objects in front. This effectively reduces the amount that the trailing objects in the background are obscured, thus brightening the overall result. When the Subtractive/Additive slider is set to Additive with Alpha Gain set to 0.0, the foreground pixels are added to the background.

When the Subtractive/Additive slider is set to Subtractive, this controls the density of the composite, similar to Blend.

Burn In

The Burn In control adjusts the amount of Alpha used to darken the objects that trail under other objects, without affecting the amount of foreground objects added. At 0.0, the blending behaves like a straight Alpha blend. At 1.0, the objects in the front are effectively added onto the objects in the back (after Alpha multiplication if in Subtractive mode). This gives the effect of the foreground objects brightening the objects in the back, as with Alpha Gain. In fact, for Additive blends, increasing the Burn In gives an identical result to decreasing Alpha Gain.

Merge Under

When enabled, the current image is placed under the generated trail, rather than the usual, over top operation. The layer order of the trailing elements is also reversed, making the last trail the topmost layer.

Common Controls

Settings Tab

The Settings tab controls are common to all Effect nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

TV [TV]

The TV node

TV Node Introduction

The TV node is a simple node designed to mimic some of the typical flaws seen in analog television broadcasts and screens. This Fusion-specific node is mostly obsolete when using DaVinci Resolve because of the more advanced Analog Damage ResolveFX.

Input

The two inputs on the TV node are used to connect a 2D image and an effect mask, which can be used to limit the area where the TV effect appears.

— **Input:** The orange input is used for the primary 2D image that gets the TV distortion applied.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the area where the TV effect to appears. An effects mask is applied to the tool after the tool is processed.
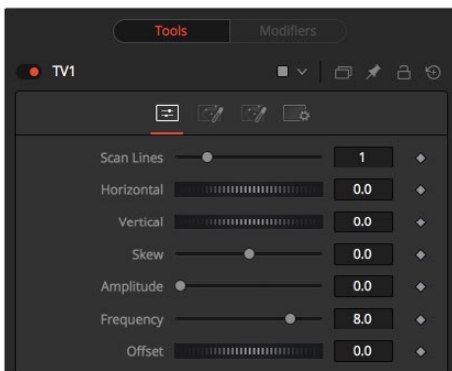
Basic Node Setup

The output of an image is connected to the input of the TV node. The style of TV interference is then customized using the controls in the Inspector.

The TV node simulates TV-style flaws in the image connected to the orange input

Inspector

TV node controls

Controls Tab

The Controls tab is the first of three tabs used to customize the analog TV distortion. The Controls tab modifies the scan lines and image distortion of the effect.

Scan Lines

This slider is used to emulate the interlaced look by dropping lines out of the image. Setting it to black, with a transparent Alpha, drops a line. A value of 1 (default) drops every second line. A value of 2 shows one line, and then drops the second and third and repeats. A value of zero turns off the effect.

Horizontal

Use this slider to apply a simple Horizontal offset to the image.

Vertical

Use this slider to apply a simple Vertical offset to the image.

Skew

This slider is used to apply a diagonal offset to the image. Positive values skew the image to the top left. Negative values skew the image to the top right. Pixels pushed off frame wrap around and reappear on the other side of the image.

Amplitude

The Amplitude slider can be used to introduce smooth sine wave-type deformation to the edges of the image. Higher values increase the intensity of the deformation. Use the Frequency control to determine how often the distortion is repeated.

Frequency

The Frequency slider sets the frequency of the sine wave used to produce distortion along the edges of the image when the amplitude control is greater than 1.

Offset

Use Offset to adjust the position of the sine wave, causing the deformation applied to the image via the Amplitude and Frequency controls to see across the image.
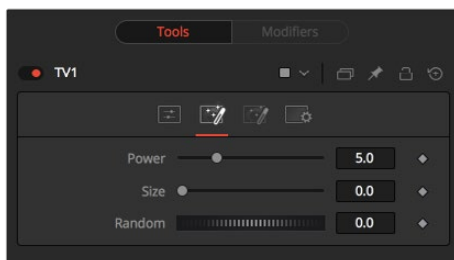
The TV Noise tab

Noise Tab

The Noise tab is the second of three tabs used to customize the analog TV distortion. The Noise tab modifies the noise in the image to simulate a weak analog antenna signal.
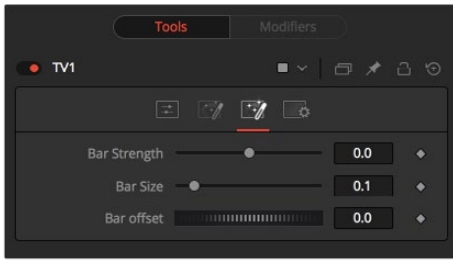
Power

Increase the value of this slider above 0 to introduce noise into the image. The higher the value, the stronger the noise.

Size

Use this slider to scale the noise map larger.

Random

If this thumbwheel control is set to 0, the noise map is static. Change the value over time to cause the static to change from frame to frame.

The TV Roll Bar tab

Roll Bar Tab

The Roll Bar tab is the third of three tabs used to customize the analog TV distortion. The Roll Bar tab animates the bar.

Bar Strength

At the default value of 0, no bar is drawn. The higher the value, the darker the area covered by the bar becomes.

Bar Size

Increase the value of this slider to make the bar taller.

Bar Offset

Animate this control to scroll the bar across the screen.
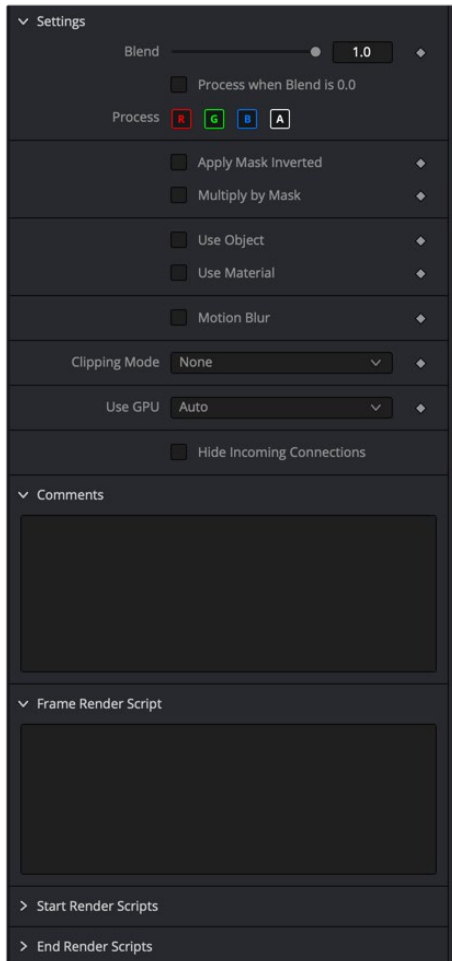
Common Controls

Settings Tab

The Settings tab controls are common to all Effect nodes, so their descriptions can be found in the following "The Common Controls" section.

The Common Controls

Effect nodes share several identical controls in the Inspector. This section describes controls that are common among Effect nodes.

Inspector

The Common Effects Settings tab

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Effects category. The Settings controls are even found on third-party Effects-type plugin tools. The controls are consistent and work the same way for each tool, although some tools do include one or two individual options, which are also covered here.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. This causes the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This is useful when this node is scripted to trigger another task, but the blend is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the red button on a Blur tool is deselected, the blur is first applied to the image, and then the red channel from the original input is copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this possess a set of like RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Control tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option causes the RGB values of the masked image to be multiplied by the mask channel's values. This causes all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels are used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option is disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on coverage and background channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID is used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the view. The image or sequence must have been rendered from a 3D software package with those channels included.

Clipping Mode

This option determines how the domain of definition rendering handles edges. The Clipping mode is most important when blur or softness is applied, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **None:** Setting this option to None does not perform any source image clipping. Any data required to process the node's effect that would usually be outside the upstream DoD is treated as black/transparent.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off hardware-accelerated rendering using the graphics card in your computer. Enabled uses the hardware. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 causes Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one full frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows for the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Film Nodes

This chapter details the Film nodes in Fusion.

The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Cineon Log [Log]
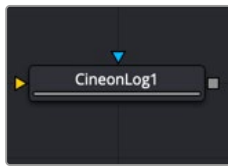
The Cineon Log node

Cineon Log Node Introduction

The Cineon Log node is used to convert several different log camera formats to linear gamma and back again. Although the name implies that it should be used with Cineon files, it handles "log" gamma from many different digital cinema sources such as Blackmagic Design, Arri, and Red cameras.

Input

There are two Inputs on the Cineon Log node: one for the log image and one for the effects mask.

— **Input:** The orange input is used for the primary 2D image that gets the highlight applied.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input restricts the log conversion to be within the pixels of the mask. An effects mask is applied to the tool after the tool is processed.
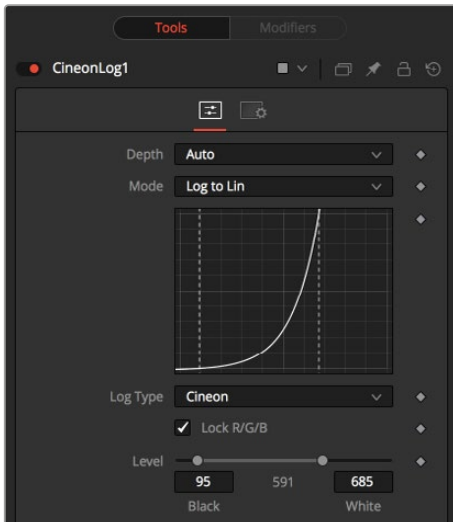
Basic Node Setup

The Cineon Log node is placed directly after a MediaIn node in DaVinci Resolve or a Loader node in Fusion Studio. It is also commonly placed before a MediaOut or Saver node to convert back to a Log-encoded image.
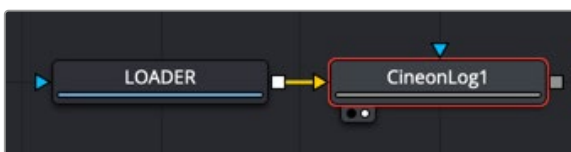
A Cineon Log node placed after a
Loader node in Fusion Studio

Inspector

Cineon Log controls

Controls Tab

The Controls tab includes settings for converting from log gamma to linear or from linear to log. You first select the Mode and then the Log Type. For instance, choose Log to Lin from the Mode menu, and then select BMD Film if you are compositing with a RAW clip from a Blackmagic Design camera. Those settings output a linear image ready for compositing.

Depth

The Depth menu is used to select the color depth used to process the input image. The default option is Auto. Auto determines the color depth based on the file format loaded. For example, JPEG files automatically process at 8 bit because the JPEG file format does not store color depths greater than 8. Blackmagic RAW files load at Float, etc. If the color depth of the format is undetermined, the default depth defined in the Frame Format preferences is used.

Mode

The Mode menu offers two options: one for converting log images to linear and one for converting linear images to logarithmic.

Log Type

The Log Type menu allows you to select the source of the file. Typically, you select the camera used to create the image, although the Josh Pines option is specific to film scan workflows. This menu contains the following camera log types:

— Cineon
— Canon Log
— Panasonic V-Log
— Viper Film Stream

— Arri Log C
— Nikon N Log
— Red Log Film
— ACESlog

— BMD Film
— Panalog
— Sony S-Log

Lock RGB

When enabled, the settings in this tab affect all color channels equally.

Disable this control to convert the red, green, and blue channels of the image using separate settings for each channel.

Level

Use this range control to set the black level and white level in the log image before converting. The left handle adjusts the black level, while the right handle adjusts the white level. Pixels with values in log space below the black level become out-of-range values below 0.0. Pixels with values above the white level become out-of-range values above 1.0 after conversion.

When processing in floating-point color space, both negative and high out-of-range values are preserved. When using 16-bit or 8-bit mode, the out-of-range values are clipped.

Soft Clip (Knee)

The Soft Clip control is used to draw values that are out of range back into the image. This is done by smoothing the conversion curve at the top and bottom of the curve, allowing more values to be represented.

Applying a soft clip of any value other than 1 causes the node to process at 16-bit integer, eliminating all out-of-range values that do not fit within the soft clip.

Film Stock Gamma, Conversion Gamma, and Conversion Table

These controls are used to set the response curves of the logarithmic data during conversion. In addition to the settings above, a custom ASCII file Lookup Table (LUT) can be created with specific conversion values. The ASCII LUT file can be loaded using the Browse button.

Common Controls

Settings Tab

The Settings tab controls are common to all Film nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Black Rolloff

Since a mathematical log() operation on a value of zero or lower results in invalid values, Fusion clips values below 1e-38 (0 followed by 38 zeros) to 0 to ensure correct results. This is almost never an issue, since values that small have no visual impact on an image. To see such tiny values, you would have to add three Brightness Contrast nodes, each with a gain set to 1,000,000. Even then, the values would hover very close to zero.

We have seen processes where instead of cropping these minimal values, they are instead scaled. So values between 0.0 and 1e-16 are scaled between 1e-18 and 1e-16. The idea is to crush the majority of the visual range in a float image into values very near to zero, then expand them again, forcing a gentle ramp to produce a small ramp in the extreme black values. Should you find yourself facing a color pipeline using this process, here is how you can mimic it with the help of a Custom node.

The process involves converting the log image to linear with a very small gamma and a wider than normal black level to white level (e.g., conversion gamma of 0.6, black of 10, white of 1010). This crushes most of the image's range into very small values. This is followed by a Custom node (described below), and then by a linear to log conversion that reverses the process but uses a slightly higher black level. The difference between the black levels defines the falloff range.

Since this lifts the blacks, the image is usually then converted back to linear one more time, using more traditional values (i.e., 95-685) to reset the black point.
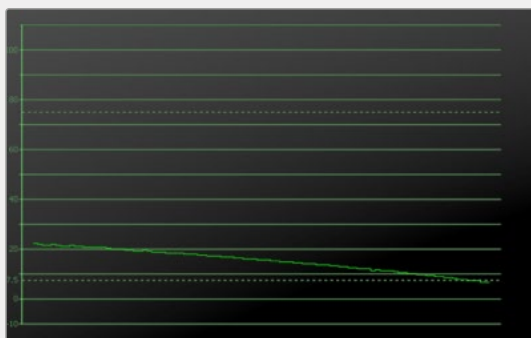
The Custom node should use the following equation in the red, green, and blue expressions:

```
if (c1< 1e-16, 1e-18 + (c1/1e-16)*(1e-16 - 1e-18), c1)
```

Falloff Comparison
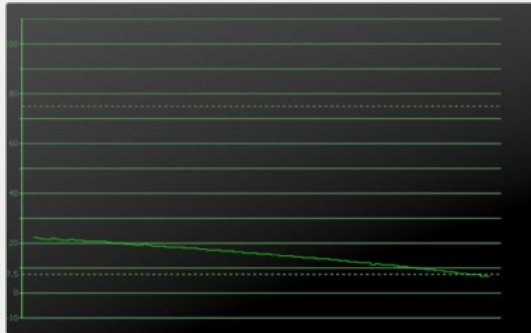
The black falloff from the native Fusion process

Virtually identical black falloff from the ramped clipping process

Film Grain [FGr]
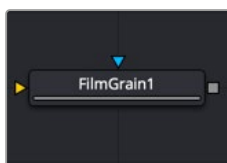
The Film Grain node

Film Grain Node Introduction

The Film Grain node adds generated film grain to an image. Typically, when compositing with different elements shot on film and in digital, the grain is removed for compositing operations and then reapplied to the final composite. This helps create the appearance that all elements were shot as a single scene with the same film stock.

NOTE: Although more accurate, the Film Grain node does not replace the older Grain node, which is still provided to allow older compositions to load and render, but in almost every case, it is better to use the Film Grain node.

Input

There are two inputs on the Film Grain node: one for the image and one for the effects mask.

— **Input:** The orange input is used for the primary 2D image that gets the grain applied.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the grain to be within the pixels of the mask. An effects mask is applied to the tool after the tool is processed.

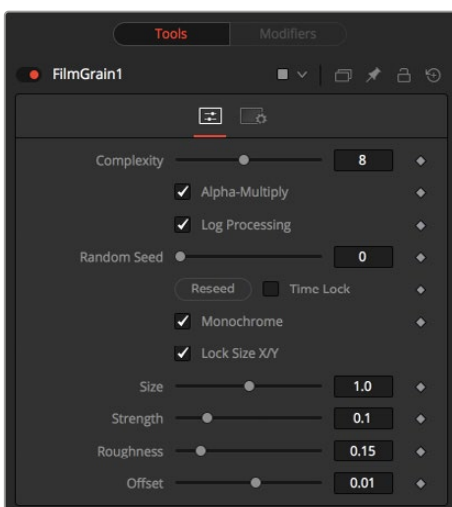## Basic Node Setup

The Film Grain node is commonly used before a MediaOut node in DaVinci Resolve or before a Saver node in Fusion Studio. Since grain is often removed from source material to achieve cleaner keys and composite, film grain is added back before the output to create a more realistic composite.
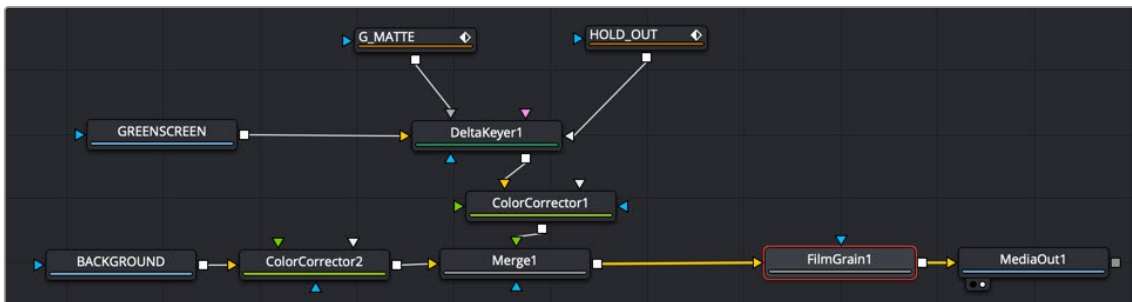
Film Grain node adding grain back before output

Inspector

Film Grain controls

Controls Tab

The Controls tab includes all the parameters for modifying the appearance of the film grain.

Complexity

The Complexity setting indicates the number of "layers" of grain applied to the image. With a complexity of 1, only one grain layer is calculated and applied to the image. When complexity is set to 4, the node calculates four separate grain layers and applies the mean combined result of each pass to the final image. Higher complexities produce visually more sophisticated results, without the apparent regularity often perceivable in digitally-produced grain.

Alpha Multiply

When the Alpha Multiply checkbox is enabled, the Film Grain node multiplies its results by the source image's Alpha channel. This is necessary when working with post-multiplied images to ensure that the grain does not affect areas of the image where the Alpha is 0.0 (transparent).

> **NOTE:** Since it is impossible to say what the final value of semitransparent pixels in the image are until after they are composited with their background, you should avoid applying log-processed grain to the elements until after they have been composited. This ensures that the strength of the grain is accurate.

Log Processing

When this checkbox is enabled (default), the grain applied to the image has its intensity applied nonlinearly to match the grain profile of most film. Roughly speaking, the intensity of the grain increases exponentially from black to white. When this checkbox is disabled, the grain is applied uniformly, regardless of the brightness of the affected pixel.

One of the primary features of grain in film is that the appearance of the grain varies radically with the exposure so that there appears to be minimal grain present in the blacks, with the amount and deviation of the grain increasing as the pixels exposure increases. In a film negative, the darkest portions of the developed image appear entirely opaque, and this obscures the grain. As the negative becomes progressively clearer, more of the grain becomes evident in the result. Chemical differences in the R, G, B, layer's response to light also cause each color component of the film to present a different grain profile, typically with the blue channel presenting the most significant amount of grain.

As a result, an essential control in the Film Grain node is the Log Processing checkbox, which should be enabled when matching film, and disabled when working with images that require a more linear grain response. Having this checkbox enabled closely mimics the results of preceding the old Grain node with a Linear to Log conversion and following with a Log to Linear conversion immediately after.

Seed

The Seed slider and Reseed button are presented whenever a Fusion node relies on a random result. Two nodes with the same seed values produce the same random results. Click on the Reseed button to randomly select a new seed value, or adjust the slider to select a new seed value manually.

Time Lock

Enabling Time Lock stops the random seed from generating new grain on every frame.

Monochrome

When the Monochrome checkbox is enabled (default), the grain is applied to the red, green, and blue color channels of the image equally. When deselected, individual control over the Size, Strength, and Roughness of the grain in each channel becomes possible.

Lock Size X/Y

Deselect the Lock Size X/Y checkbox to control the size of the grain along the X- and Y-axis individually.

Size

The grain size is calculated relative to the size of a pixel. Consequently, changing the resolution of the image does not impact the relative appearance of the grain. The default grain size of 1.0 produces grain kernels that cover roughly 2 pixels.

Strength

Grain is expressed as a variation from the original color of a pixel. The stronger the grain's strength, the wider the possible variation from the original pixel value. For example, given a pixel with an original value of p, and a Grain node with complexity = 1 size = 1; roughness = 0; log processing = off; the grain produces an output value of p +/- strength. In other words, a pixel with a value of 0.5 with a grain strength of 0.02 could end up with a final value between 0.48 and 0.52.

Once again, that's a slight oversimplification, especially when the complexity exceeds 1. Enabling the Log Processing checkbox also causes that variation to be affected such that there is less variation in the blacks and more variation in the whites of the image.

> **NOTE:** When visualizing the effect of the grain on the image, the more mathematically inclined may find it helps to picture a sine wave, where each lobe of the sine wave covers 1 pixel when the Grain Size is 1.0. The Grain Size controls the frequency of the sine wave, while the Grain Strength controls its amplitude. Again, this is something of an oversimplification.

Roughness

The Roughness slider applies low frequency variation to give the impression of clumping in the grain. Try setting the roughness to 0, and observe that the grain produced has a very even luminance variation across the whole image. Increase the roughness to 1.0 and observe the presence of "cellular" differences in the luminance variation.

Offset

The Offset control helps to match the intensity of the grain in the deep blacks by offsetting the values before the intensity (strength) of the grain is calculated. So an offset of 0.1 would cause a pixel with a value of 0.1 to receive grain as if its value was 0.2.
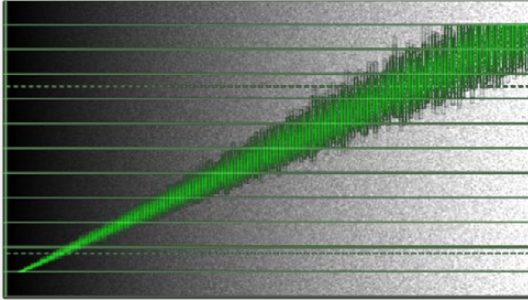
Common Controls

Settings Tab

The Settings tab controls are common to all Film nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Processing Examples

Log Processing On

In the default setting, the different amounts of Grain are applied to the blacks and the whites of the image

Log Processing Off
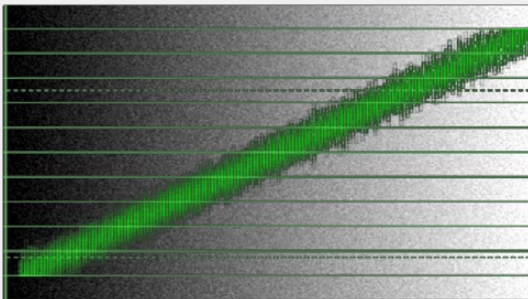
When Log processing is off, the Grain is applied evenly to the entire image, as shown here

Grain [Grn]

The Grain node

Grain Node Introduction

The Grain node offers an older film grain emulation than the more modern and accurate Film Grain node. The Grain node is still provided to allow older compositions to load and render, but in almost every case, it is better to use the Film Grain node.

Input

There are two Inputs on the Grain node: one for the 2D image and one for the effects mask.

— **Input:** The orange input is used for the primary 2D image that gets the grain applied.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the grain to be within the pixels of the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Grain node is commonly placed before a MediaOut node in DaVinci Resolve, or before a Saver node in Fusion Studio. It adds back grain previously removed in a composite.
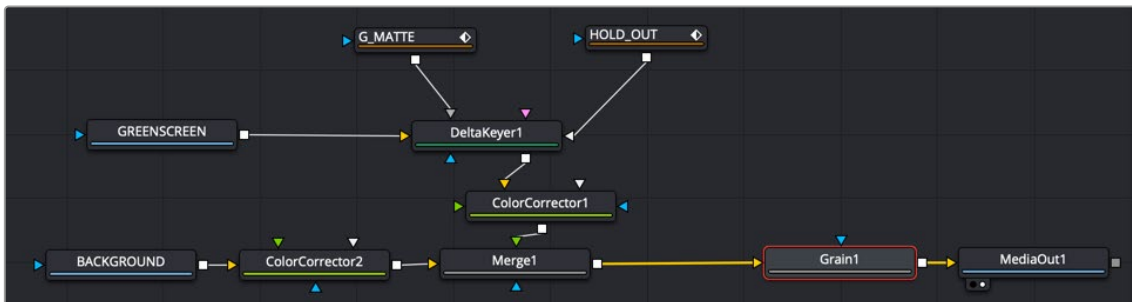
A Grain node used to add grain back for a more realistic composite

Inspector
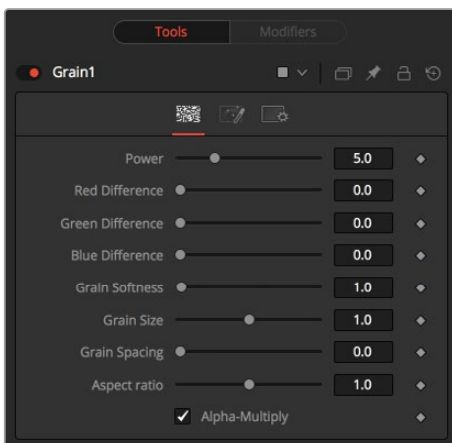
Grain controls

Controls Tab

The Controls tab includes all the parameters for modifying the appearance of the grain.

Power

This slider determines the strength of the grain. A higher value increases visibility, making the grain more prevalent.

RGB Difference

Separate Red, Green, and Blue sliders are used to modify the strength of the effect on a per channel basis.

Grain Softness

This slider controls the blurriness or fuzziness of the grain. Smaller values cause the grain to be more sharp or coarse.

Grain Size

This slider determines the size of the grain particles. Higher values increase the grain size.
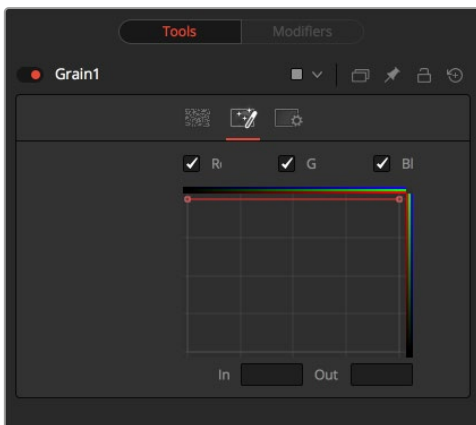
Grain Spacing

This slider determines the density or amount of grain per area. Higher values cause the grain to appear more spaced out.

Aspect Ratio

This slider adjusts the aspect of the grain so that it can be matched with anamorphic images.

Alpha-Multiply

When enabled, this checkbox multiplies the image by the Alpha, clearing the black areas of any grain effect.

Grain Spread controls

Spread Tab

The Spread tab uses curves for the red, green, and blue channels to control the amount of grain over each channel's tonal range.

RGB Checkboxes

The red, green, and blue checkboxes enable each channel's custom curve, allowing you to control how much grain appears in each channel. To mimic usual film responses, more grain would appear in the blue channel than the red, and the green channel would receive the least. Right-clicking in the spline area displays a contextual menu containing options related to modifying spline curves.

For more information on the LUT Editor's controls *see Chapter 105, "LUT Nodes,"* in the DaVinci Resolve Reference Manual, or Chapter 43 in the Fusion Reference Manual.

In and Out

This control provides direct editing of points on the curve by setting In/Out point values.

Examples

Default Spread

In the default setting, the grain is applied evenly to the entire image, as shown here. However, film often shows a different amount of grain in the blacks, mids, and whites.
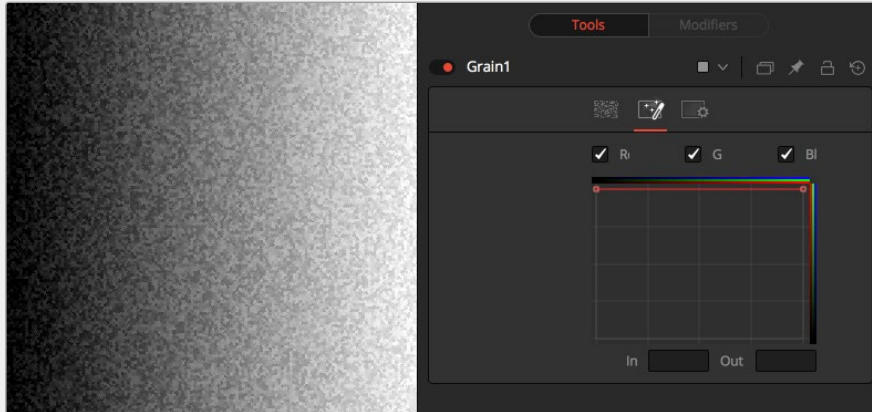
Bell-Shaped Spread

Setting a bell shape is often a good starting point to create a more realistic-looking grain. Here we have a non-uniform distribution with different amounts of grain in the red, green, and blue channels.

In both examples, the grain's power has been exaggerated to show the effect a bit better.
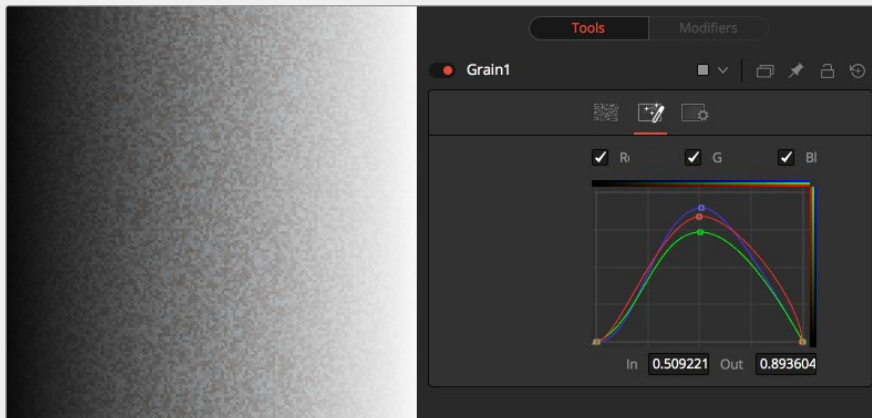
Common Controls

Setting Tab

The Settings tab controls are common to all Film nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Light Trim [LT]

The Light Trim node

Light Trim Node Introduction

This node emulates film scanner light trims. By design, this node works best with logarithmic data, such as the images stored by Cineon, Arri, or Blackmagic RAW files. When logarithmic data is provided, the Light Trim node can be used to increase or decrease the apparent exposure level of the image.

Inputs

There are two Inputs on the Light Trim node: one for the 2D image and one for the effects mask.

— **Input:** The orange input is used for the primary Log 2D image that gets its exposure adjusted.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the exposure change to be within the pixels of the mask. An effects mask is applied to the tool after the tool is processed.
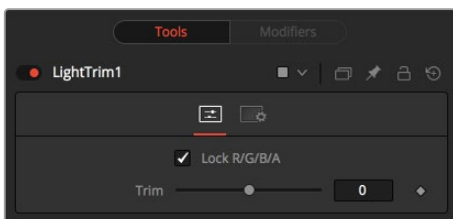
Basic Node Setup

The Light Trim node is placed after a LOG clip but before the Log clip is converted by a Cineon LOG node.

A Light Trim node used to adjust exposure on a LOG clip

Inspector

Light Trim controls

Controls Tab

The Controls tab includes a single slider that adjusts the exposure of the image.

Lock RGBA

When selected, the Lock RGBA control collapses control of all image channels into one slider. This selection is on by default. To manipulate the various color channels independently, deselect this checkbox.
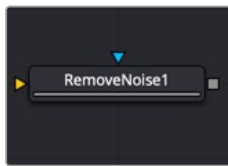
Trim

This slider shifts the color in film, optical printing, and lab printing points. 8 points equals one stop of exposure.

Common Controls

Settings Tab

The Settings tab controls are common to all Film nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Remove Noise [RN]

The Remove Noise node

Remove Noise Node Introduction

The Remove Noise node provides simple noise management. The basic operation is that the node blurs the image channels, and then compares the blurred image to the original to extract the noise. A sharpness is then applied to the image, except where noise was detected.

To use this node, view the image and look at the red channel. Then increase the Red Softness until the grain appears to be gone. Next, increase the sharpness until the detail reappears, but stop before the grain reappears. Repeat for the green and blue channels.

Inputs

There are two inputs on the Remove Noise node: one for the 2D image and one for the effects mask.

— **Input:** The orange input is used for the primary 2D image that gets noise removed.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the noise removal change to be within the pixels of the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Remove Noise node can be used on any clip with noise. For example, it is used below to remove noise prior to keying the clip using the DeltaKeyer.
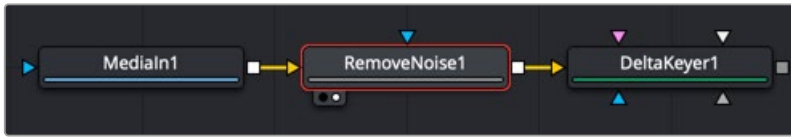
A Remove Noise node used to remove noise prior to keying

Inspector
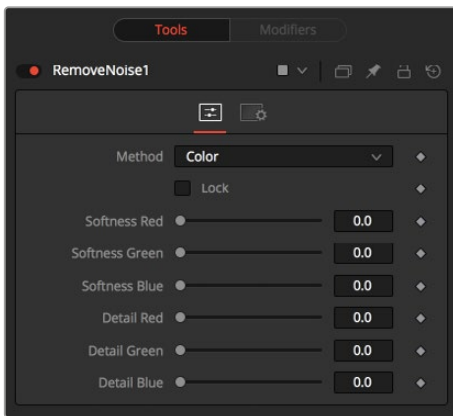
Remove Noise controls

Controls Tab

The Controls tab switches the noise removal between two methods: Color and Chroma. When the Method is set to Color, the Controls tab adjusts the amount of blur and sharpness individually for each RGB channel. When the Method is set to Chroma, the blur and sharpness is adjusted based on Luma and Chroma controls.

Method

This menu is used to choose whether the node processes color using the Color or Chroma method. This also gives you a different set of control sliders.

Lock

This checkbox links the Softness and Detail sliders of each channel together.

Softness Red, Green, and Blue

The Softness sliders determine the amount of blur applied to each channel of the image. In Chroma mode, you have sliders for the softness in the Luminance and Chrominance channels, respectively.

Detail Red, Green, and Blue

The Sharpness sliders determine how much detail is reintroduced into each channel after each channel is softened. In Chroma mode, you have sliders for Luminance and Chrominance channels, respectively.
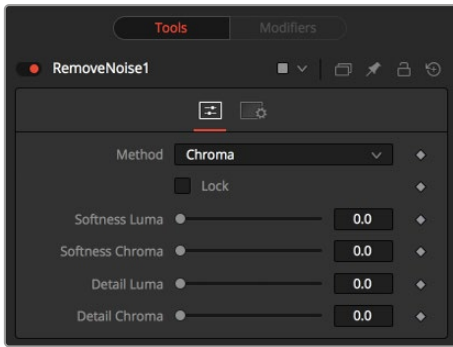
Chroma Method controls

Common Controls

Settings Tab

The Settings tab controls are common to all Film nodes, so their descriptions can be found in the following "The Common Controls" section.

The Common Controls

Film nodes share a number of identical controls in the Inspector. This section describes controls that are common among Film nodes.

Inspector
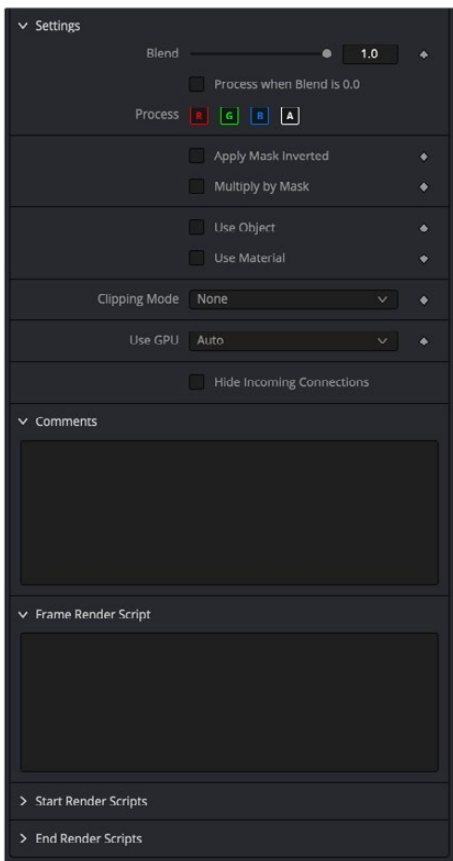
The Common Film settings inspector

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Film category. The Settings controls are even found on third-party Film-type plugin tools. The controls are consistent and work the same way for each tool, although some tools do include one or two individual options, which are also covered here.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Commonly, this causes the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if processing of this node is scripted to trigger another task, but the value of the node is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the red button on a Blur tool is deselected, the blur is first applied to the image, and then the red channel from the original input is copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option causes the RGB values of the masked image to be multiplied by the mask channel's values. This causes all pixels not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels are used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID is used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the view. The image or sequence must have been rendered from a 3D software package with those channels included.

Clipping Mode

This option determines how edges are handled when performing domain of definition rendering. This is mostly important for nodes like Blur, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that would normally be outside the upstream DoD is treated as black/transparent.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off hardware-accelerated rendering using the graphics card in your computer. Enabled uses the hardware. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Filter Nodes

This chapter details the Filter nodes available in Fusion.

> The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.
>
> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Create Bump Map [CBu]

The Create Bump Map node

Create Bump Map Node Introduction

The Create Bump Map node converts a grayscale (height map) image into a bump map. Unlike the Bump Map node that turns an image into a 3D material, the Create Bump Map node creates bump vector data and provides the output as an RGB image so other image-processing operations can be applied.

Input

The Create Bump Map node includes two inputs: one for the main image and the other for an effect mask to limit the area where the bump map is created.

— **Input:** The orange input takes the RGBA channels from an image to calculate the bump map.
— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the creation of the bump map to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.
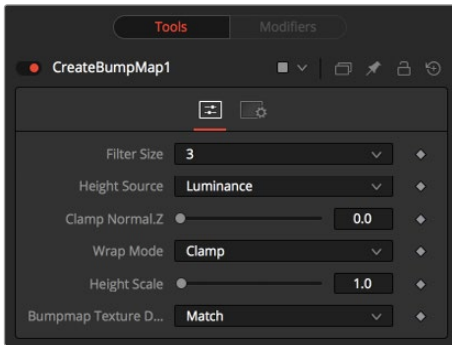
Basic Node Setup

The Create Bump Map node accepts a 2D grayscale image like a fast noise, which can then go through various 2D image-processing filters to create the bump map texture.

A Create Bump Map node produces a bump map as an RGB image for further image processing

Inspector

Create Bump Map controls

Controls Tab

The Controls tab contains all parameters for creating the bump map.

Filter Size

This menu sets the filter size for creating the bump map. You can set the filter size at 3 x 3 pixels or 5 x 5 pixels, thus determining the radius of the pixels sampled. The larger the size, the more time it takes to render.

Height Source

The Height Source menu selects the channel for extracting the grayscale information.

Clamp Normal.Z

This slider clips the lower values of the blue channel in the resulting bump texture.

Wrap Mode

This menu determines how the image wraps at the borders, so the filter produces a correct result when using seamless tiling textures.

Height Scale

The height scale menu modifies the contrast of the resulting values in the bump map. Increasing this value yields in a more visible bump map.

Bump Map Texture Depth

This menu matches or converts the resulting bump texture into the desired bit depth.

> **NOTE:** The below definitions are provided to clarify some of the terminology used in the Create Bump Map node and other similar types of nodes.
>
> — **Height Map:** A grayscale image containing a height value per pixel.
> — **Bump Map:** An image containing normals stored in the RGB channels used for modifying the existing normals (usually given in tangent space).
> — **Normal Map:** An image containing normals stored in the RGB channels used for replacing the existing normals (usually given in tangent or object space).

Common Controls

Settings Tab

The Settings tab controls are common to all Filter nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Custom Filter Node [CFlt]
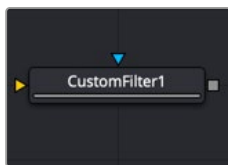
The Custom Filter node

Custom Filter Node Introduction

The Custom Filter node is used to apply custom convolution filters to images. A custom convolution filter can give a wide variety of image effects. For example, emboss, relief, sharpen, blurring, and edge detection are all convolution filters. There are many supplied custom filters in the Filters directory that can be loaded by right-clicking on the control header and selecting Settings > Load from the contextual menu.

The Custom filter uses an array (or grid) of either 3 x 3, 5 x 5, or 7 x 7 values. (Note: The array in the Inspector always shows a 7 x 7 grid; however, setting the Matrix Size to 3 x 3 uses only the center 9 cells.) The center of the array represents the current pixel, and entries nearby represent adjacent pixels. A value of 1 applies the full value of the pixel to the filter. A value of 0 ignores the pixel's value. A value greater than 1 multiplies the pixel's effect on the result. Negative values can also be entered, where the value of the pixel is subtracted from the average. Only integer values can be entered; 0.x is not valid.

Input

The Custom Filter node includes two inputs: one for the main image and the other for an effect mask to limit the area where the custom filter is applied.

— **Input:** The orange input takes the RGBA channels from an image to calculate the custom filter.
— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the custom filter to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Custom Filter node can be inserted after an image, mask, or any node that needs a custom convolution filter applied.
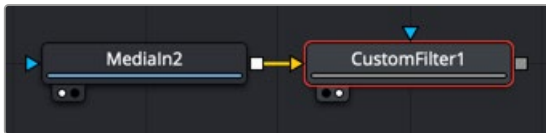
A Custom Filter node placed after a
MediaIn node in DaVinci Resolve

Inspector
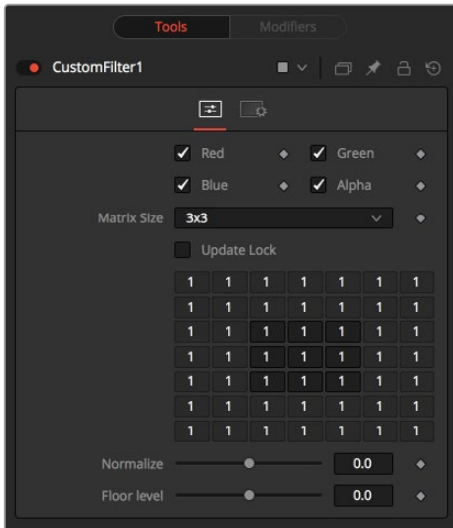
Custom Filter controls

Controls Tab

The Controls tab is used to set the filter size and then use the filter matrix to enter convolution filter values.

Color Channels (RGBA)

The custom filter defaults to operating on R, G, B, and A channels. Selective channel editing is possible by enabling or disabling the checkboxes beside each channel.

This is not the same as the RGBA checkboxes found under the Common Controls. The node takes these controls into account before it processes. Deselecting a channel causes the node to skip that channel when processing, speeding up the rendering of the effect. In contrast, these controls under the Common Controls tab are applied after the node has processed.

Matrix Size

This menu is used to set the size of the filter at 3 x 3 pixels, 5 x 5 pixels, or 7 x 7 pixels, thus setting the radius of the pixels sampled. The larger the size, the more time it takes to render.

Update Lock

When this control is selected, Fusion does not render the filter. This is useful for setting up each value of the filter, and then turning off Update Lock and rendering the filter.

Filter Matrix

The Filter Matrix control is a 7 x 7 grid of text boxes where a number is entered to represent how much influence each pixel has on the overall convolution filter. The text box in the center represents the pixel that is processed by the filter. The text box to the left of the center represents the pixel to the immediate left, and so forth.

The default Matrix size is 3 x 3. Only the pixels immediately adjacent to the current pixel are analyzed. If a larger Matrix size is set, more of the text boxes in the grid are enabled for input.

Normalize

This controls the amount of filter normalization that is applied to the result. Zero gives a normalized image. Positive values brighten or raise the level of the filter result. Negative values darken or lower the level.

Floor Level

This adds or subtracts a minimum, or Floor Level, to the result of the filtered image. Zero does not add anything to the image. Positive values add to the filtered image, and negative values subtract from the image.

Examples

Original Image Example

For example, a filter with the values...

```
0  0  0
0  1  0
0  0  0
```

...has zero effect from its neighboring pixels, and the resulting image would be unchanged.

Original image

Softening Example

A slight softening effect would be...

```
1  1  1
1  1  1
1  1  1
```

...where the neighboring pixels are averaged with the center.

Before and after averaging neighboring pixels to soften the image

Emboss Example

The example below subtracts five times the value from the top left and adds five times the value from the lower right.

–5 0 0

0 1 0

0 0 5

If parts of the processed image are very smooth in color, the neighboring values are very similar.

In parts of the image where the pixels are different (e.g., an edge), the results are different and tend to highlight or emboss edges in the image.

A Custom Filter adding and subtracting neighboring pixels to create an embossed image

Exposure Example

Using the values...

1 1 1

1 1 1

1 1 1

...and adjusting Normalize to a positive value makes the image brighter or glow, simulating film overexposure.

The Custom Filter Normalize slider used to change exposure

Relief Example

Using the values...

−1 0 0

0 0 0

0 0 1

... and adjusting Floor Level to a positive value creates a Relief filter.

Custom Filter Floor Level slider used to create a relief

Common Controls

Settings Tab

The Settings tab controls are common to all Filter nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Erode Dilate Node [ErDl]

The Erode Dilate node

Erode Dilate Node Introduction

The Erode Dilate node contracts or expands the image, depending on whether the Amount slider is set to a negative or positive value.

Inputs

The Erode Dilate node includes two inputs: one for the main image and the other for an effect mask to limit the area where the erode or dilate is applied.

— **Input:** The orange input takes the RGBA channels from an image to calculate the custom filter.
— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the erode or dilate to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Erode Dilate node is commonly used to contract or expand mattes. Below, a Luma Keyer is connected to the Erode Dilate and passes the modified key to a Matte Control where it is embedded into the image.
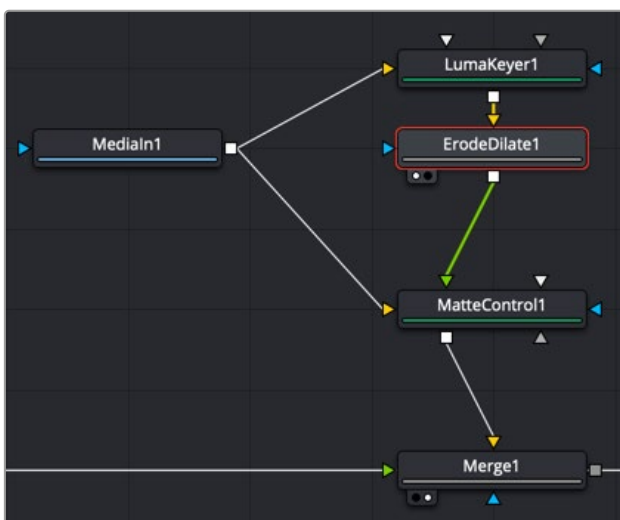
An Erode Dilate node placed after a Luma
Keyer to operate on a Matte Control

Inspector
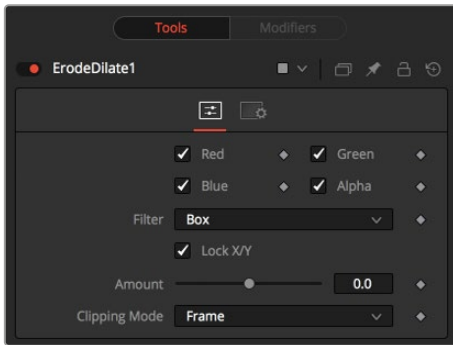
Erode Dilate controls

Controls Tab

The Controls tab includes the main Amount slider that determines whether you are performing an erode by entering a negative value or a dilate by entering a positive value.

Color Channels (RGBA)

The Erode Dilate node defaults to operating on R, G, B, and A channels. Selective channel editing is possible by enabling or disabling the checkboxes beside each channel.

This is not the same as the RGBA checkboxes found under the Common Controls. The node takes these controls into account before it processes. Deselecting a channel causes the node to skip that channel when processing, speeding up the rendering of the effect. In contrast, the channel controls under the Common Controls tab are applied after the node has processed.

Lock X/Y

The Lock X/Y checkbox is used to separate the Amount slider into amount X and amount Y, allowing a different value for the effect on each axis.

Amount

A negative value for Amount causes the image to erode. Eroding simulates the effect of an underexposed frame, shrinking the image by growing darker areas of the image so that they eat away at brighter regions.

A positive value for Amount causes the image to dilate, similar to the effect of overexposing a camera. Regions of high luminance and brightness grow, eating away at the darker regions of the image. Both techniques eradicate fine detail in the image and tend to posterize fine gradients.

The Amount slider scale is based on the input image width. An amount value of 1 = image width. So, if you want to erode or dilate by exactly 1 pixel on an HD image, you would enter 1/1920, or 0.00052083.

Common Controls

Settings Tab

The Settings tab controls are common to all Filter nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Filter Node [Fltr]

The Filter node

Filter Node Introduction

The Filter node contains several standard convolution filters, easily selectable from a list. This node enables a variety of effects, from radically changing the look of an image to adding subtle randomly-generated film grain. The Sobel and Laplacian settings are often used for edge detection.

Inputs

The Filter node includes two inputs: one for the main image and the other for an effect mask to limit the area where the filter is applied.

— **Input:** The orange input is used for the primary 2D image that gets the filter applied.
— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the filter to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Filter node can be inserted after an image, mask, or any node that needs a filter applied. Below, it is used to create an edge matte, which is then used to mask the soft glow around the keyed foreground.
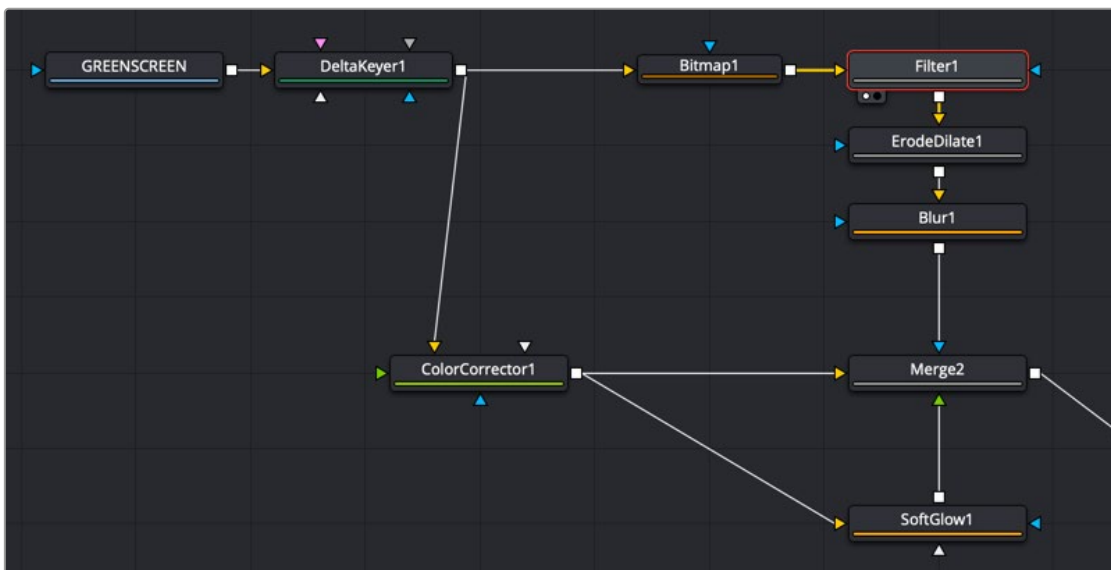
A Filter node using the Sobel setting to extract an edge matte

Inspector

Filter controls

Controls Tab

The Controls tab is used to set the filter type, the channels the filter is applied to, and the amount it blends with the original image.

Filter Type

The Filter Type menu provides a selection of filter types described below.

— **Relief:** This appears to press the image into metal, such as an image on a coin. The image appears to be bumped and overlaid on gray.

— **Emboss Over:** Embosses the image over the top of itself, with adjustable highlight and shadow height and direction.

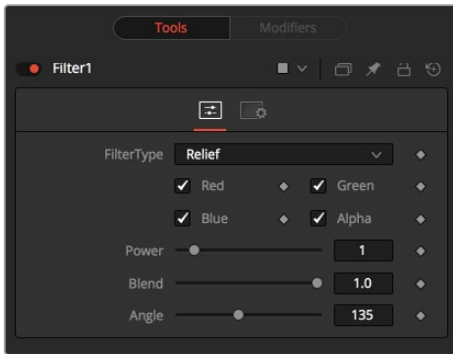— **Noise:** Uniformly adds noise to images. This is often useful for 3D computer-generated images that need to be composited with live action, as it reduces the squeaky-clean look that is inherent in rendered images. The frame number acts as the random generator seed. Therefore, the effect is different on each frame and is repeatable.

— **Defocus:** This filter type blurs the image.

— **Sobel:** Sobel is an advanced edge detection filter. Used in conjunction with a Glow filter, it creates impressive neon light effects from live-action or 3D-rendered images.

— **Laplacian:** Laplacian is a very sensitive edge detection filter that produces a finer edge than the Sobel filter.

— **Grain:** Adds noise to images similar to the grain of film (mostly in the midrange). This is useful for 3D computer-generated images that need to be composited with live action as it reduces the squeaky-clean look that is inherent in rendered images. The frame number acts as the random generator seed. Therefore, the effect is different on each frame and is repeatable.

Color Channels (RGBA)

The Filter node defaults to operating on R, G, B, and A channels. Selective channel editing is possible by enabling or disabling the checkboxes beside each channel.

Power

Values range from 1 to 10. Power proportionately increases the amount by which the selected filter affects the image. This does not apply to the Sobel or Laplacian filter type.

Angle

This control has a range from 0 to 315 degrees and changes the effect in increments of 45 degrees. This applies only to the Relief and Emboss filters.

Median

Depending on which Filter Type is selected, the Median control may appear. It varies the Median filter's effect. A value of 0.5 produces the true median result, as it finds the middle values. A value of 0.0 finds the minimums, and 1.0 finds the maximums. This applies to the Median setting only.

Seed

This control is visible only when applying the Grain or Noise filter types. The Seed slider can be used to ensure that the random elements of the effect are seeded with a consistent value. The randomizer always produces the same result, given the same seed value.

Animated

This control is visible only when applying the Grain or Noise filter types. Select the checkbox to cause the noise or grain to change from frame to frame. To produce static noise, deselect this checkbox.

Common Controls

Settings Tab

The Settings tab controls are common to all Filter nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Rank Filter Node [RFlt]

The Rank Filter node

Rank Filter Node Introduction

The Rank Filter examines nearby pixels, sorts the pixels by value, and then replaces the color of the examined pixels with the color of the pixel with the selected rank.

Inputs

The Rank Filter node includes two inputs: one for the main image and the other for an effect mask to limit the area where the filter is applied.

— **Input:** The orange input is used for the primary 2D image that gets the Rank filter applied.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the rank filter to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Rank Filter node can be placed anywhere in a node tree to apply an effect to an image.
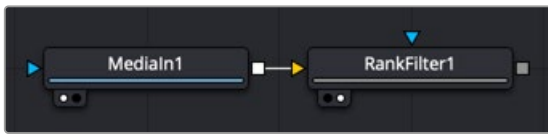
A Rank Filter node placed after a MediaIn node in DaVinci Resolve

Inspector
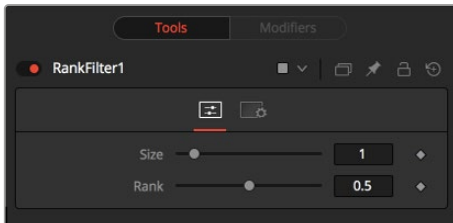
Rank Filter controls

Controls Tab

The Controls tab is used to set the size and rank value of the filter.

Size

This control determines the size in pixels of the area sampled by the filter. A value of 1 samples 1 pixel in each direction, adjacent to the center pixel. This produces a total of 9 pixels, including the center sampled pixel. Larger values sample from a larger area.

Low Size settings are excellent for removing salt and pepper style noise, while larger Size settings produce an effect similar to watercolor paintings.

Rank

The Rank slider determines which value from the sampled pixels is chosen. A value of 0 is the lowest value (darkest pixel), and 1 is the highest value (brightest pixel).

Example

Below is a before and after example of a Rank filter with Size set to 7 and a Rank of 0.7 to create a watercolor effect.

Before and after of a Rank Filter producing a watercolor-style effect

The Common Controls

Filter nodes share a number of identical controls in the Inspector. This section describes controls that are common among Filter nodes.

Inspector
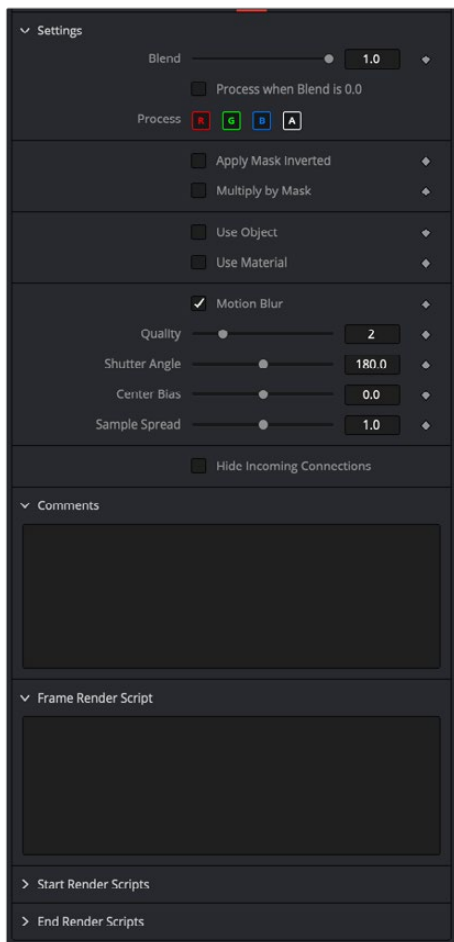
Common Filter settings Inspector

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Filter category. The Settings controls are even found on third-party filter-type plugin tools. The controls are consistent and work the same way for each tool, although some tools do include one or two individual options, which are also covered here.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this causes the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if processing of this node is scripted to trigger another task, but the value of the node is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the red button on a Blur tool is deselected, the blur is first applied to the image, and then the red channel from the original input is copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option causes the RGB values of the masked image to be multiplied by the mask channel's values. This causes all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object ID and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels are used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID is used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the view. The image or sequence must have been rendered from a 3D software package with those channels included.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off hardware-accelerated rendering using the graphics card in your computer. Enabled uses the hardware. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 causes Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one whole frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows for the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Flow Nodes

This chapter details the Sticky Note and Underlay features available in Fusion.

> The abbreviations next to each feature name can be used in the Select Tool dialog when searching for tools and in scripting references.
>
> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Sticky Note [Nte]

The Sticky Note

Sticky Note Introduction

A Sticky Note is not a node at all. It is a useful way of attaching notes, comments, and history to a specific area of a comp. By changing their size and color, they can provide unobtrusive comments or important notices, as required. Sticky Notes make an excellent complement to the Comments tab in the Inspector.

Usage
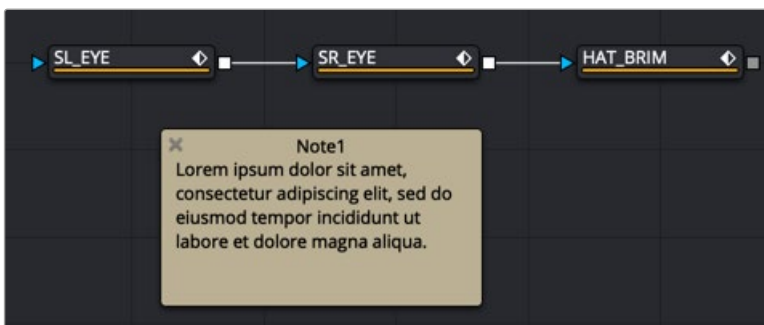
A Sticky Note added to a node tree can provide a description for other people or as a reminder to yourself

To create a Sticky Note, click in an empty area of the Node Editor where you want a Sticky Note to appear. Then, from the Effects Library, click the Sticky Note effect located in the Tools > Flow category or press Shift-Spacebar and search for the Sticky Note in the Select Tool window.

Like Groups, Sticky Notes are created in a smaller, collapsed form. They can be expanded by double-clicking on them. Once expanded, they can be resized using any side or corner of the note or moved by dragging on the name header. To collapse the Sticky Note again, click the icon in the top-left corner.

To rename, delete, copy, or change the color of the note, right-click over the note and choose from the contextual menu. Using this menu, you can also lock the note to prevent editing.

To edit the text in a Sticky Note, first expand it by double-clicking anywhere on the note, and then click below its title bar. If the note is not locked, you can edit the text.

Underlay [Und]

The Underlay

Underlay Introduction

Underlays are a convenient method of visually organizing areas of a composition. As with Groups, Underlays can improve the readability of a comp by separating it into labeled functional blocks. While Groups are designed to streamline the look of a comp by collapsing complex layers down to single nodes, Underlays highlight, rather than hide, and do not restrict outside connections.

Usage
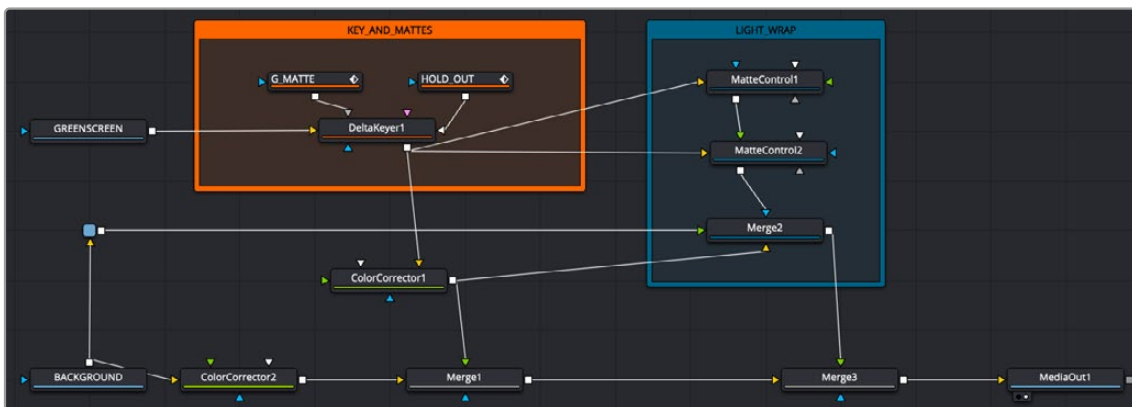
Underlay node structure

As with Sticky Notes , an Underlay can be added to a comp by selecting it from the Flow category in the Effects Library or searching for it in the Select Tool window. The Underlay to the Node Editor with its title bar is centered on the last-clicked position.

Underlays can be resized using any side or corner. This will not affect any nodes.

Underlays can also be used as simple selection groups. Activating an Underlay, by clicking its title, will select all the tools contained wholly within it as well, allowing the entire set to be moved, duplicated, passed through, and so on.

To rename an Underlay, first ensure that nodes contained within the Underlay are not selected. Then, Option-click on the Underlay title to select the Underlay without selecting the nodes it contains. Once selected, right-click over the title and choose Rename. Underlays can be assigned a color using the same right-click contextual menu.

Flow Organizational Nodes

This chapter details the Groups, Macro, and Pipe Router nodes, which are designed to help organize your compositions, making the node tree easier to see and understand.

> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Groups

The Group node

Groups Introduction

Groups are used to keep complex node trees organized. You can select any number of nodes in the node tree and then group them to create a single node icon in the Node Editor. Groups are non-destructive and can be opened at any time.

Usage

— To group nodes, select them in the Node Editor, and then right-click over any of the selected nodes and choose Group from the contextual menu.

Selecting nodes to group

— To edit the individual nodes in a group, right-click and choose Expand Group from the contextual menu. All individual nodes contained in the group are displayed in a floating node tree window. When opened, groups hover over existing elements, allowing editing of the enclosed nodes.
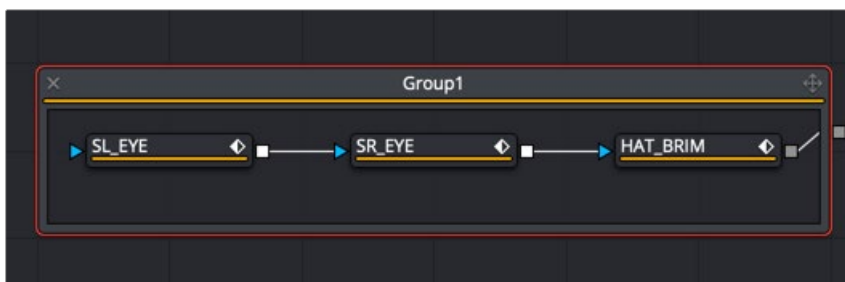
An opened node group

— To remove or decompose a group and retain the individual nodes, right-click the group and choose Ungroup.

Macro

Macros are not technically a node. Instead, they are a group of nodes that act as a single node.

Macro Introduction

Macros can be used to combine multiple nodes and expose a user-definable set of controls.

They are meant as a fast and convenient way of building custom nodes.

Usage

To create a Macro, select the nodes intended for the macro. The order in which the nodes are selected becomes the order in which they are displayed in the Macro Editor. Right-click on any of the selected nodes and choose Macro > Create Macro from the contextual menu.
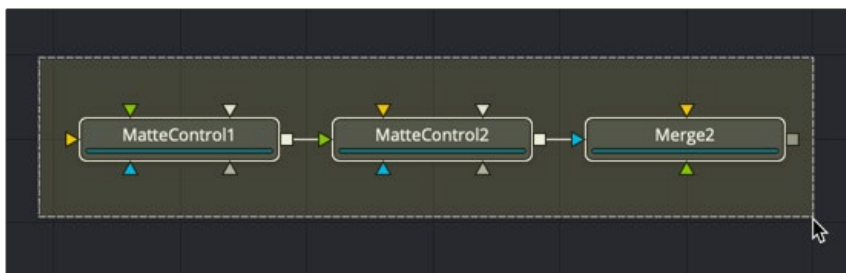
Select the nodes to include in the macro

Macro Editor

The Macro Editor allows you to specify and rename the controls that are exposed in the final macro tool.

In the example below, the tool is named Light_Wrap at the top. The Blur slider for Matte Control 1 is enabled and renamed to Softness, as it will appear in the Inspector.
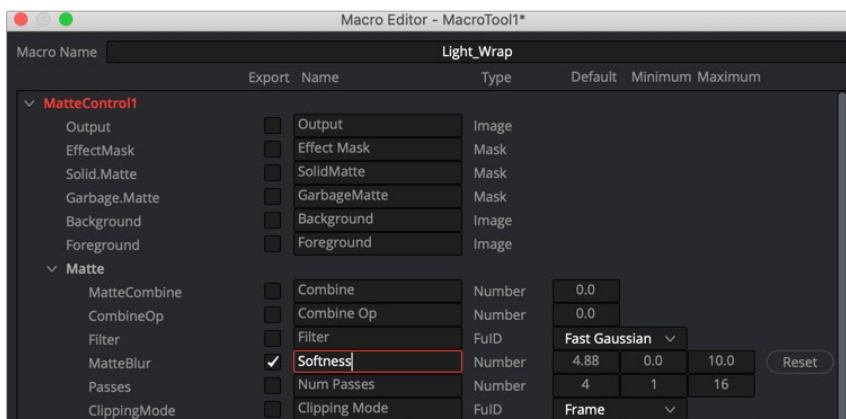
The Macro Editor

After setting up the macro to your liking, click the Close button in the lower-right corner of the dialog. Then, in the Save dialog, click Yes to save the macro, click No to leave Macro Editor without saving the changes, or click Cancel to return to the Macro Editor.
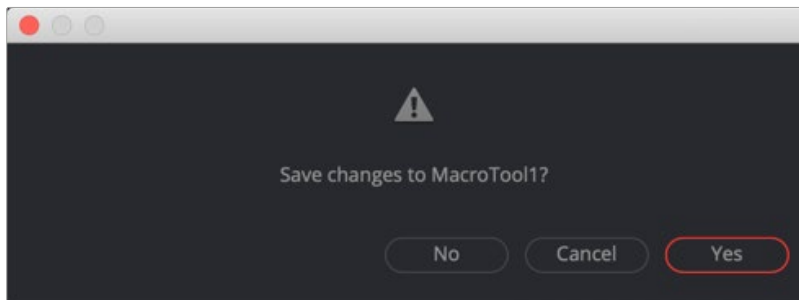
The Macro Editor save dialog

To add the macro to your node tree, right-click anywhere on the node tree and select Macro > [NameOfYourMacro] from the contextual menu.

Saving a Macro as a Title Template in the Edit Page (DaVinci Resolve)

When using DaVinci Resolve, macros are available only in the Fusion page. However, if the macro is a title animation, you can save it to the Titles Templates folder and have it appear in the Edit page Effects Library.

To save a title macro so it appears in the Edit page Effects Library, save the macro to:

— **macOS:** Users > UserName > Library > Application Support > Blackmagic Design > DaVinci Resolve > Fusion > Templates > Edit > Titles

— **Windows:** C Drive > Users > UserName > AppData > Roaming > Blackmagic Design > DaVinci Resolve > Support > Fusion > Templates > Edit > Titles

The Final Macro

The final macro looks and behaves just like any other node in Fusion.

As another example, you could take a single Channel Boolean, set it to Add mode, and make it into a macro exposing no controls at all, thus creating the equivalent of an Add Mix node like the one that can be found in programs like Nuke.

Pipe Router

Pipe Routers are another type of organizational tool you use to improve the layout and appearance of the node tree.

Router Introduction

Routers can be used to neatly organize your comps by creating "elbows" in your node tree, so the connection lines do not overlap nodes, making them easier to understand. Routers do not have any influence on render times.

Usage
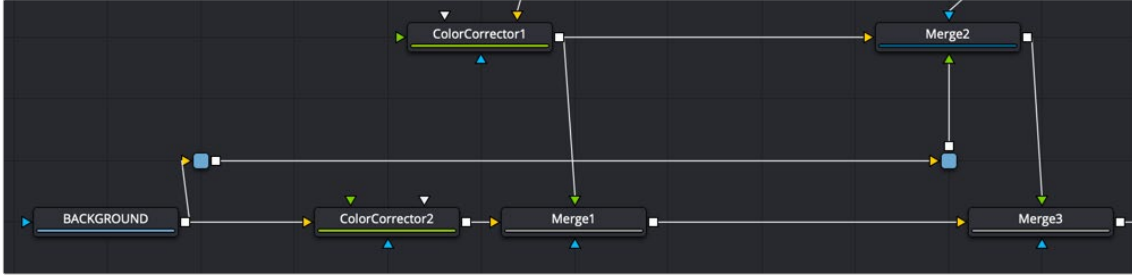
A Pipe Router usage example

Router

To insert a router along a connection line, Option- or Alt-click on the line. The router can then be repositioned to arrange the connections as needed.

Although routers have no actual controls, they still can be used to add comments to a comp.
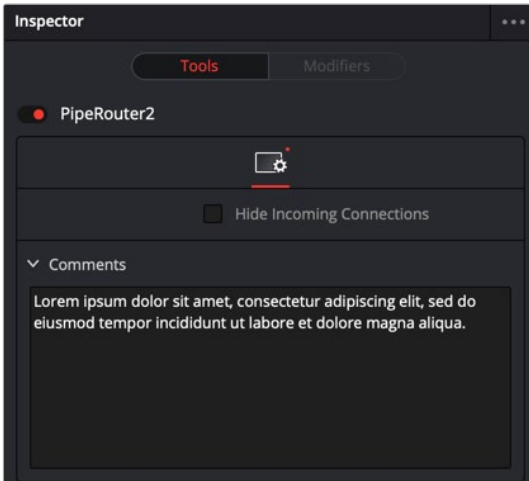
An example comment in a Router node

Fuses

This chapter introduces Fuses, which are scriptable plugins that can be used within Fusion.

Contents

Fuses [Fus]

A Fuse node

Fuses Introduction

Fuses are plugins. The difference between a Fuse and an Open FX plugin is that a Fuse is created using a Lua script. Fuses can be edited within Fusion or DaVinci Resolve, and the changes you make compile on-the-fly.

Using a Lua script makes it easy for even non-programmers to prototype and develop custom nodes. A new Fuse can be added to a composition, edited and reloaded, all without having to close the current composition. They can also be used as modifiers to manipulate parameters, curves, and text very quickly. ViewShader Fuses can make use of the GPU for faster performance. This makes Fuses much more convenient than an Open FX plugin that uses Fusion's OFX SDK. However, this flexibility comes at a cost. Since a Fuse is compiled on-the-fly, it can be significantly slower than the identical node created using the Open FX SDK.

As an example, Fuses could generate a mask from the over-exposed areas of an image, or create initial particle positions based on the XYZ position stored within a text file.

Please contact Blackmagic Design for access to the SDK (Software Developer Kit) documentation.

Installing Fuses

Fuses are installed in the Fusion:\Fuses path map. By default this folder is located at Users/ User_Name/Library Application Support/Blackmagic Design/Fusion (or DaVinci Resolve)/Fuses on macOS or C:\Users\User_Name\AppData\Roaming\Blackmagic Design\Fusion (or DaVinci Resolve)\Fuses, on Windows. Files must use the extension .fuse, or they will be ignored by Fusion.

Working with Fuses in a Composition

Fuses can be designed to appear in any category in the Effects Library. Once installed, they are added to a composition exactly as any native or third-party plugin node. However, since a Fuse is just a text document, it can be edited by clicking the Edit button that appears at the top of the Inspector when the Fuse node is selected. This will open the Fuse in the default script editor specified in the Global Preferences/Scripting panel.

> **NOTE:** Any changes made to a Fuse's script do not immediately affect other copies of the same Fuse node already added to a composition. To use the updated Fuse script on all similar Fuses in the composition, either close and reopen the composition, or click on the Reload button in each Fuse's Inspector.

When a composition containing a Fuse node is opened, the currently saved version of the Fuse script is used. The easiest way to ensure that a composition is running the current version of a Fuse is to close and reopen the composition.

Generator Nodes

This chapter details the Generator nodes available in Fusion.

> The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.
>
> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Background [Bg]

The Background node

Background Node Introduction

The Background node can be used to produce anything from a solid color background to complex looped gradients.

Inputs

There is one input on the Background node for an effect mask input.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the background color to only those pixels within the mask.

Basic Node Setup

The Background node generates a color and passes the color off to another node like the Paint node below.

A Background node generating a background for the Paint tool

Inspector
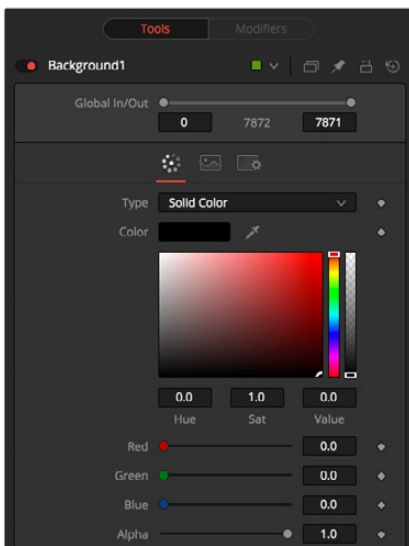
The Background node controls

Color Tab

The Color tab is used to control the style and color(s) of the generated background.

Type

This control is used to select the style of background generated by the node. Four selections are available:

— **Solid Color:** This default creates a single-color image.

— **Horizontal:** This creates a two-color horizontal gradation.

— **Vertical:** This creates a two-color vertical gradation.

— **Four Corner:** This creates a four-color corner gradation.
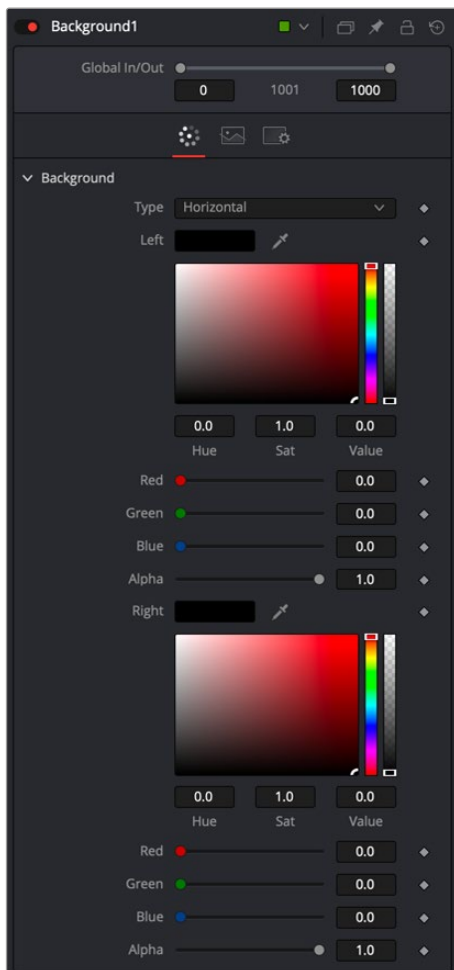
Horizontal gradient controls

Horizontal/Vertical/Four Point

When the Type menu is set to Horizontal, Vertical, or Four Corner, two- or four-color swatches are displayed where the left/right, top/bottom, or four corners of the gradient colors can be set.

Gradient controls

Gradient

When the Type menu is set to Gradient, additional controls are displayed where the gradient colors' direction can be customized.

Gradient Type

This menu selects the form used to draw the gradient. There are six choices:

— **Linear:** Draws the gradient along a straight line from the starting color stop to the ending color stop.

— **Reflect:** Draws the gradient by mirroring the linear gradient on either side of the starting point.

— **Square:** Draws the gradient by using a square pattern when the starting point is at the center of the image.

— **Cross:** Draws the gradient using a cross pattern when the starting point is at the center of the image.

— **Radial:** Draws the gradient in a circular pattern when the starting point is at the center of the image.

— **Angle:** Draws the gradient in a counterclockwise sweep when the starting point is at the center of the image.

Viewer Start and End Position

The Start and End positions in the viewer are represented by two red control points connected by a green line. They determine where the gradient begins and ends.

Gradient Colors

This gradient color bar is used to select the colors for the gradient. The default two color stops set the start and end colors. You can change the colors used in the gradient by selecting the color stop, and then using the Eyedropper or color swatch to set a new color.

You can add, move, copy, and delete color from the gradient using the gradient bar.

To modify one of the colors, select the triangle below the color on the bar.

To add a color stop to the gradient bar:

1   Click anywhere along the bottom of the gradient bar.

2   Use the Eyedropper or color swatch to select the color for the color stop.

To move a color stop on the gradient bar:

—   Drag a color stop left or right along the bar.

To copy a color stop:

—   Hold Command (macOS) or Ctrl (Windows) while you drag a color stop.

To delete a color stop:

—   Drag the color stop up past the gradient bar.

Interpolation Space

This menu determines what color space is used to calculate the colors between color stops.

Offset

The Offset control is used to offset the position of the gradient relative to the start and end markers. This control is most useful when used in conjunction with the repeat and ping-pong modes described below.

Repeat

This menu includes three options used to set the behavior of the gradient when the Offset control scrolls the gradient past its start and end positions. Selecting Once keeps the color continuous for offset. Selecting Repeat loops around to the start color when the offset goes beyond the end color. Selecting Ping-pong repeats the color pattern in reverse.

Sub-Pixel

The Sub-Pixel menu controls the sub-pixel precision used when the edges of the gradient become visible in repeat mode, or when the gradient is animated. Higher settings will take significantly longer to render but are more precise.

Gradient Contextual Menu

Gradients have their own contextual menu that you can bring up by right-clicking on the gradient bar. The Gradient contextual menu includes options for animating, publishing, and connecting one gradient to another. There is also a gradient-specific modifier that builds a custom gradient by sampling colors from the output of a node.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are duplicated in many Generator nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Day Sky [DS]
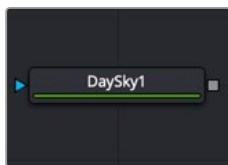
The Day Sky node

Day Sky Node Introduction

The Day Sky node aims to produce a simulation of the daylight produced at a specific time and location on the earth, and generates a high dynamic range image that represents a map of that light. It is not a sky generator, although it could be combined with a cloud generator or Noise node to produce one.

> **NOTE:** This generator is a practical implementation of the research paper, *A Practical Analytical Model for Daylight*, by Preetham, Shirley, and Smits. A copy of the original paper can be found at the website for the Visual Simulation Group at the University of Utah [https://www.cs.utah.edu/~shirley/papers/sunsky/sunsky.pdf].

Inputs

There is a single input on the Day Sky node for an effect mask to limit the area where the day sky simulation occurs is applied.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Day Sky to only those pixels within the mask.

Basic Node Setup

The Day Sky node is a generator, so it typically starts the branch of a node tree and connects to some other node, like a Merge node.
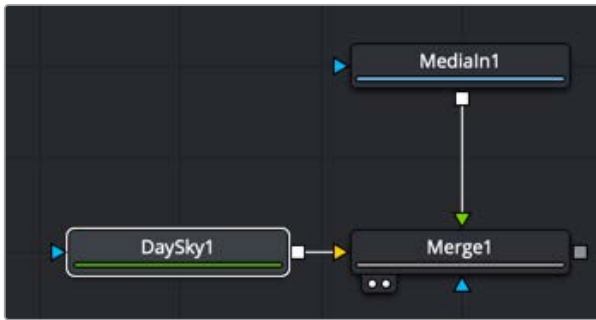
A Day Sky connected as the background to a Merge node

Inspector
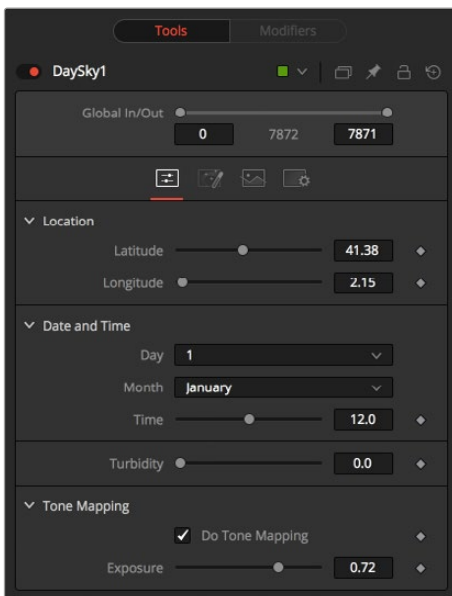
Day Sky node controls

Controls Tab

The Controls tab is used to set the location and time of the daylight simulation. This will determine the overall look that is generated.

Location

The Latitude and Longitude sliders are used to specify the location used to create the Day Sky simulation.

Date and Time

The Day, Month, and Time controls are used to select the specific time for the Day Sky simulation.

Turbidity

Turbidity causes light to be scattered and absorbed instead of transmitted in straight lines through the simulation. Increasing the turbidity will give the sky simulation a murky feeling, as if smoke or atmospheric haze were present.

Do Tone Mapping

Since the simulation is calculated in 32-bit floating-point color space, it generates color values well above 1.0 and well below 0.0. Tone mapping is a process that takes the full dynamic range of the resulting simulation and compresses the data into the desired exposure range while attempting to preserve as much detail from the highlights and shadows as possible. Deselect this checkbox to disable any tone mapping applied to the simulation.

Generally, this option should be deselected only if the resulting image will later be color corrected as part of a floating-point color pipeline.

Exposure

Use this control to select the exposure used for tone mapping.
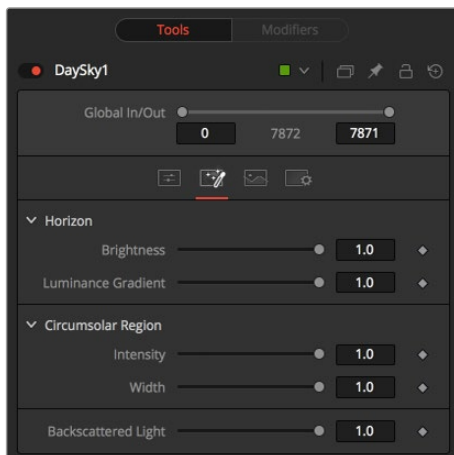
Day sky node controls

Advanced Tab

The Advanced tad provides more specific controls over the brightness and width of the different ranges in the generated sky.

Horizon Brightness

Use this control to adjust the brightness of the horizon relative to the sky.

Luminance Gradient

Use this control to adjust the width of the gradient separating the horizon from the sky.

Circumsolar Region Intensity

Use this control to adjust the intensity or brightness of the sky nearest to the sun.

Circumsolar Region Width

Use this control to adjust the width or size of the area in the sky affected by the sun.

Backscattered Light

Use this control to increase or decrease the backscatter light in the simulation.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are duplicated in many Generator nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Fast Noise [FN]

The Fast Noise node

Fast Noise Node Introduction

The Fast Noise node is a very fast and flexible Perlin Noise generator. It can be useful for a wide range of effects, from clouds, to swirling fog, waves, water caustics, stylized fire, and smoke, and other organic textures. It is also invaluable as a noise source for other effects, such as heat shimmers, particle systems, and dirtiness maps.

Inputs

The two map inputs on the Fast Noise node allow you to use masks to control the value of the noise detail and brightness controls for each pixel. These two optional inputs can allow some interesting and creative effects. There is also a standard effect mask input for limiting the Fast Noise size.

— **Noise Detail Map:** A soft-edged mask connected to the gray Noise Detail Map input will give a flat noise map (zero detail) where the mask is black, and full detail where it is white, with intermediate values smoothly reducing in detail. It is applied before any gradient color mapping. This can be very helpful for applying maximum noise detail in a specific area, while smoothly falling off elsewhere.

— **Noise Brightness Map:** A mask connected to this white input can be used to control the noise map completely, such as boosting it in certain areas, combining it with other textures, or if Detail is set to 0, replacing the Perlin Noise map altogether.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Fast Noise to only those pixels within the mask.

Basic Node Setup

The Fast Noise node is used to generate images for other nodes to take advantage of. For example, below the Fast Noise node is used as a bitmap source in the Particle Emitter.
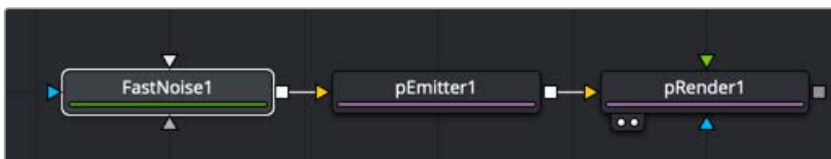
A Fast Noise node used as a bitmap source for a Particle Emitter

Inspector
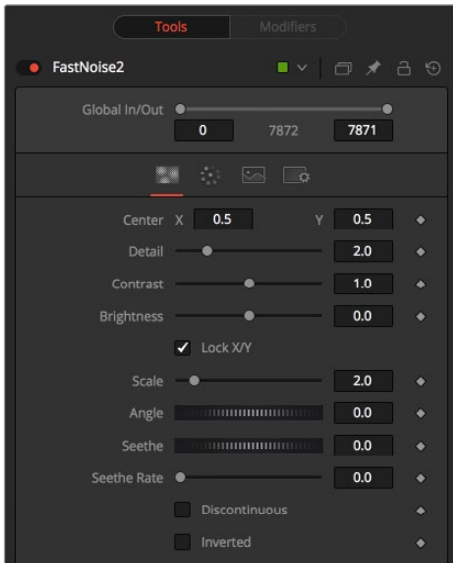
Fast Noise controls

Noise Tab

The Noise tab controls the shape and pattern of the noise for the Fast Noise node.

Discontinuous

Normally, the noise function interpolates between values to create a smooth continuous gradient of results. Enable this checkbox to create hard discontinuity lines along some of the noise contours. The result will be a dramatically different effect.

Inverted

Select this checkbox to invert the noise, creating a negative image of the original pattern. This is most effective when Discontinuous is also enabled.

Center

Use the Center coordinate control to pan and move the noise pattern.

Detail

Increase the value of this slider to produce a greater level of detail in the noise result. Larger values add more layers of increasingly detailed noise without affecting the overall pattern. High values take longer to render but can produce a more natural result.

Brightness

This control adjusts the overall brightness of the noise map, before any gradient color mapping is applied. In Gradient mode, this has a similar effect to the Offset control.

Contrast

This control increases or decreases the overall contrast of the noise map, prior to any gradient color mapping. It can exaggerate the effect of the noise and widen the range of colors applied in Gradient mode.

Lock and Scale X/Y

The size of the noise map can be adjusted using the Scale slider, changing it from gentle variations over the whole image to a tighter overall texture effect. The Scale slider can be separated into independent X- and Y-axis scale sliders by clicking on the Lock X/Y checkbox immediately above, which can be useful for a brushed-metal effect.

Angle

Use the Angle control to rotate the noise pattern.

Seethe

Adjust this thumbwheel control to interpolate the noise map against a different noise map. This will cause a crawling shift in the noise, as if it was drifting or flowing. This control must be animated to affect the gradient over time, or you can use the Seethe Rate control below.

Seethe Rate

As with the Seethe control above, the Seethe Rate also causes the noise map to evolve and change. The Seethe Rate defines the rate at which the noise changes each frame, causing an animated drift in the noise automatically, without the need for spline animation.
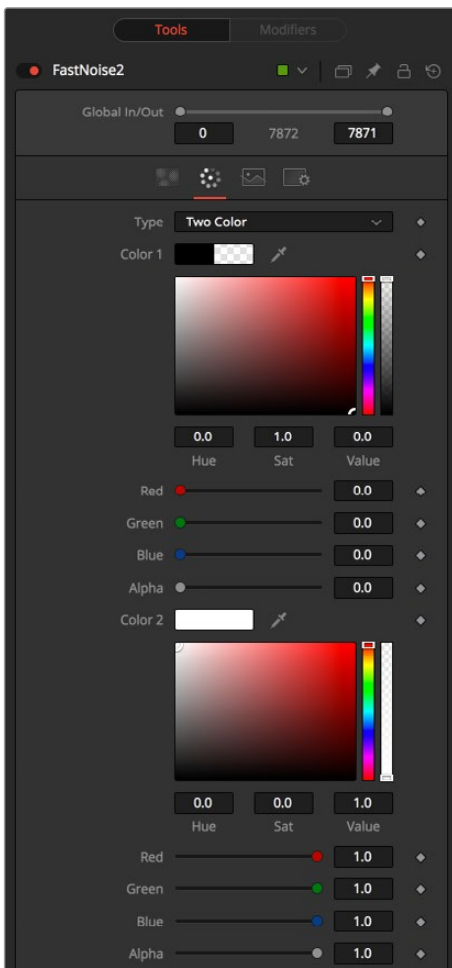
Fast Noise node Color tab

Color Tab

The Color tab allows you to adjust the gradient colors used in the generated noise pattern.

Two Color

A simple two-color gradient is used to color the noise map. The noise function will smoothly transition from the first color into the second.

Gradient

The Advanced Gradient control in Fusion is used to provide more control over the color gradient used with the noise map.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are duplicated in many Generator nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Mandelbrot [Man]

The Mandelbrot node

Mandelbrot Node Introduction

This node creates an image pattern based on the Mandelbrot fractal theory set.

Inputs

The one input on the Mandelbrot node is for an effect mask to limit the area where the fractal noise is applied.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the fractals to only those pixels within the mask.

Basic Node Setup

The Mandelbrot node generates images that can be used for motion graphics and sci-fi effects. Below, it is used to generate motion graphics for the background of a title.
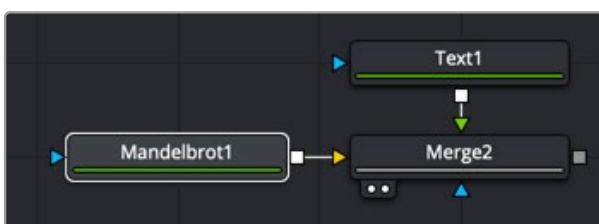
A Mandelbrot node applied as a background to the Merge node

Inspector
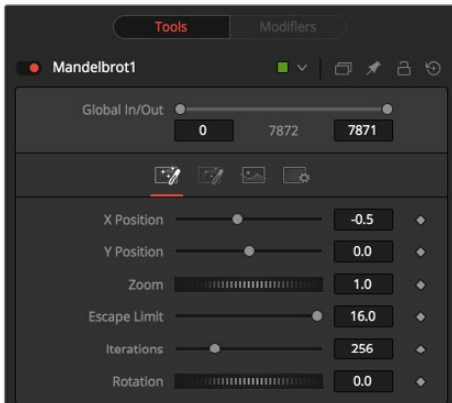
Mandelbrot node controls

Noise Tab

The Noise tab controls the shape and pattern of the noise for the Mandelbrot node.

Position X and Y

This chooses the image's horizontal and vertical position or seed point.

Zoom

Zoom magnifies the pattern in or out. Every magnification is recalculated so that there is no practical limit to the zoom.

Escape Limit

Defines a point where the calculation of the iteration is aborted. Low values lead to blurry halos.

Iterations

This determines the repetitiveness of the set. When animated, it simulates a growing of the set.

Rotation

This rotates the pattern. Every new angle requires recalculation of the image.
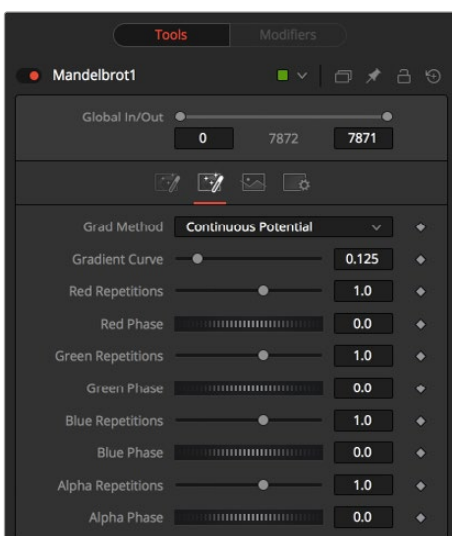
The Mandelbrot node Color tab

Color Tab

The Color tab allows you to adjust the gradient and repetition of the gradient colors for the generated pattern.

Grad Method

Use this control to determine the type of gradation applied at the borders of the pattern.

Continuous Potential

This causes the edges of the pattern to blend to the background color.

Iterations

This causes the edges of the pattern to be solid.

Gradient Curve

This affects the width of the gradation from the pattern to the background color.

R/G/B/A Phase and Repetitions

These controls set the color values of the pattern generators.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are duplicated in other generator nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Plasma [Plas]
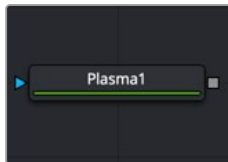
The Plasma node

Plasma Node Introduction

The Plasma node is a background generation node that uses four circular patterns to generate images similar to plasma. It is useful as a deforming pattern for the Shadow and Deform nodes and to create a variety of other useful shapes and patterns. It is similar to the Fast Noise node.

Inputs

The one input on the Plasma node is for an effect mask to limit the area where the plasma pattern is applied.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the plasma to only those pixels within the mask.

Basic Node Setup

Like the Mandelbrot node, the Plasma node generates images that can be used for motion graphics and sci-fi effects. Below, it is used to generate motion graphics for the background of a title.
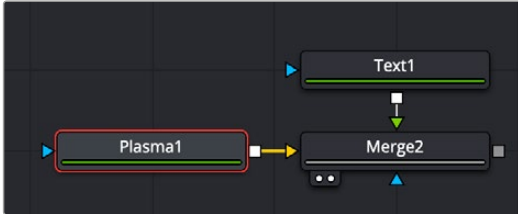
A Plasma node applied as a background to a Merge node

Inspector
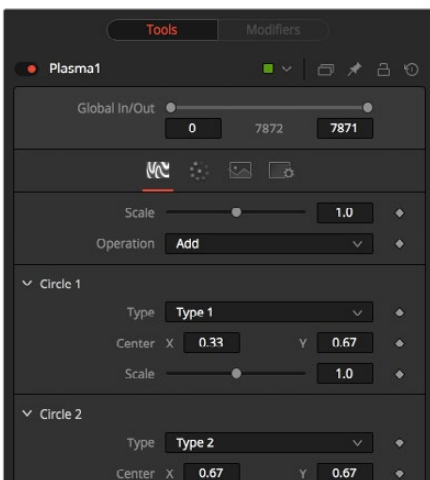
Plasma node Circles tab

Circles Tab

The Circles tab controls the shape and pattern generated by the Plasma node.

Scale

The Scale control is used to adjust the size of the pattern created.

Operation

The options in this menu determine the mathematical relationship among the four circles whenever they intersect.

Circle Type

Select the type of circle to be used.

Circle Center

Report and change the position of the circle center.

Circle Scale

Determines the size of the circle to be used for the pattern.

Color Tab

The Color tab allows you to adjust the colors and location within the pattern of the colors for the generated plasma.
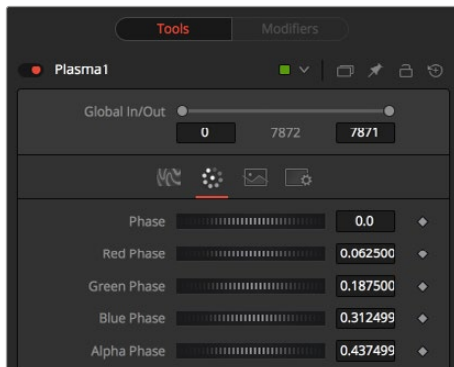
Plasma node Color tab

Phase

Phase changes the color phase of the entire image. When animated, this creates psychedelic color cycles.

R/G/B/A Phases

Changes the phase of the individual color channels and the Alpha. When animated, this creates color cycling effects.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are duplicated in many Generator nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Text+ [Txt+]
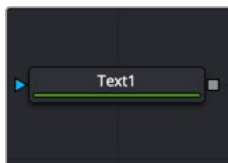
The Text+ node

Text+ Node Introduction

Fusion's Text+ node is an advanced character generator capable of multiple styles, 3D transformations, and several layers of shading. Text can be laid out to a user-defined frame, circle, or along a path.

Any TrueType, OpenType, or PostScript 1 font installed on the computer can be used to create text. Support for multibyte and Unicode characters allows text generation in any language, including right to left and vertically oriented text.

This node generates a 2D image. To produce extruded 3D text with optional beveling, see the Text 3D node.

Inputs

The one input on the Text+ node is for an effect mask to crop the text.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the text to only those pixels within the mask.

Basic Node Setup

The Text+ node can be combined with many other nodes to create motion graphics. Below it is connected as the foreground of a Merge node.

Text+ node connected as the foreground to a Merge node
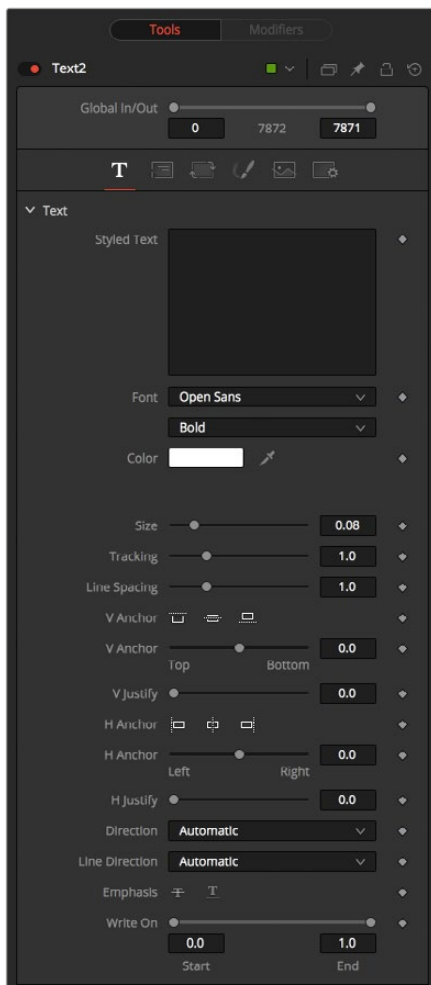
Inspector

Text+ Text tab controls

Text Tab

The Text tab in the Inspector is divided into three sections: Text, Advanced Controls, and Tab Spacing. The Text section includes parameters that will be familiar to anyone who has used a word processor. It includes commonly used text formatting options. The Advanced controls are used for kerning options, and the Tab Spacing is used to define the location and alignment of tabs in the layout.

Styled Text

The edit box in this tab is where the text to be created is entered. Any common character can be typed into this box. The common OS clipboard shortcuts (Command-C or Ctrl-C to copy, Command-X or Ctrl-X to cut, Command-V or Ctrl-V to paste) will also work; however, right-clicking in the edit box displays a custom contextual menu. More information on these modifiers can be found at the end of this section.

The Styled Text contextual menu includes the following options:

— **Animate:** Used to animate the text over time.
— **Character Level Styling:** Used to change the font, color, size and transformations of individual characters or words through the Modifiers tab.
— **Comp Name:** Places the name of the composition in the Styled text box for creating slates.
— **Follower:** A text modifier used to ripple animation across each character of the text.
— **Publish:** Publishes the text for connection to other text nodes.
— **Text Scramble:** A text modifier used to randomize the characters in the text.
— **Text Timer:** A text modifier used to display a countdown or the current date and time.
— **Time Code:** A text modifier used to display Time Code for the current frame.
— **Connect To:** Used to connect the text to the published output of another node.

Font

Two Font menus are used to select the font family and typeface, such as Regular, Bold, and Italic.

Color

Sets the basic fill color of the text. This is the same control displayed in the Shading tab color swatch.

Size

This control is used to increase or decrease the size of the text. This is not like selecting a point size in a word processor. The size is relative to the width of the image.

Tracking

The Tracking parameter adjusts the uniform spacing between each character of text.

Line Spacing

Line Spacing adjusts the distance between each line of text. This is sometimes called leading in word-processing applications.

V Anchor

The vertical anchor controls consist of three buttons and a slider. The three buttons are used to align the text vertically to the top of the text, middle of the text, or bottom baseline. The slider can be used to customize the alignment. Setting the vertical anchor will affect how the text is rotated as well as the location for line spacing adjustments. This control is most often used when the Layout type is set to Frame in the Layout tab.

V Justify

The vertical justify slider allows you to customize the vertical alignment of the text from the V Anchor setting to full justification so it is aligned evenly along the top and bottom edges. This control is most often used when the Layout type is set to Frame in the Layout tab.

H Anchor

The horizontal anchor controls consist of three buttons and a slider. The three buttons justify the text alignment to the left edge, middle, or right edge of the text. The slider can be used to customize the justification. Setting the horizontal anchor will affect how the text is rotated as well as the location for tracking (leading) spacing adjustments. This control is most often used when the Layout type is set to Frame in the Layout tab.

H Justify

The horizontal justify slider allows you to customize the justification of the text from the H Anchor setting to full justification so it is aligned evenly along the left and right edges. This control is most often used when the Layout type is set to Frame in the Layout tab.

Direction

This menu provides options for determining the Direction in which the text is to be written, either horizontally or vertically in either direction. This allows certain Asian languages to flow properly during animation.

Line Direction

These menu options are used to determine the text flow from top to bottom, bottom to top, left to right, or right to left.

Underline and Strikeout

These buttons apply the addition of emphasis styles to the text.

Write On

This range control is used to quickly apply simple Write On and Write Off effects to the text. To create a Write On effect, animate the End portion of the control from 1 to 0 over the length of time required. To create a Write Off effect, animate the Start portion of the range control from 0 to 1.
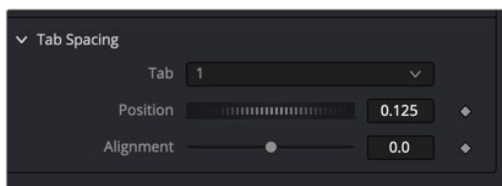
Tab Spacing

Tab Spacing

The controls in the Tabs section are used to configure the horizontal screen positions of eight separate tab stops. Any tab characters in the text will conform to these positions.

You can add tabs directly in the Styled Text input as you type. You can also add tabs by copying from another document, such as Text on macOS or Notepad on Windows, and paste it into the text box.

Position

This control is used to set the horizontal position of the tab in the frame. The values range from -0.5 to 0.5, where 0 is the center. The position of the tab will be indicated in the viewer by a thin vertical white line when the Text node is selected. At the top of each tab line in the viewer is a handle. The handle can be used to position the tab manually.

Alignment

Each tab can be set either left aligned, right aligned, or centered. This slider ranges from -1.0 to 1.0, where -1.0 is a left-aligned tab, 0.0 is a centered tab and 1.0 is a right-aligned tab. Clicking the tab handles in the viewer will toggle the alignment of the tab among the three states.
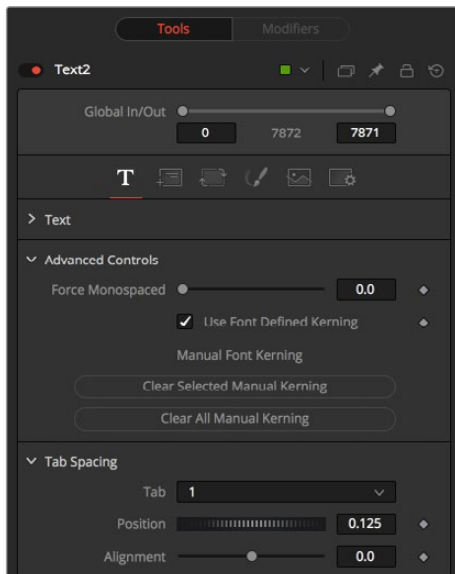
Advanced font controls

Reading Direction

These options allow you to set the reading direction of the text, either automatically or manually. You can specify Left to Right languages like English, German, etc. or Right to Left languages like Arabic and Hebrew.

Force Monospaced

This slider control can be used to override the kerning (spacing between characters) defined in the font. Setting this slider to zero (the default value) will cause Fusion to rely entirely on the kerning defined with each character. A value of one will cause the spacing between characters to be completely even, or monospaced.

Use Font Defined Kerning

This enables kerning as specified in the TrueType font and is on by default.

Use Ligatures

If your font supports ligatures, you can activate them here by choosing All Scripts. Ligatures combine individual letters into single glyphs, like **ff** and **fl**. If you're animating individual text letters, often you want the ligature letters separated individually rather than as a single glyph, so None is the default for Latin characters. Ligatures are required to render some languages like Arabic correctly, and use the Non-Latin setting.

Style Can Split Ligatures

Check this box If your font supports the option to split ligatures.

Stylistic Set

If your font includes stylistic sets, you can select them in the drop-down menu.

Font Features

This allows you to set OpenType 4 letter tags to activate certain font features. For example, "smcp" will show SMALL CAPITALS, and "frac" will display fractions as ½ instead of 1/2. Not all features are supported by a particular font. A full list of OpenType feature codes can be found here: https://docs. microsoft.com/en-us/typography/opentype/spec/featurelist

Manual Font Kerning/Placement

Right-clicking on this label will display a contextual menu that can be used to animate the kerning of the text. See the "Toolbar" section of this node later in this chapter for details on manual kerning.

Layout Tab

The controls used to position the text are located in the Layout tab. One of four layout types can be selected using the Type drop-down menu.

— **Point:** Point layout is the simplest of the layout modes. Text is arranged around an adjustable center point.

— **Frame:** Frame layout allows you to define a rectangular frame used to align the text. The alignment controls are used for justifying the text vertically and horizontally within the boundaries of the frame.

— **Circle:** Circle layout places the text around the curve of a circle or oval. Control is offered over the diameter and width of the circular shape. When the layout is set to this mode, the Alignment controls determine whether the text is positioned along the inside or outside of the circle's edge, and how multiple lines of text are justified.

— **Path:** Path layout allows you to shape your text along the edges of a path. The path can be used simply to add style to the text, or it can be animated using the Position on Path control that appears when this mode is selected.

Center X, Y, and Z

These controls are used to position the center of the layout element in space. X and Y are onscreen controls, and Center Z is a slider in the node controls.

Size

This slider is used to control the scale of the layout element.

Perspective

This slider control is used to add or remove perspective from the rotations applied by the Angle X, Y, and Z controls.

Rotation

Rotation consists of a series of buttons allowing you to select the order in which 3D rotations are applied to the text. Angle dials can be used to adjust the angle of the Layout element along any axis.

Width and Height

The Width control is visible when the Layout mode is set to Circle or Frame. The Height control is visible only when the Layout mode is set to Frame. Width and Height controls are used to adjust the dimensions and aspect of the Layout element.

Fit Characters

This menu control is visible only when the Layout type is set to Circle. This menu is used to select how the characters are spaced to fit along the circumference.

Position on Path

The Position on Path control is used to control the position of the text along the path. Values less than 0 or greater than 1 will cause the text to move beyond the path in the same direction as the vector of the path between the last two keyframes.

Background Color

The text generated by this node is normally rendered with a transparent background. This Color Picker control can be used to set a background color.

Right-Click Here for Shape Animation

This label appears only when the Layout type is set to Path. It is used to provide access to a contextual menu that provides options for connecting the path to other paths in the node tree, and animating the shape of the path over time.

For more information, *see Chapter 73, "Animating with Motion Paths,"* in the DaVinci Resolve Reference Manual, or Chapter 11 in the Fusion Reference Manual.
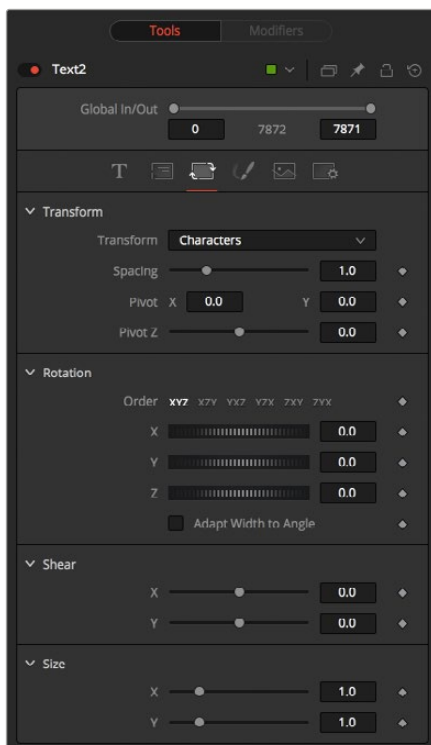
Text+ Transform tab controls

Transform Tab

The Transform tab is used to move, rotate, shear and scale text based on a character, word, or line.

Transform

The Transform menu is used to determine the portion of the text affected by the transformations applied in this tab. Transformations can be applied to line, word, and character levels simultaneously. This menu is only used to keep the visible controls to a reasonable number.

— **Characters:** Each character of text is transformed along its own center axis.

— **Words:** Each word is transformed separately on the word's center axis.

— **Lines:** Each line of the text is transformed separately on that line's center axis.

Spacing

The Spacing slider is used to adjust the space between each line, word, or character. Values less than 1 will usually cause the characters to begin overlapping.

Pivot X, Y, and Z

This provides control over the exact position of the axis. By default, the axis is positioned at the calculated center of the line, word, or character. The Axis control works as an offset, such that a value of 0.1, 0.1 in this control would cause the axis to be shifted downward and to the right for each of the text elements. Positive values in the Z-axis slider will move the axis of rotation further along the axis (away from the viewer). Negative values will bring the axis of rotation closer.

Rotation

These buttons are used to determine the order in which transforms are applied. X, Y, and Z would mean that the rotation is applied to X, then Y, and then Z.

X, Y, and Z

These controls can be used to adjust the angle of the text elements in any of the three dimensions.

Shear X and Y

Adjust these sliders to modify the slanting of the text elements along the X- and Y-axis.

Size X and Y

Adjust these sliders to modify the size of the text elements along the X- and Y-axis.
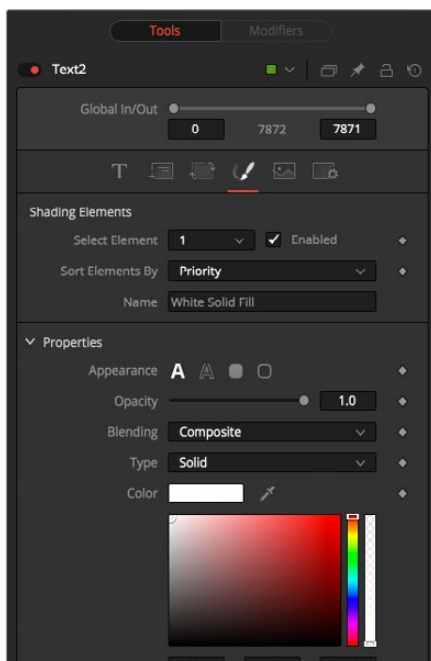
Text+ Shading tab controls

Shading Tab

The Shading tab provides controls to adjust the shading, texture, and softness of the text. Transformations can be controlled from this tab as well, applying additional transformations to as many as eight separate text shading elements independently.

Shading Element

The eight number values in the menu are used to select the element affected by adjustments in this tab.

Enabled

Select this checkbox to enable or disable each layer of shading elements. Element 1, which is the fill color, is enabled by default. The controls for a shading element will not be displayed unless this checkbox is selected.

Sort By

This menu allows you to sort the shading elements by number priority, with 1 being the topmost element and 8 being the bottommost element, or Z depth, based on the Z Position parameter.

Name

This text label can be used to assign a more descriptive name to each shading element you create.

Appearance

The four Appearance buttons determine how the shading element is applied to the text. Different controls will appear below depending on the appearance type selected.

— **Text Fill:** The shading element is applied to the entire text. This is the default mode.
— **Text Outline:** The shading element is drawn as an outline around the edges of the text.
— **Border Fill:** The shading element fills a border surrounding the text. Five additional controls are provided with this shading mode.
— **Border Outline:** The Border Outline mode draws an outline around the border that surrounds the text. It offers several additional controls.

Opacity

The Opacity slider controls the overall transparency of the shading element. It is usually better to assign opacity to a shading element than to adjust the Alpha of the color applied to that element.

Blending

This menu is used to select how the renderer deals with an overlap between two characters in the text.

— **Composite:** Merges the shading over the top of itself.
— **Solid:** Sets the pixels in the overlap region to opaque.
— **Transparent:** Sets the pixels in the overlap region to transparent.

Thickness

(Outline only) Thickness adjusts the thickness of the outline. Higher values equal thicker outlines.

Adapt Thickness to Perspective

(Outline only) Selecting this checkbox will cause your outline to become thinner where the text is farther away from the camera, and thicker where it is closer. This will create a more realistic outline for text transformed in 3D but takes significantly longer to render.

Outside Only

(Outline only) Selecting this checkbox will cause the outline to be drawn only on the outside edge of the text. By default, the outline is centered on the edge and partially overlaps the text.

Join Style

(Outline only) These buttons provide options for how the corners of the outline are drawn. Options include Sharp, Rounded, and Beveled.

Line Style

(Outline only) This menu offers additional options for the style of the line. Besides the default solid line, a variety of dash and dot patterns are available.

Level

(Border Fill only) This is used to control the portion of the text border filled.

— **Text:** This draws a border around the entire text.
— **Line:** This draws a border around each line of text.
— **Word:** This draws a border around each word.
— **Character:** This draws a border around each character.

Extend Horizontal and Extend Vertical

(Border only) Use this slider to change the dimensions of each border.

Round

(Border Fill and Border Outline only) This slider is used to round off the edges of the border.

Color Types

Besides solid shading, it is also possible to use a gradient fill or map an external image onto the text. This menu is used to determine if the color of the shading element is derived from a user-selected color or gradient, or if it comes from an external image source. Different controls will be displayed below depending on the Color Type selected.

— **Solid:** When the Type menu is set to Solid mode, color selector controls are provided to select the color of the text.
— **Image:** The output of a node in the node tree will be used to texture the text. The node used is chosen using the Color Image control that is revealed when this option is selected.
— **Gradient:** When the Type menu is set to Gradient, additional controls are displayed where the gradient colors can direction can be customized.

Using the Gradient Color Bar

The gradient color bar is used to select the colors for the gradient. The default two color stops set the start and end colors. You can change the colors used in the gradient by selecting the color stop, and then using the Eyedropper or color swatch to set a new color.

You can add, move, copy and delete color using the gradient bar.

To modify one of the colors, select the triangle below the color on the bar.

To add a color stop to the gradient bar:

1 Click anywhere along the bottom of the gradient bar.

2 Use the Eyedropper or color swatch to select the color for the color stop.

To move a color stop on the gradient bar:

— Drag a color stop left or right along the bar.

To copy a color stop:

— Hold Command (macOS) or Ctrl (Windows) while you drag a color stop.

To delete a color stop:

— Drag the color stop up past the gradient bar.

Image Source

(Image Mode only) The Image Source menu includes three options for acquiring the image used to fill the text.

— **Tool:** Displays a Color image text field where you can add a tool from the node tree as the fill for text.

— **Clip:** Provides a Browse button to select a media file from your hard drive as the fill for text.

— **Brush:** Displays a Color Brush menu where you can select one of Fusion's paint brush bitmaps as the fill for text.

Color Image/Color Brush

(Image Mode only) The Color Image text box and Color Brush menu are used to select the tool or paint brush that will fill the text. In the Color Image text box, enter the name of the node in the node tree that will provide the image. You can type the name in with the keyboard, drag the node from the node tree into the text box, or right-click and select Connect To from the contextual menu to select the image to be used. For the Color Brush, select the Brush from the menu.

Image Sampling

(Image Mode only) This menu is used to select the sampling type for shading rendering and transformations. The default of Pixel shading is sufficient for 90% of tasks. To reduce detectable aliasing in the text, set the sampling type to Area. This is slower but may produce better-quality results. A setting of None will render faster, but with no additional sampling applied so the quality will be lower.

Image Edges

(Image Mode only) This menu is used to choose how transformations applied to image shading elements are handled when they wrap off the text's edges.

Shading Mapping

(Image Mode only) This menu is used to select whether the entire image is stretched to fill the text or scaled to fit, maintaining the aspect ratio but cropping part of the image as needed.

Mapping Angle

(Image and Gradient Modes only) This control rotates the image or gradient on the Z-axis.

Mapping Size

(Image and Gradient Modes only) This control scales the image or gradient.

Mapping Aspect

(Image and Gradient Modes only) This control vertically stretches or shrinks the image or gradient.

Mapping Level

(Image and Gradient Modes only) The Mapping Level menu is used to select how the image is mapped to the text.

— **Full Image:** Applies the entire image to the text.
— **Text:** Applies the image to fit the entire set of text.
— **Line:** Applies the image per line of text.
— **Word:** Applies the image per each word of text.
— **Character:** Applies the image per individual character.

Softness X and Y

These sliders control the softness of the text outline used to create the shading element. Control is provided for the X- and Y-axis independently.

Apply Softness to Fill Color

Selecting this checkbox will cause blur (softness) to be applied to the shading element. The effect is best seen when applied to a shading element using an external image.

Softness Glow

This slider will apply a glow to the softened portion of the shading element.

Softness Blend

This slider controls the amount that the result of the softness control is blended back with the original. It can be used to tone down the result of the soften operation.

Priority Back/Front

Only enabled when the Sort By menu is set to Priority, this slider overrides the priority setting and determines the layer's order for the shading elements. Slide the control to the right to bring an element closer to the front. Move it to the left to tuck one shading element behind another.

Offset X, Y, and Z

These controls are used to apply offset from the text's global center (as set in the Layout tab) for the shading elements. A value of X0.0, Y0.1 in the coordinate controls would place the shading element centered with 10 percent of the image further down the screen along the Y-axis. Positive values in the Z-Offset slider control will push the center further away from the camera, while positive values will bring it closer to the camera.

Pivot X, Y, and Z

These controls are used to set the exact position of the axis for the currently selected shading element. By default, the axis is positioned at the calculated center of the line, word, or character.

The axis control works as an offset, such that a value of 0.1, 0.1 in this control would cause the axis to be shifted downward and to the right for the shading element. Positive values in the Z-axis slider will move the axis of rotation further along the axis (away from the viewer). Negative values will bring the axis of rotation closer.

Rotation X, Y, and Z

These controls are used to adjust the angle of the currently selected shading element in any of the three dimensions.

Shear X and Y

Adjust these sliders to modify the slanting of the currently selected shading element along the X and Y axis.

Size X and Y

Adjust these sliders to modify the size of the currently selected shading element along the X and Y axis.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are duplicated in many Generator nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Text+ Toolbar

When the Text node is selected, a toolbar will appear in the viewer. Each button is described below from left to right.

Text+ toolbar

Allow Typing in Viewer

When selected, you can type and edit text directly in the viewer. Clicking on the text in the viewer will move the insert cursor within the text. Left and Right Arrow keys will move the insert cursor between characters. Using the Up and Down Arrows will move between text lines.

Allow Manual Kerning

The Manual Kerning button overrides the automatic kerning normally applied to text. Clicking the small red handle at the bottom of any character selects the character for kerning. You can also draw a selection rectangle around the characters you want to kern. Once any characters are selected, hold down the Option (macOS) or Alt key (Windows) while pressing the Left or Right Arrow key to make small adjustments to the kerning of the selected characters. Hold the Alt + Shift (Windows) or Option + Shift (macOS) keys down while pressing arrow keys to move the character in larger increments.

To animate the position of each character, right-click on the control label Manual Font Kerning in the Inspector's Advanced Controls and select Animate from the contextual menu. A new key will be set on the animation spline each time a character is moved. All characters are animated with the same spline, as with polyline mask animation.

No Text Outline

When this button is selected, it disables the drawing of any outline around the edges of the text. The outline is not a part of the text; it is an onscreen control used to help identify the position of the text. This is a three-way toggle with the Text Outline Outside Frame Only, and Show Always Text Outline buttons.

Text Outline Outside Frame Only

This button draws an outline around the edges of text, which is outside the visible frame. This is useful for locating text that has moved offscreen and is no longer rendering a visible result. This is a three-way toggle with the No Text Outline, and Show Always Text Outline buttons.

Show Always Text Outline

This button draws an outline around the edges of text, whether or not the text is visible within the frame. This is a three-way toggle with the No Text Outline, and Text Outline Outside Frame Only buttons.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are duplicated in many Generator nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Text+ Modifiers
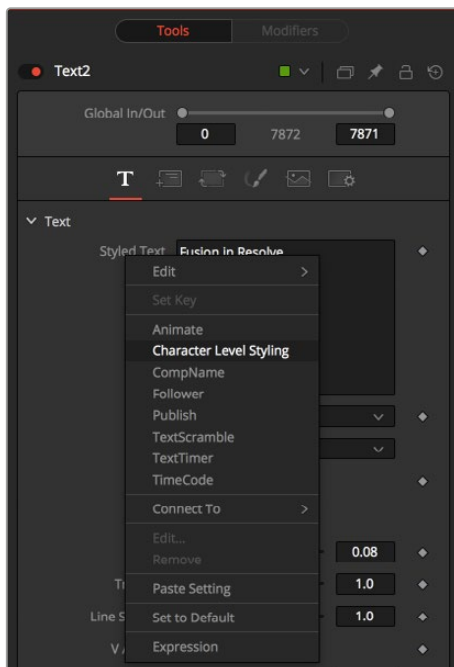
Text+ modifiers

Text modifiers can be assigned by right-clicking in the Styled Text box and selecting a modifier from the contextual menu. Once a modifier is selected, its controls are found in the Modifiers tab at the top of the Inspector.

Character Level Styling

The Character Level Styling modifier works only on Text+ nodes. Once applied, individual characters can be selected directly in the viewer, and different text attributes can be applied to them using the controls in the Modifiers tab.

> **NOTE:** Character Level Styling can only be directly applied to Text+ nodes, not to Text 3D nodes. However, styled text from a Text+ node can be applied to a Text 3D node by copying the Text+, right-clicking on the Text 3D, and choosing Paste Settings.

Inspector
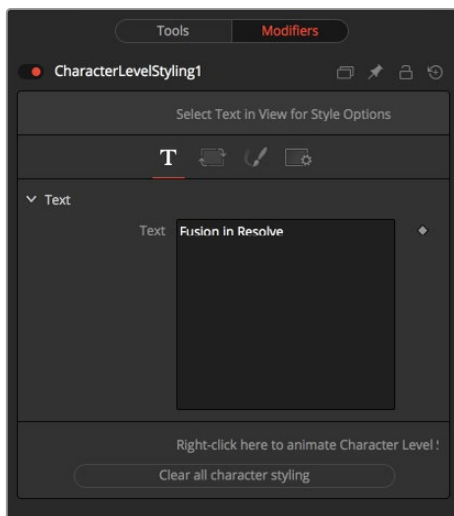
Character Level Styling Modifiers tab

Text Tab

The Styled Text box in the Modifiers tab displays the same text in the Tools tab of the Text+ Inspector. However, individual characters you want to modify cannot be selected in the Styled Text box; they must be selected in the viewer. Once text is selected in the viewer, the Text tab includes familiar text formatting options that will apply only to the selected characters.

Clear Character Styling on Selection

All changes made to the currently selected characters will be reset.

Clear All Character Styling

All character attributes will be reset to their original values.

Transform Tab and Shading Tab

For details on these Text+ tabs, see the "Text+" section above.

Comp Name

The Comp Name sets the styled text to become the current Composition Name. This is quite useful to automate burn-ins for daily renderings. See also the TimeCode modifier. It can be applied by right-clicking in the Styled Text field of a Text+ node and selecting Comp Name.

Controls

This modifier has no controls.

Follower

The Follower modifier allows sequencing text animations. The modifier is applied by right-clicking in the Styled Text field of a Text+ node and selecting Follower. In the Modifiers tab, you start by animating the parameters of the text (note that changing any parameter in the Modifiers tab will not be visible unless a keyframe is added.) Then, in the Timing tab you set the animation's delay between characters.

Inspector
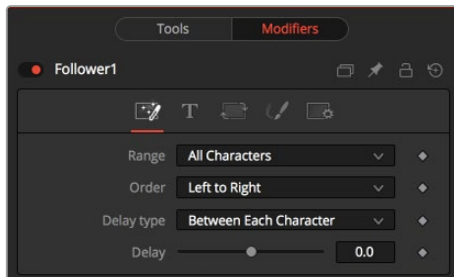
Follower Timing tab

Timing Tab

Once the text is animated using the controls in the Modifiers tab, the Timing tab is used to choose the direction and offset of the animation.

Range

The Range menu is used to select whether all characters should be influenced or only a selected range. To set the range, you can drag-select over the characters directly in the viewer.

Order

The Order menu determines in which direction the characters are influenced. Notice that empty spaces are counted as characters as well. Available options include:

— **Left to right:** The animation ripples from left to right through all characters.
— **Right to left:** The animation ripples from right to left through all characters.
— **Inside out:** The animation ripples symmetrically from the center point of the characters toward the margin.
— **Outside in:** The animation ripples symmetrically from the margin toward the center point of the characters.

— **Random but one by one:** The animation is applied to randomly selected characters but only influences one character at a time.

— **Completely random:** The animation is applied to randomly selected characters, influencing multiple characters at a time.

— **Manual curve:** The affected characters can be specified by sliders.

Delay Type

Determines what sort of delay is applied to the animation. Available options include:

— **Between Each Character:** The more characters there are in your text, the longer the animation will take to the end. A setting of 1 means the first character starts the animation, and the second character starts 1 frame later, the third character starts 1 frame after the second, and so on.

— **Between First and Last Character:** No matter how many characters are in your text, the animation will always be completed in the selected amount of time.

Clear All Character Styling

All character attributes will be reset to their original values.

Text Controls, Alignment, Transform, and Shading Tabs

In these tabs, the actual animation for the characters is done. Observe that simply changing a value in these tabs will have no influence at all. The value must be animated for the effect to show.

For a detailed description on the various parameters, see the Text+ node documentation.

Text Scramble

The Text Scramble randomly replaces the characters with others from a user definable set. It can be applied by right-clicking into the Styled Text field of a Text+ node and selecting Text Scramble.

The Controls for the Text Scramble are then adjusted in the Modifiers tab.

Inspector
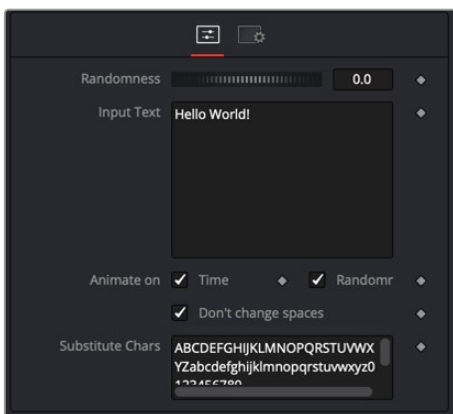
Text Scramble modifier Controls tab

Controls Tab

The Controls tab in the Text Scramble modifier is used to enter text and scramble it using the Randomness control. The scrambled characters are taken from the Substitute Chars field at the bottom of the Inspector.

Randomness

Defines how many characters are exchanged randomly. A value of 0 will change no characters at all. A value of 1 will change all characters in the text. Animating this thumbwheel to go from 0 to 1 will gradually exchange all characters.

Input Text

This reflects the original text in the Text+ Styled Text field. Text can be entered either here or in the Text+ node.

Animate on Time

When enabled, the characters are scrambled randomly on every new frame. This switch has no effect when Randomness is set to 0.

Animate on Randomness

When enabled, the characters are scrambled randomly on every new frame, when the Randomness thumbwheel is animated.

This switch has no effect when Randomness is set to 0.

Don't Change Spaces

When enabled, spaces are not scrambled, allowing the length of the single words to stay the same, although their characters get scrambled around.

Substitute Chars

This field contains the characters used to scramble the text.

Text Timer

The Text Timer makes the Text+ node either a Countdown, a Timer, or a Clock. This is useful for onscreen real-time displays or to burn in the creation time of a frame into the picture. It can be applied by right-clicking in the Styled Text field of a Text+ node and selecting Text Timer.

Inspector
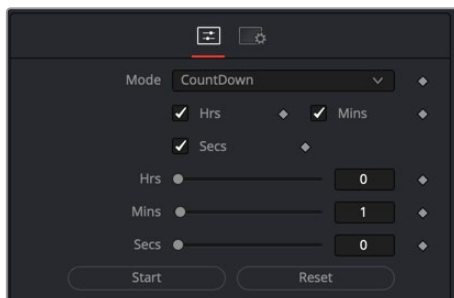
Text Timer modifier Controls tab

Controls Tab

The Controls tab for the Text Timer modifier is used to set up the type of time display that is generated by this modifier.

Mode

This menu sets the mode the timer is working in. The choices are CountDown, Timer, and Clock. In Clock mode, the current system time will be displayed.

Hrs, Mins, Secs (Checkboxes)

Defines which parts of the clock should be shown onscreen.

Hrs, Mins, Secs (Sliders)

Sets the start time for the CountDown and Timer mode.

Start

Starts the Counter or Timer. Toggles to Stop once the timer is running.

Reset

Resets the Counter and Timer to the values set by the sliders.

Time Code

The Time Code only works on Text+ nodes. It sets the Styled text to become a counter based on the current frame. This is quite useful for automating burn-ins for daily renderings.

It can be applied by right-clicking in the Styled Text field of a Text+ node and selecting Time Code.

Inspector
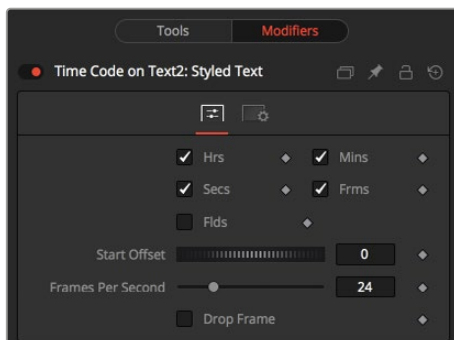
Time Code modifier Controls tab

Controls Tab

The Controls tab for the Time Code modifier is used to set up the time code display that is generated by this modifier.

Hrs, Mins, Secs, Frms, Flds

Activate or deactivate these options to customize the time code display to show hours, minutes, seconds, frames, and fields, respectively. Activating frames will only give you a plain frame counter.

Start Offset

Introduce a positive or negative offset to Fusion's current time to match up with existing time codes.

Frames per Second

This should match with your composition's FPS setting to provide accurate time measurement.

Drop Frame

Activate this checkbox to match the time code with footage that has drop frames—for example, certain NTSC formats.

The Common Controls

Nodes that generate images share a number of identical controls in the Inspector. This section describes controls that are common among Generator nodes.

Inspector
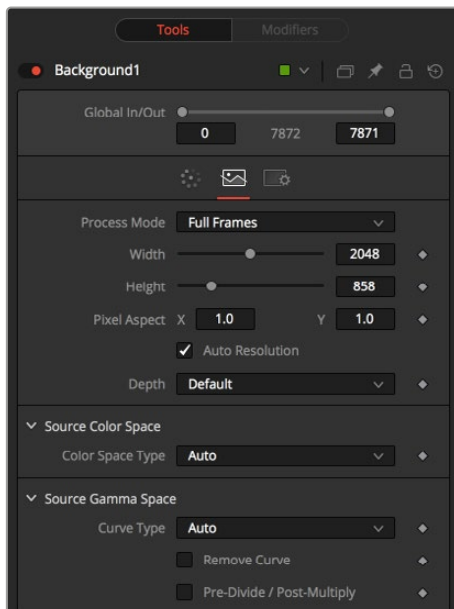
Background node Image tab

Image Tab

The controls in this tab are used to set the resolution, color depth, and pixel aspect of the image produced by the node.

Process Mode

Use this menu control to select the Fields Processing mode used by Fusion to render changes to the image. The default option is determined by the Has Fields checkbox control in the Frame Format preferences.

Global In and Out

Use this control to specify the position of this node within the project. Use Global In to specify which frame that starts the clip and Global Out to specify which frame ends the clip (inclusive) within the project's Global Range.

The node will not produce an image on frames outside this range.

Use Frame Format Settings

When this checkbox is selected, the width, height, and pixel aspect of the image created by the node will be locked to values defined in the composition's Frame Format preferences. If the Frame Format preferences change, the resolution of the image produced by the node will change to match. Disabling this option can be useful to build a composition at a different resolution than the eventual target resolution for the final render.

Width/Height

This pair of controls is used to set the Width and Height dimensions of the image to be created by the node.

Pixel Aspect

This control is used to specify the Pixel Aspect ratio of the created images. An aspect ratio of 1:1 would generate a square pixel with the same dimensions on either side (like a computer display monitor), and an aspect of 0.9:1 would create a slightly rectangular pixel (like an NTSC monitor).

> **NOTE:** Right-click on the Width, Height, or Pixel Aspect controls to display a menu listing the file formats defined in the preferences Frame Format tab. Selecting any of the listed options will set the width, height, and pixel aspect to the values for that format, accordingly.

Depth

The Depth drop-down menu is used to set the pixel color depth of the image created by the Creator node. 32-bit pixels require 4X the memory of 8-bit pixels but have far greater color accuracy. Float pixels allow high dynamic range values outside the normal 0...1 range, for representing colors that are brighter than white or darker than black.

Source Color Space

You can use the Source Color Space menu to set the Color Space of the footage to help achieve a linear workflow. Unlike the Gamut tool, this doesn't perform any actual color space conversion, but rather adds the source space data into the metadata, if that metadata doesn't exist. The metadata can then be used downstream by a Gamut tool with the From Image option, or in a Saver, if explicit output spaces are defined there. There are two options to choose from:

— **Auto:** Automatically reads and passes on the metadata that may be in the image.
— **Space:** Displays a Color Space Type menu where you can choose the correct color space of the image.

Source Gamma Space

Using the Curve type menu, you can set the Gamma Space of the footage and choose to remove it by way of the Remove Curve checkbox when working in a linear workflow. There are three choices in the Curve type menu:

— **Auto:** Automatically reads and passes on the metadata that may be in the image.
— **Space:** Displays a Gamma Space Type menu where you can choose the correct gamma curve of the image.
— **Log:** Brings up the Log/Lin settings, similar to the Cineon tool. For more information, *see Chapter 98, "Film Nodes."* in the DaVinci Resolve Reference Manual, or Chapter 36 in the Fusion Reference Manual.

Remove Curve

Depending on the selected Gamma Space or on the Gamma Space found in Auto mode, the Gamma Curve is removed from, or a log-lin conversion is performed on, the material, effectively converting it to a linear output space.

Fast Noise Image Tab Options

The following controls are specific the Image tab in the Fast Noise Node.

Mask Map Inputs

These external connections allow you to use masks to control the value of the Noise Detail and Brightness controls individually for each pixel. This can allow some interesting and creative effects.

Noise Detail Map

A soft-edged mask connected to the Noise Detail Map will give a flat noise map (zero detail) where the mask is black, and full detail where it is white, with intermediate values smoothly reducing in detail. It is applied before any gradient color mapping. This can be very helpful for applying maximum noise detail in a specific area, while smoothly falling off elsewhere.

Noise Brightness Map

A mask connected to this input can be used to control the noise map completely, such as boosting it in certain areas, combining it with other textures, or if Detail is set to 0, replacing the Perlin Noise map altogether.

Settings Tab

The Settings Tab in the Inspector can be found on every tool in the Color category. The Settings controls are even found on third-party Color-type plugin tools. The controls are consistent and work the same way for each tool, although some tools do include one or two individual options, which are also covered here.
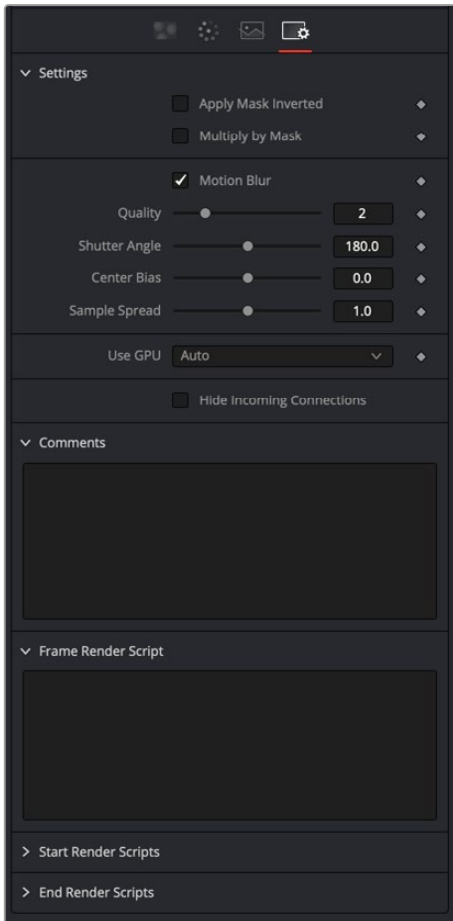
Common Generator settings

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming. Normally, this will cause the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if processing of this node is scripted to trigger another task, but the value of the node is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the red button on a Blur tool is deselected, the blur will first be applied to the image, and then the red channel from the original input will be copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this will generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option will cause the RGB values of the masked image to be multiplied by the mask channel's values. This will cause all pixels of the image not in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object ID and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels will be used if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID will be used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the view. The image or sequence must have been rendered from a 3D software package with those channels included.

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 will cause Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one whole frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Use GPU

The user GPU menu has three settings. Setting the menu to Disable turns off hardware-accelerated rendering using the graphics card in your computer. Enabled uses the hardware, and Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available

Comments

The Comments field is used to add notes to a tool. Click in the field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

I/O Nodes

This chapter details the input and output of media using Loader and Saver nodes within Fusion Studio as well as the MediaIn and MediaOut nodes in DaVinci Resolve.

> The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

Contents

Loader Node [Ld]

The Loader node

Loader Node Introduction

> **NOTE:** The Loader node in DaVinci Resolve is only used for importing EXR files.

When using Fusion Studio, the Loader node is the node you use to select and load footage from your hard drive into the Node Editor. There are three ways to add a Loader node, and consequently a clip, to the Node Editor.

— Add the Loader from the Effects Library or toolbar (Fusion Studio only), and then use Loader's file browser to bring a clip into the Node Editor
— Drag clips from an OS window directly into the Node Editor, creating a Loader node in the Node Editor.
— Choose File > Import > Footage (Fusion Studio only), although this method creates a new composition as well as adds the Loader node to the Node Editor.

When a Loader is added to the Node editor, a File dialog is displayed automatically to allow the selection of a clip from your hard drives.

> **NOTE:** You can disable the automatic display of the file browser by disabling Auto Clip Browse in the Global > General Preferences.

Once clips are brought in using the Loader node, the Loader is used for trimming, looping, and extending the footage, as well as setting the field order, pixel aspect, and color depth. The Loader is arguably the most important tool in Fusion Studio.

Inputs

The single input on the Loader node is for an effect mask to crop the image brought in by the Loader.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the loaded image to appear only within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Loader node is a 2D image of any format supported in Fusion Studio. It is limited to an EXR format in DaVinci Resolve. Below, the LOADER imports an image, which is then masked using an Ellipse matte. The output of the masked LOADER is passed onto 2D image-processing nodes.
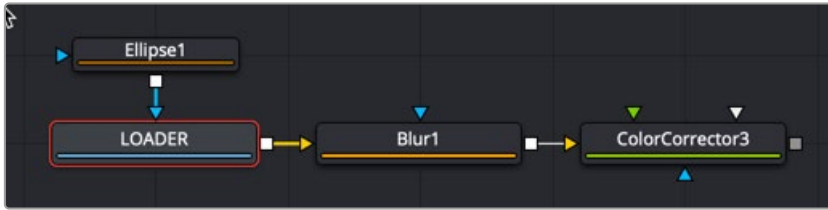
A Loader node used for importing images

Inspector
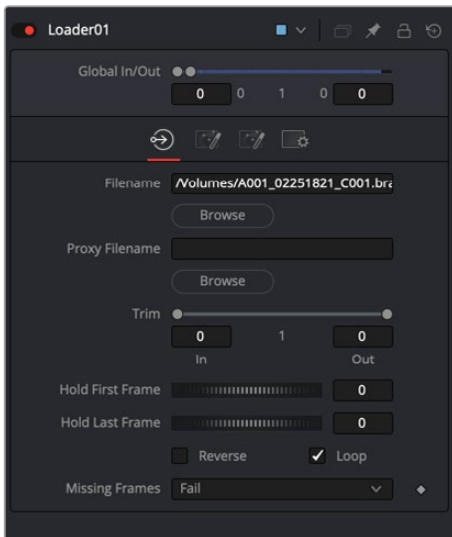
Loader node File tab

File Tab

The File tab for the Loader includes controls for trimming, creating a freeze frame, looping, and reversing the clip. You can also reselect the clip that the Loader links to on your hard drive.

Global In and Out

The Global In and Out handles are used to specify the position of this node within the project. Use Global In to specify the frame on which that the clip starts and use Global Out to specify the frame on which the clip ends within the project's global range. The node does not produce an image on frames outside of this range.

If the Global In and Out values are decreased to the point where the range between the In and Out values is smaller than the number of available frames in the clip, Fusion automatically trims the clip by adjusting the Trim range control. If the Global In/Out values are increased to the point where the range between the In and Out values is larger than the number of available frames in the clip, Fusion automatically lengthens the clip by adjusting the Hold First/Last Frame controls. Extended frames are visually represented in the range control by changing the color of the held frames to green in the control.

To slide the clip in time or move it through the project without changing its length, place the mouse pointer in the middle of the range control and drag it to the new location, or enter the value manually in the Global In value box.

Filename

The Filename field shows the file path of the clip imported to the Node Editor by the Loader node. Clicking on the Browse button opens a standard file browser. The path to the footage can also be typed directly using the field provided. The text box supports filename completion. As the name of a directory or file is typed in the text box, Fusion displays a pop-up that lists possible matches. Use the arrow keys to select the correct match and complete the path.

> **NOTE:** Loading image sequences is common practice for compositing, whether the image sequence comes from a 3D renderer or a digital cinema camera. If the last part of a file's name is a number (not counting the file extension), Fusion automatically scans the directory looking for files that match the sequence. For example, the following filenames would be valid sequences:
>
> `image.0001.braw , image.0002.braw, image.0003.braw …`
>
> or
>
> `image151.exr , image152.exr, image153.exr …`
>
> The following would not be considered a sequence since the last characters are not numeric.
>
> `shot.1.fg.jpg, shot.2.fg.jpg, shot.3.fg.jpg`
>
> It is not necessary to select the first file in the sequence. Fusion searches the entire folder for files matching the sequence in the selected filename. Also, Fusion determines the length of the sequence based on the first and last numeric value in the filenames. Missing frames are ignored. For example, if the folder contains two files with the following names:
>
> `image.0001.exr, image.0100.exr`
>
> Fusion sees this as a file sequence with 100 frames, not an image sequence containing two frames. The Missing Frames drop-down menu is used to choose how Fusion handles missing frames.
>
> The Trim In/Trim Out control's context menu can also be used to force a specific clip length or to rescan the folder. Both controls are described in greater detail below.
>
> Occasionally, you want to load only a single frame out of a sequence—e.g., a photograph from a folder containing many other files as well. By default, Fusion detects those as a sequence, but if you hold Shift while dragging the file from the OS window to the Node Editor, Fusion takes only that specific file and disregards any sequencing.

Proxy Filename

The Proxy Filename control only appears once the filename control points to a valid clip. The Proxy Filename can specify a clip that is loaded when the Proxy mode is enabled. This allows smaller versions of the image to be loaded to speed up file I/O from disk, and processing. For example, create a 1/4-scale version of an 8K EXR sequence to use as EXR proxy files. Whenever the Proxy mode of the Composition is enabled, the smaller resolution proxy clip is loaded from disk, and all processing is performed at the lower resolution, significantly improving render times. This is particularly useful when working with large RAW plates stored on a remote file server. Lower-resolution versions of the plates can be stored locally, reducing network bandwidth, interactive render times, and memory usage.The proxy clip must have the same number of frames as the source clip, and the sequence numbers for the clip must start and end on the same frame numbers. It is strongly suggested that the proxies are the same format as the main files. In the case of formats with options, such as Cineon, DPX, and OpenEXR, the proxies use the same format options as the primary.

Trim

The Trim range control is used to trim frames from the start or end of a clip. Adjust the Trim In to remove frames from the start and adjust Trim Out to specify the last frame of the clip. The values used here are offsets. A value of 5 in Trim In would use the fifth frame in the sequence as the start, ignoring the first four frames. A value of 95 would stop loading frames after the 95th.

Hold First Frame/Hold Last Frame:

The Hold First Frame and Hold Last Frame controls hold the first or last frame of the clip for the specified amount of frames. Held frames are included in a loop if the footage is looped.

Reverse

Select this checkbox to reverse the footage so that the last frame is played first, and the first frame is played last.

Loop

Select this checkbox to loop the footage until the end of the project. Any lengthening of the clip using Hold First/Last Frame or shortening using Trim In/Out is included in the looped clip.

Missing Frames

The Missing Frames menu determines the Loader behavior when a frame is missing or is unable to load for any reason.

— **Fail:** The Loader does not output any image unless a frame becomes available. Rendering is aborted.

— **Hold Previous Output:** The last valid frame is held until a frame becomes available again. This fails if no valid frame has been seen—for example, if the first frame is missing.

— **Output Black:** Outputs a black frame until a valid frame becomes available again.

— **Wait:** Fusion waits for the frame to become available, checking every few seconds. This is useful for rendering a composition simultaneously with a 3D render. All rendering ceases until the frame appears.

The Magic Comp Variable

Loaders and Savers use the absolute file paths for the location of media. However, if you are using Fusion Studio, you can use a file path that is relative to the saved composition location. The Comp variable works for Loaders and Savers and helps you to keep your work organized. Entering Comp:\ in place of the full file path name is a shortcut for the folder where your Fusion composition document is saved.

You can either enter the Comp:\ manually into the filename field of a Loader, or turn on the Enable Reverse Mapping of Paths Preferences checkbox in the Path Map preferences. Enabling the Path Map preference check box will use the Comp:\ automatically.

So as long as all your source footage is stored in subfolders of your Comp folder, Fusion finds that footage regardless of the actual hard drive or network share name.

You could, for example, copy an entire shot from the network to your local drive, set up your Loaders and Savers to use the Comp variable, work all your magic locally (i.e., set up your composition), and then copy just the composition back to the server and issue a net-render. All render slaves automatically find the source footage.

Some examples:

Your composition is stored in:

`X:\Project\Shot0815\Fusion\Shot0815.comp`

Your source footage sits in:

`X:\Project\Shot0815\Fusion\Greenscreen\0815Green_0000.dpx`

The relative path in the Loader node would then be:

`Comp:\Greenscreen\0815Green_0000.dpx`

If your source footage is stored in:

`X:\Project\Shot0815\Footage\Greenscreen\0815Green_0000.dpx`

The relative path in the Loader node would then be:

`Comp:\..\Footage\ Greenscreen\0815Green_0000.dpx`

Observe how the two dots .. set the directory to go up one folder, much like CD .. in a Command Shell window.
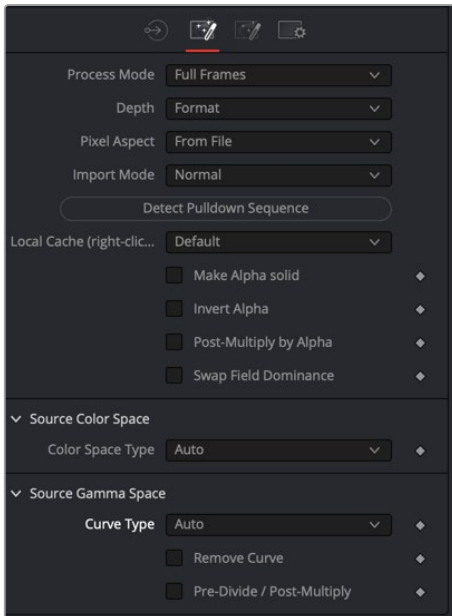
Loader Import tab

Import Tab

The Import tab includes settings for the frame format and how to deal with fields, pixel aspect, 3:2 pull down/pull up conversion, and removing gamma curve types for achieving a linear workflow.

Process Mode

Use this menu to select the Fields Processing mode used by Fusion when loading the image. The Has Fields checkbox control in the Frame Format preferences determines the default option, and the default height as well. Available options include:

— Full frames

— NTSC fields

— PAL/HD fields

— PAL/HD fields (reversed)

— NTSC fields (reversed).

The two reversed options load fields in the opposite order and thus result in the fields being spatially swapped both in time order and in vertical order as well.

Use the Swap Field Dominance checkbox (described below) to swap fields in time only.

Depth

The Depth menu is used to select the color depth used to process footage from this Loader. The default option is Format.

— **Format:** The color depth is determined by the color depth supported in the file format loaded. For example, JPEG files automatically process at 8 bit because the JPEG file format does not store color depths greater than 8. EXR files load at Float. If the color depth of the format is undetermined, the default depth defined in the Frame Format preferences is used. Formats that support multiple color depths are set to the appropriate color depth automatically.

— **Default:** The color depth is determined by the settings in the composition's Frame Format Preferences panel.

— **Int 8 Bit/Int 16 Bit/Float 16/Float 32:** These options set the color depth for processing the image.

Pixel Aspect

This menu is used to determine the image's pixel aspect ratio.

— **From File:** The loader conforms to the image aspect detected in the saved file. There are a few formats that can store aspect information. TIFF, JPEG, and OpenEXR are examples of image formats that may have the pixel aspect embedded in the file's header. When no aspect ratio information is stored in the file, the default frame format method is used.

— **Default:** Any pixel aspect ratio information stored in the header of the image file is ignored. The pixel aspect set in the composition's frame format preferences is used instead.

— **Custom:** Select this option to override the preferences and set a pixel aspect for the clip manually. Selecting this button causes the X/Y Pixel Aspect control to appear.

Custom Pixel Aspect

This control is visible only when Custom is selected from the Pixel Aspect menu. Enter the desired X and Y aspect or right-click on the control to display a menu of common frame formats and their aspects.

Import Mode

This menu provides options for removing pull-up from an image sequence. Pull-up is a reversible method of combining frames used to convert 24 fps footage into 30 fps. It is commonly used to broadcast NTSC versions of films.

— **Normal:** This passes the image without applying pull-up or pull-down 2:3.

— **Pull Up:** This removes existing 3:2 pull-down applied to the image sequence, converting from 30 fps to 24 fps 2:3.

— **Pull Down:** The footage has pull-down applied, converting 24 fps footage to 30 fps by creating five frames out of every four. The process mode of a Loader set to Pull Down should always be Full Frames.

First Frame

This menu appears when the Import Mode is set to either Pull Up or Pull Down. It is used to determine which frame of the 3:2 sequence is used as the first frame of the loaded clip.

Detect Pull-Down Sequence

This button is used to detect and set the pull-up sequence of the footage automatically. It only works if Pull-Up or Pull-Down is first selected from the Import Mode menu. If it succeeds in detecting the order, the First Frame control automatically sets to the correct value.

Make Alpha Solid

When enabled, the original Alpha channel of the clip is cleared and set to solid white (completely opaque).

Invert Alpha

When enabled, the original Alpha channel of the clip is inverted. This may also be used in conjunction with Make Alpha Solid to set the Alpha to pure black (completely transparent).

Post-Multiply by Alpha

Enabling this option causes the color value of each pixel to be multiplied by the Alpha channel for that pixel. This option can be used to convert subtractive (non-premultiplied) images to additive (premultiplied) images.

Swap Field Dominance

When enabled, the field order (dominance) of the image is swapped, so the order in time the fields appear is reversed. Unlike the Process Mode control, this is done without spatially swapping the scanlines in the image

Color Space Type

This menu is used to set the Color Space of the footage to help achieve a linear color space workflow. Unlike the Gamut tool, this doesn't perform any actual color space conversion, but instead adds the source color space data into the metadata, if that metadata doesn't already exist. The metadata can then be used downstream by a Gamut tool with the From Image option, or in a Saver if explicit output spaces are defined there.

— **Auto:** Passes along any metadata that might be in the incoming image.
— **Space:** Allows the user to set the color space based on the recording device used to capture content or software settings used when rendering the content in another application.

Curve Type

This menu is used to determine the gamma curve of the footage. Once the Gamma Curve Type is set, you can choose to remove the curve to help achieve a linear workflow.

— **Auto:** Passes along any metadata that might be in the incoming image.
— **Space:** Allows the user to set the gamma curve based on the recording device used to capture content or software settings used when rendering the content in another application.
— **Log:** Displays the Log/Lin settings, similar to the Cineon Log node. For more information on the Log settings, refer to Chapter 38, "Film Nodes" in the Fusion Reference Manual or Chapter 98 in the DaVinci Resolve Reference Manual.

Remove Curve

Depending on the selected Curve Type or on the Gamma Space found in Auto mode, the associated Gamma Curve is removed from, or a log-lin conversion is performed on, the material, effectively converting it to a linear output space.
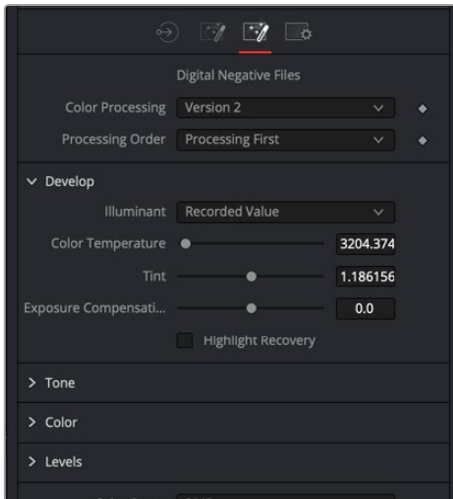
Format tab

Format Tab

The Format tab contains file format-specific controls that dynamically change based on the selected Loader and the file it links to. Some formats contain a single control or no controls at all. Others like Camera RAW formats contain RAW-specific debayering controls. A partial format list is provided below for reference.

— **OpenEXR:** EXR provides a compact and flexible format to support high dynamic range images (float). The format also supports a variety of extra channels and metadata.The Format tab for OpenEXR files provides a mechanism for mapping any non-RGBA channels to the channels supported natively in Fusion. Using the Format tab, you can enter the name of a channel contained in the OpenEXR file into any of the edit boxes next to the Fusion channel name. A command line utility for dumping the names of the channels can be found at https://www. openexr.com.

— **QuickTime:** QuickTime files can potentially contain multiple tracks. Use the format options to select one of the tracks.

— **Cinema DNG:** CinemaDNG is an open format capable of high-resolution raw image data with a wide dynamic range. It was one of the formats recorded by Blackmagic Design cameras before switching over to the BRAW format.

— **Photoshop PSD Format:** Fusion can load any one of the individual layers stored in the PSD file, or the completed image with all layers. Transformation and adjustment layers are not supported. To load all layers in a PSD file with appropriate blend modes, use File > Import > PSD.

Common Controls

Settings Tab

The Settings tab controls are common to both Loader and Saver nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

MediaIn Node [MI]

The MediaIn node

MediaIn Node Introduction

> **NOTE:** The MediaIn node is only available in DaVinci Resolve.

The MediaIn node is the foundation of every composition you create in DaVinci Resolve's Fusion page. In most cases, it replaces the Loader node used in Fusion Studio for importing clips. There are four ways to add a MediaIn node to the Node Editor.

— In the Edit or Cut page, position the playhead over a clip in the Timeline, and then click the Fusion page button. The clip from the Edit or Cut page Timeline is represented as a MediaIn node in the Node Editor.

— Drag clips from the Media Pool into the Node Editor, creating a MediaIn node in the Node Editor.

— Drag clips from an OS window directly into the Node Editor, creating a MediaIn node in the Node Editor.

— Choose Fusion > Import> PSD when importing PSD files into the Node Editor. Each PSD layer is imported as a separate MediaIn node.

> **NOTE:** Although a MediaIn tool is located in the I/O section of the Effects Library, it is not used as a method to import clips.

When clips are brought in from the Media Pool, dragged from the OS window, or via the Import PSD menu option, you can use the MediaIn node's Inspector for trimming, looping, and extending the footage, as well as setting the source's color and gamma space.

Inputs

The single input on the MediaIn node is for an effect mask to crop the image brought in by the MediaIn.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the source image to appear only within the mask. An effects mask is applied to the tool after the tool is processed.
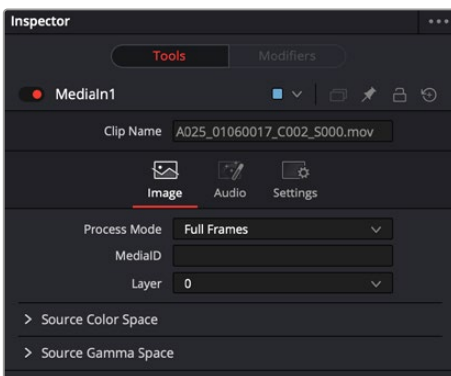
Basic Node Setup

The MediaIn node is typically the starting point for all composites done in the Fusion page of DaVinci Resolve. It contains the clip from the Edit page or Cut page. Any clip brought in from the Media Pool is also added as a MediaIn node.
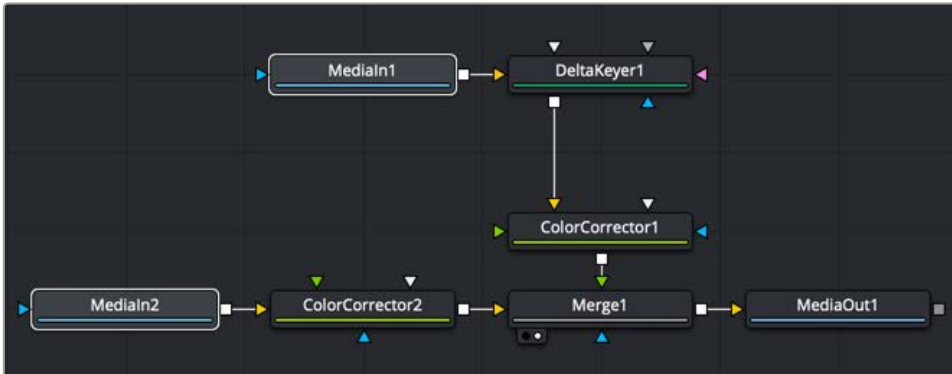
Two MediaIn nodes: one from the Edit page Timeline and one from the Media Pool

Inspector

MediaIn node Image tab

Image Tab

When brought in from the Media Pool or dragged from an OS window, the MediaIn node's Image tab includes controls for trimming, creating a freeze frame, looping, and reversing the clip. You can also reselect the clip the MediaIn links to on your hard drive. A subset of these controls is available when the MediaIn node is brought in from the Edit or Cut page Timeline.

Global In and Out

Only used when a clip is brought in through the Media Pool or an OS window, the Global In and Out handles are used to specify the start and end of this node within the Fusion effect. Use Global In to specify the frame on which the clip starts and use Global Out to specify the frame on which the clip ends within the comp's global range. The node does not produce an image on frames outside of this range.

If the Global In and Out values are decreased to the point where the range between the In and Out values is smaller than the number of available frames in the clip. Fusion automatically trims the clip by adjusting the Trim range control. If the Global In/Out values are increased to the point where the

range between the In and Out values is larger than the number of available frames in the clip, Fusion automatically lengthens the clip by adjusting the Hold First/Last Frame controls. Extended frames are visually represented in the range control by changing the color of the held frames to green in the control.

To slide the clip in time or move it through the project without changing its length, place the mouse pointer in the middle of the range control and drag it to the new location, or enter the value manually in the Global In value box.

Process Mode

Use this menu to select the Fields Processing mode used by Fusion when loading the image. The Has Fields checkbox control in the Frame Format preferences determines the default option, and the default height as well. Available options include:

— Full frames
— NTSC fields
— PAL/HD fields
— PAL/HD fields (reversed)
— NTSC fields (reversed).

The two reversed options load fields in the opposite order and thus result in the fields being spatially swapped both in time order and in vertical order as well.

Media Source

Selects where the media is linked from, allowing you to access Edit track composite results.

— **Timeline:** Links the clip from the Edit Timeline (default).
— **Background:** In cases where the clip is composited over another clip in the Edit Timeline, the background clip is loaded instead.
— **Media Pool:** Links the clip directly from the Media Pool, bypassing the Edit Timeline..

Layer

Used to identify the layer in a PSD file or compound clip. When a PSD file is brought in from the Media Pool, the drop-down menu allows you to select an individual layer for output instead of the entire PSD composite.

Trim

The Trim range control is used to trim frames from the start or end of a clip. Adjust the Trim In to remove frames from the start and adjust Trim Out to specify the last frame of the clip. The values used here are offsets. A value of 5 in Trim In would use the fifth frame in the sequence as the start, ignoring the first four frames. A value of 95 would stop loading frames after the 95th frame.

Hold First Frame/Hold Last Frame:

The Hold First Frame and Hold Last Frame controls hold the first or last frame of the clip for the specified amount of frames. Held frames are included in a loop if the footage is looped.

Reverse

Select this checkbox to reverse the footage so that the last frame is played first, and the first frame is played last.

Loop

Select this checkbox to loop the footage until the end of the project. Any lengthening of the clip using Hold First/Last Frame or shortening using Trim In/Out is included in the looped clip.

Source Color Space

Lets you choose a color space for the image data output by this MediaIn node.

— **Auto:** uses the Timeline color space, or whichever color space is assigned by Resolve Color Management (RCM) if it's enabled.

— **Space:** Space lets you choose a specific setting from a Color Space drop-down menu, while a visual "horseshoe" graph lets you see a representation of the color space you've selected.

Source Gamma Space

Lets you choose a gamma setting for the image data output by this MediaIn node. Once the gamma curve type is set, you can choose to remove the curve to help achieve a linear workflow.

— **Auto:** Uses the Timeline gamma, or whichever gamma is assigned by Resolve Color Management (RCM) if it's enabled.

— **Space:** Lets you choose a specific setting from a Gamma Space drop-down menu, while a visual graph lets you see a representation of the gamma setting you've selected.

— **Log:** Displays the Log Type drop-down menu where you can choose a specific log encoding profile. A visual graph shows a representation of the log setting chosen from the menu. Additional Lock RGB, Level, Soft Clip, Film Stock Gamma, Conversion Gamma, and Conversion table options are presented to finesse the gamma output.

— **Remove Curve:** The associated gamma curve is removed from, or a log-lin conversion is performed on, the material, effectively converting it to a linear output space.

— **Pre-Divide/Post-Multiply:** Lets you convert "straight" Alpha channels into premultiplied Alpha channels, when necessary.
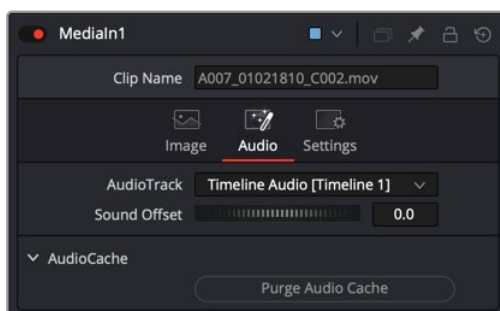
The Audio tab in the MediaIn node is used to select the track for playback, slip the audio timing and reset the audio cache.

Audio Tab

The Inspector for the MediaIn node contains an Audio tab, where you can choose to solo the audio from the clip or hear all the audio tracks in the Timeline.

If the audio is out of sync when playing back in Fusion, the Audio tab's Sound Offset wheel allows you to slip the audio in subframe frame increments. The slipped audio is only modified in the Fusion page. All other pages retain the original audio placement.

Audio with Media Pool Clips

Audio from a clip brought in through the Media Pool is muted by default. Hearing the audio from a Media Pool clip is a two step process.

To hear audio from a clip brought in through the Media Pool, do the following:

1 Select the clip in the Node Editor.

2 In the Inspector, click the Audio tab and select the clip name from the Audio Track drop-down menu.

If more than one MediaIn node exists in the comp, the audio last selected in the Inspector is heard. You can use the Speaker icon in the toolbar to switch between the MediaIn node audio files.

3 Right-click the Speaker icon in the toolbar, then choose the MediaIn for the clip you want to hear.

Purging the Audio Cache

The audio and its settings are cached for faster performance. If you change which audio tracks you want to play back in Fusion, or you use the Sound Offset wheel to slip the audio tracks, you need to purge the audio cache. Also, if you return to the Edit, Cut, or Fairlight page and modify the audio levels, you need to purge the audio cache.

To purge the audio cache after any change to the audio playback:

— Click the Purge Audio Cache button in the Inspector.

The audio will be updated when you next playback the composition.

MediaOut Node [MO]
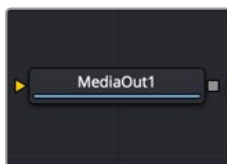
The MediaOut node

> **NOTE:** The MediaOut node is only available in DaVinci Resolve.

MediaOut Node Introduction

Every composition you create in DaVinci Resolve's Fusion page must include a MediaOut node. The MediaOut node sends the final output back to your Timeline on DaVinci Resolve's Edit or Cut page. In most cases, it replaces the Saver node used in Fusion Studio for exporting clips.

The composition output by the Fusion page's MediaOut node is propagated via the Color page's source inputs, with the sole exception that if you've performed transforms or added plugins to that clip in the Edit or Cut page, then the handoff from the Fusion page to the Color page is as follows:

Fusion Effects ▸ Edit Page Plugins ▸ Color Effects

When using Resolve Color Management or ACES, each MediaOut node converts the output image back to the Timeline color space for handoff to the Color page.

> **NOTE:** Additional MediaOut nodes can be added to the Node Editor from the Effects Library. Additional MediaOut nodes are used to pass mattes to the Color page.

Inputs

The single input on the MediaOut node is where you connect the final composite image you want rendered back into the Edit page.

— **Input:** The orange input is a required input. It accepts any 2D image that you want rendered back into the Edit page.

Basic Node Setup

Multiple MediaOut nodes can exist in a single comp. The first MediaOut node always renders the image back to the Edit page Timeline. Additional MediaOut nodes can be used to pass mattes to the Color page.
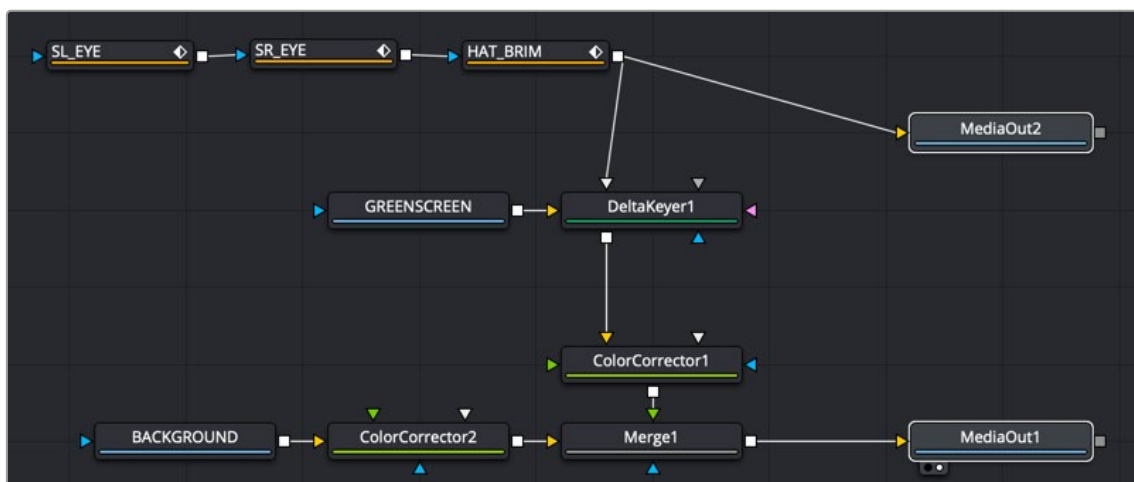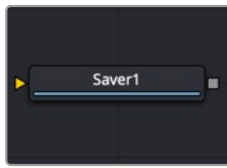

MediaOut1 node rendering to the Edit page, and MediaOut2 sending mattes to the Color page

Saver Node [Sv]

The Saver node

> **NOTE:** The Saver node in DaVinci Resolve is only used for exporting EXR files.

Saver Node Introduction

The Saver node represents the final composition output from Fusion Studio. It is used to render out movie files or sequential images but can be inserted into a composition at any point to render out intermediate stages of a composition. A composition can contain any number of Saver nodes for rendering different branches of a comp as well as different formats.

The Saver node can also be used to add scratch track audio to your composition, which can be heard during interactive playback.

Inputs

The single input on the Saver node is for the final composition you want to render.

— **Image Input:** The orange input is used to connect the resulting image you want rendered.
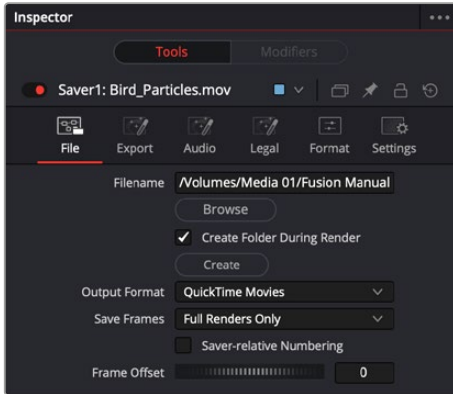
Basic Node Setup

The Saver node is placed at the end of the composition. Multiple Savers can be placed in a comp to render different formats or to render different parts of a composition.

Saver node added to the end of a node tree to render the composition

Inspector

Saver File tab

File Tab

The Saver File tab is used to set the location and output format for the rendered file.

Filename

The Filename dialog is used to select the name and path of the rendered image output. Click on the Browse button to open a file browser and select a location for the output.

Sequence numbering is automatically added to the filename when rendering a sequential image file format. For example, if c\renders\image.exr is entered as the filename and 30 frames of output are rendered, the files are automatically numbered as image0000.tga, image0001.exr, image0003.exr… and so on. Four-digit padding is automatically used for numbers lower than 10000.

You can specify the number of digits to use for padding by explicitly entering the digits into the filename.

For example, image000000.exr would apply 6-digit padding to the numeric sequence, image.001.exr would use 3-digit padding, and image1.exr would use none.

Output Format

This menu is used to select the image format to be saved. Be aware that selecting a new format from this menu does not change the extension used in the filename to match. Modify the filename manually to match the expected extension for that format to avoid a mismatch between name and image format.

Save Frames

This control selects between two modes of rendering: Full Renders Only or High Quality Interactive.

— **Full Renders Only:** This is the common setting for most situations. Images are saved to disk when a final render is started using the Start Render button in the Time Ruler.

— **High Quality Interactive:** This render mode is designed for real-time rendering when painting and rotoscoping. Fusion saves each frame to disk as it is processed interactively. When used correctly, this feature can eliminate the need to perform a final render after rotoscoping.

> **NOTE:** The High Quality Interactive setting can easily cause confusion when used in conjunction with a node tree that contains spline-animated parameters. If these splines are modified in such a way that frames already saved interactively are changed, the frames already on the disk do not automatically re-render. Either step through each frame again or perform a final render to make certain that the result is correct

Frame Offset

This thumbwheel control can be used to set an explicit start frame for the number sequence applied to the rendered filenames. For example, if Global Start is set to 1 and frames 1-30 are rendered, files are normally numbered 0001-0030. If the Sequence Start Frame is set to 100, the rendered output would be numbered from 100-131.
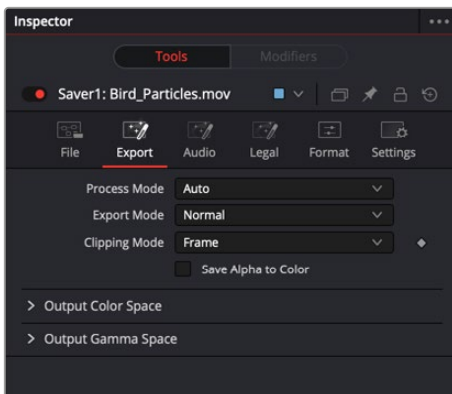
Saver Export tab

Export Tab

Process Mode

Use this menu to select the Fields Processing mode used by Fusion when saving the images or movie file to disk. The Has Fields checkbox control in the Frame Format preferences determines the default option, and the default height as well. Available options include:

— Full frames
— NTSC fields
— PAL/HD fields
— PAL/HD fields (reversed)
— NTSC fields (reversed).

The two reversed options save fields in the opposite order and thus result in the fields being spatially swapped both in time order and in vertical order as well.

Export Mode

This menu is used to render out the file normally or apply a SMPTE standard 3:2 pulldown to the footage, converting the footage from 24 fps to 30 fps.

Clipping Mode

This menu, sometimes considered source image clipping, defines how the edges of the image should be treated.

— **Frame:** The default Frame setting clips to the parts of the image visible within its visible dimensions. It breaks any infinite-workspace behavior. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **None:** This setting does not perform any source image clipping at all. This means that any data that would normally be needed outside the upstream DoD is treated as black/transparent. Be aware that this might create humongous images that can consume a considerate amount of disk space. So you should use this option only if really needed.

For more information about ROI, DoD, and Infinite Workspace, *see Chapter 69, "Using Viewers."* in the DaVinci Resolve Reference Manual, or Chapter 7 in the Fusion Reference Manual.

Save Alpha to Color

When enabled, this control causes the Alpha channel to be saved into the color channels as a grayscale image. This completely overwrites any existing color information.

Color Space Type

This menu is used to set the Color Space of the output file so you could, for example, color space convert images from linear to Rec709, etc., thereby delivering linear EXRs, 709 Quicktimes, etc. from a single source. The images are not being converted in the Comp, only in the images saved to disk are converted.

— **Auto:** Passes along any metadata that might be in the rendered image.

— **Space:** Allows the user to set the color space based on the output format.

Curve Type

This menu is used to select a Gamma curve of the rendered file. Once the gamma curve type is set, you can choose to apply the curve for output.

— **Auto:** Passes along any metadata that might be in the incoming image.

— **Space:** Allows the user to set the gamma curve based on the selected file format.

— **Log:** Displays the Log/Lin settings, similar to the Cineon Log node. For more detail on the Log settings, *see Chapter 98, "Film Nodes."* in the DaVinci Resolve Reference Manual, or Chapter 36 in the Fusion Reference Manual.

Apply Curve

Depending on the selected Curve Type or on the Gamma Space found in Auto mode, the associated Gamma Curve is applied, effectively converting from a linear working space.

Audio Tab
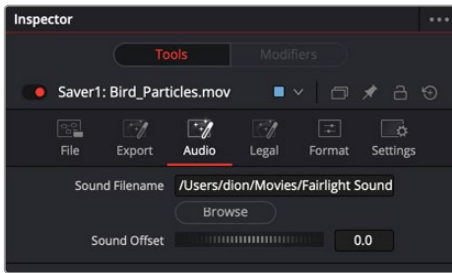
Saver Audio tab

NOTE: This tab is only available in Fusion Studio's Saver node.

The audio functionality is included in Fusion Studio for scratch track (aligning effects to audio and clip timing) purposes only. Final renders should almost always be performed without audio. The smallest possible audio files should be used, as Fusion loads the entire audio file into memory for efficient display of the waveform in the Timeline. The audio track is included in the saved image if a Quicktime file format is selected. Fusion currently supports playback of WAV audio.

Source Filename

You can enter the file path and name of the audio clip you want to use in the Source Filename field. You can also click the Browse button to open a file browser window and locate the audio scratch track. Select the WAV file of choice, and then in the keyframes panel expand the Saver bar to view the audio waveform. Drag the pointer over the audio wave in the Timeline layout to hear the track.

Sound Offset

Drag the control left or right to slide the Timeline position of the audio clip, relative to other nodes in the Node Editor.

Legal Tab

The Legal tab includes settings for creating "broadcast safe" saturation and video range files for output.
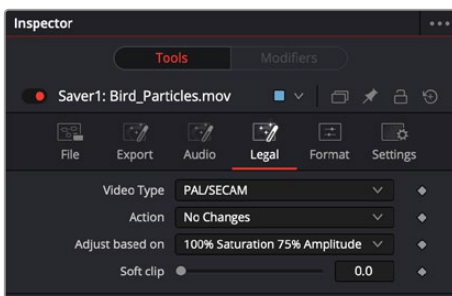
Saver Legal tab

Video Type

Use this menu to select the standard to be used for broadcast legal color correction. NTSC, NHK, or PAL/SECAM can be chosen.

Action

Use this menu to choose how Fusion treats illegal colors in the image.

— **Adjust to Legal:** This causes the images to be saved with legal colors relevant to the Video Type selected.

— **Indicate as Black:** This causes the illegal colors to be displayed as black in the views.

— **Indicate as White:** This causes the illegal colors to be displayed as white in the views.

— **No Changes:** This causes the images to be saved unaffected.

Adjust Based On

This menu is used to choose whether Fusion makes legal the image to 75% or 100% amplitude. Very few broadcast markets permit 100% amplitude, but for the most part this should be left to 75%.

Soft Clip

The Soft Clip control is used to draw values that are out of range back into the image. This is done by smoothing the conversion curve at the top and bottom of the curve, allowing more values to be represented.

Format Tab
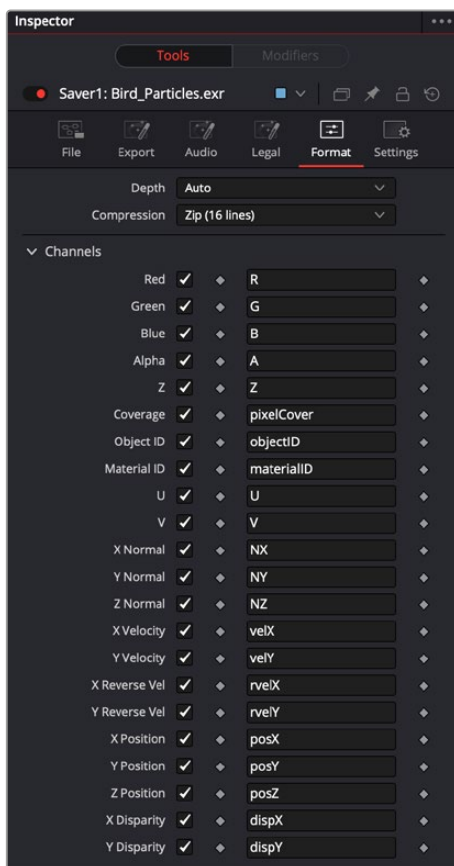
Saver Format tab

The Format tab contains information, options, and settings specific to the image format being saved. The controls for an EXR sequence is entirely different from the ones displayed when a MOV file is saved.

EXR is displayed above for reference.

When the Saver node is set to DPX, it's important to understand the reason for the Bypass Conversion > Data is Linear option. When saving log data into a DPX, and not using the Saver's node's own lin-log conversion (that is, Bypass Conversion is checked), the Data Is Linear option should be off. This indicates whether the reason for checking Bypass Conversion is because the data is linear, or whether it's already log.

If Data Is Linear is enabled, then the DPX is marked in its header as containing linear data. In turn, that means that when the DPX is loaded back into Fusion, or into other apps that evaluate the header, those apps think the data is linear and do not perform any log-lin conversion.

Common Controls

Settings Tab

The Settings tab controls are common to both Loader and Saver nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Clipping Mode

This menu, sometimes considered source image clipping, defines how the edges of the image should be treated.

— **Frame:** The default Frame setting clips to the parts of the image visible within its visible dimensions. It breaks any infinite-workspace behavior. If the upstream DoD is smaller than the frame, the remaining areas in the frame are treated as black/transparent.

— **None:** This setting does not perform any source image clipping at all. This means that any data that would normally be needed outside the upstream DoD is treated as black/transparent. Be aware that this might create humongous images which can consume a considerable amount of disk space. So you should use this option only if really needed.

For more information about ROI, DoD, and Infinite Workspace, *see Chapter 69, "Using Viewers."* in the DaVinci Resolve Reference Manual, or Chapter 7 in the Fusion Reference Manual.

Save Alpha to Color

When enabled, this control causes the Alpha channel to be saved into the color channels as a grayscale image. This completely overwrites any existing color information.

Color Space Type

This menu is used to set the Color Space of the output file so you could, for example, color space convert images from linear to Rec709, etc., thereby delivering linear EXRs, 709 Quicktimes, etc. from a single source. The images are not being converted in the Comp, only in the images saved to disk are converted.

— **Auto:** Passes along any metadata that might be in the rendered image.
— **Space:** Allows the user to set the color space based on the output format.

Curve Type

This menu is used to select a Gamma curve of the rendered file. Once the gamma curve type is set, you can choose to apply the curve for output.

— **Auto:** Passes along any metadata that might be in the incoming image.

— **Space:** Allows the user to set the gamma curve based on the selected file format.

— **Log:** Displays the Log/Lin settings, similar to the Cineon Log node. For more detail on the Log settings, *see Chapter 98, "Film Nodes."* in the DaVinci Resolve Reference Manual, or Chapter 36 in the Fusion Reference Manual.
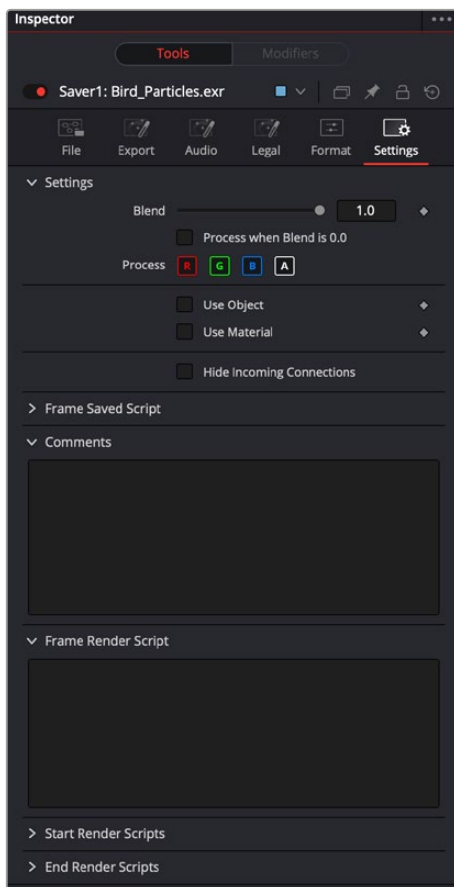
Apply Curve

Depending on the selected Curve Type or on the Gamma Space found in Auto mode, the associated Gamma Curve is applied, effectively converting from a linear working space.

The Common Controls

I/O nodes share a number of identical controls in the Inspector. This section describes controls that are common among I/O nodes.

Inspector

Common Saver settings inspector

Settings Tab

The Settings tab in the Inspector can be found on the Loader, Saver, MediaIn, and MediaOut nodes. The controls are consistent and work the same way for each tool, although some parameters are only available on individual nodes but are covered here.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this causes the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if processing of this node is scripted to trigger another task, but the value of the node is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the red button on a Blur tool is deselected, the blur is first applied to the image, and then the red channel from the original input is copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option causes the RGB values of the masked image to be multiplied by the mask channel's values. This causes all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels are used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID is used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the view. The image or sequence must have been rendered from a 3D software package with those channels included.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node are displayed in the Inspector. Dragging a connected node from the node tree into the field hides that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line reappears.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

LUT Nodes

This chapter details the LUT nodes available in Fusion.

The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn/MediaOut nodes in DaVinci Resolve are interchangeable with Loader/Saver nodes in Fusion Studio, unless otherwise noted.

Contents

File LUT [FLU]

The File LUT node

File LUT Node Introduction

The File LUT node applies a Lookup table (LUT) to the image: either a simple 1D LUT or a supported 3D LUT. Unlike the Color Curves node, it does not use a spline-based LUT. Instead, it loads the LUT from a file stored on your computer or server.

This approach has two advantages. The first is that the only part of the LUT stored in the composition is the path to the file. Since LUT files can be large, this can dramatically reduce the file size of a composition when several LUTs are present. The second advantage is that it becomes possible to adjust all File LUT nodes using the same file at the same time, just by changing the contents of the LUT. This can be useful when the same LUT-based color correction is applied in many different compositions.

Inputs

The File LUT node includes two inputs: one for the main image and the other for an effect mask to limit the area where the LUT is applied.

— **Input:** This orange input is the only required connection. It accepts a 2D image output that gets the LUT applied.
— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the applied LUT to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The File LUT node can be placed after a MediaIn node in DaVinci Resolve or a Loader node in Fusion Studio. Sometimes this setup is used to convert the camera-original image to linear color space for compositing. Other times, as in the example below, the File LUT is applied at the end as a grading LUT to apply a look from the colorist.

A File LUT node applied at the end of a node tree as a colorist's look

Inspector
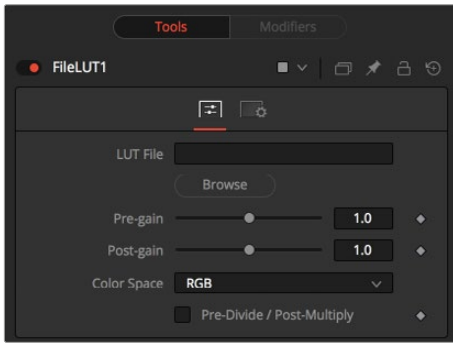
File LUT controls

Controls Tab

The Controls tab includes options for loading a LUT and making adjustments to the gain, color space, and Alpha channel, if one exists.

LUT File

This field is used to enter the path to the LUT file. Clicking the Browse button opens a file browser window to locate the LUT file instead of entering it manually into the LUT File field. Currently, this node supports LUTs exported from Fusion in .LUT and .ALUT formats, DaVinci Resolve's .CUBE format, and several 3D LUT formats. The node fails with an error message on the Console if it is unable to find or load the specified file.

Pre-Gain:

This slider is a gain adjustment before the LUT being applied. This can be useful for pulling in highlights before the LUT clips them.

Post-Gain

This slider is a gain adjustment after the LUT is applied.

Color Space

This menu is used to change the color space the LUT is applied in. The default is to apply the curves described in the LUT to the RGB color space, but options for YUV, HLS, HSV, and others are also available.

Pre-Divide/Post-Multiply

Selecting the Pre-Divide/Post-Multiply checkbox causes the image pixel values to be divided by the Alpha values before applying the LUT, and then re-multiplied by the Alpha value after the correction.

This helps to prevent the creation of illegally additive images, particularly around the edges of a blue/green key or when working with 3D-rendered objects.

Settings Tab

The Settings tab in the Inspector is also duplicated in other LUT nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

LUT Cube Analyzer [LCA]

The LUT Cube Analyzer node

LUT Cube Analyzer Node Introduction

The LUT Cube Analyzer takes an image originated by the LUT Cube Creator as an input and allows the user to create a 3D LUT file in ALUT3, ITX, or 3DL format.

Feeding the original LUT Cube Creator image into the node results in an unaltered, or 1:1, LUT file, and nothing is displayed in the viewer.

You can, however, modify, grade, and color correct the original cube image with as many nodes as you like and feed the result into the LUT Cube Analyzer. This creates a LUT that exactly resembles your color pipeline.

Inputs

The LUT Cube Analyzer includes a single orange input.

— **Input:** The orange input is used to take the output of any node modifying an image that originated with the LUT Cube Creator.
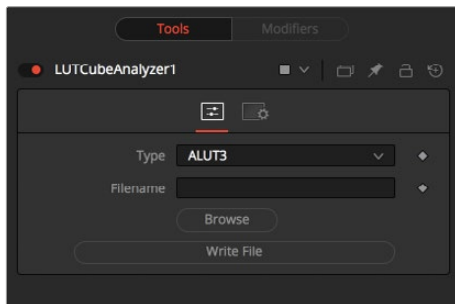
Basic Node Setup

The example below shows a node tree starting with a LUT Cube Creator node and going through two color adjustments. The adjusted image is then connected to a LUT Cube Analyzer, which generates the LUT file.

Generating a LUT starts with the LUT Cube Creator and ends with a LUT Cube Analyzer.

Inspector

LUT Cube Analyzer controls

Controls Tab

The Controls tab for the LUT Cube Analyzer node is used to select the desired LUT output format, specify a filename, and write the 3D LUT to disk.

Type

Select the desired output format of the 3D LUT.

Filename

Enter the path where you want the file saved and enter the name of the LUT file. Alternatively, you can click the Browse button to open a file browser to select the location and filename.

Write File

Press this button to generate the 3D LUT file based on the settings above.

Settings Tab

The Settings tab in the Inspector is also duplicated in other LUT nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

LUT Cube Apply [LCP]
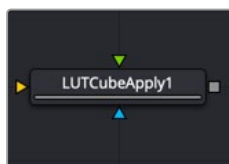
The LUT Cube Apply node

LUT Cube Apply Node Introduction

The LUT Cube Apply node takes an image created by the LUT Cube Creator as the foreground input and applies that LUT to the image connected to the background input.

Feeding the original image into the node would result in an unaltered, or 1:1, output.

You can, however, modify, grade, and color correct the original cube image with as many nodes as you like and feed the result into the LUT Cube Apply. Or, take a LUT image that has been graded beforehand to apply the LUT without having to write an actual 3D LUT using the LUT Cube Analyzer.

Inputs

The LUT Cube Apply has three inputs: a green input where the output of the LUT Cube Creator is connected, an orange input for the image to have the LUT applied, and a blue effect mask input

— **Input:** This orange input accepts a 2D image that gets the LUT applied.
— **Reference Image:** The green input is used to connect the output of the LUT Cube Creator or a node that is modifying the image originating in the LUT Cube Creator.
— **Effect Mask:** The optional effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the LUT Cube Apply to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The example below shows a node tree starting with a LUT Cube Creator node and going through two color adjustments. The adjusted image is then connected to the green Reference input of the LUT Cube Apply. The image you want to apply the LUT to is connected to the orange Input.
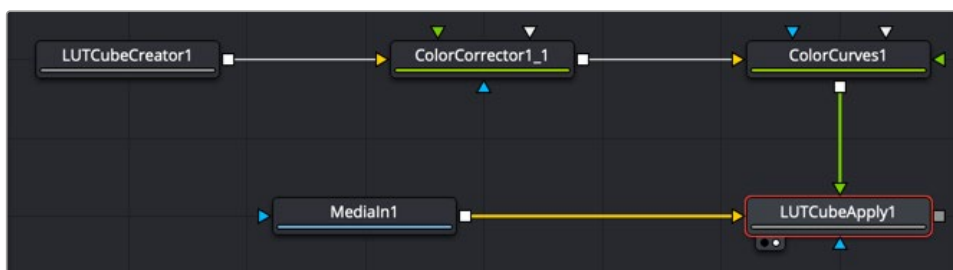
The LUT generated by the LUT Cube Creator is applied to an image using the LUT Cube Apply node.

Inspector

There are no controls for the LUT Cube Apply node. The LUT connected to the green foreground input is applied to the image connected to the orange background input without having to write an actual 3D LUT using the LUT Cube Analyzer.

Settings Tab

The Settings tab in the Inspector is also duplicated in other LUT nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

LUT Cube Creator [LCC]

The LUT Cube Creator node

LUT Cube Creator Node Introduction

The LUT Cube Creator generates an image for further use with the LUT Cube Analyzer or LUT Cube Apply nodes. The output can be graded, color corrected, or modified with any tool inside and outside of Fusion. If working outside Fusion, make sure to keep the image in 32-bit floating point to preserve color accuracy.

Feeding the original LUT Cube Creator image into the LUT Cube Analyzer node results in an unaltered, or 1:1, LUT file, and nothing is displayed in the viewer.

Inputs

There are no inputs on the LUT Cube Creator. The purpose of the node is to generate an image that can be used to create a LUT.

Basic Node Setup

The example below shows a node tree starting with a LUT Cube Creator node and going through two color adjustments. The adjusted image is then connected into a LUT Cube Analyzer that generates the LUT file.

Generating a LUT starts with the LUT Cube Creator and ends with a LUT Cube Analyzer.

Inspector
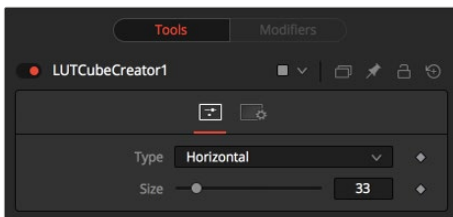
LUT Cube Creator controls

Controls Tab

The Controls tab creates a test pattern of sorts used to create a 3D LUT. The controls here determine the complexity of the pattern used to create a LUT using the LUT Cube Analyzer.

Type:

The Type menu is used to create a pattern of color cubes.

— **Horizontal:** Creates a long, horizontal strip representing a color cube.

— **Vertical:** Creates a long, vertical strip representing a color cube.

— **Rect:** Creates a rectangular image, as depicted below, representing a color cube.
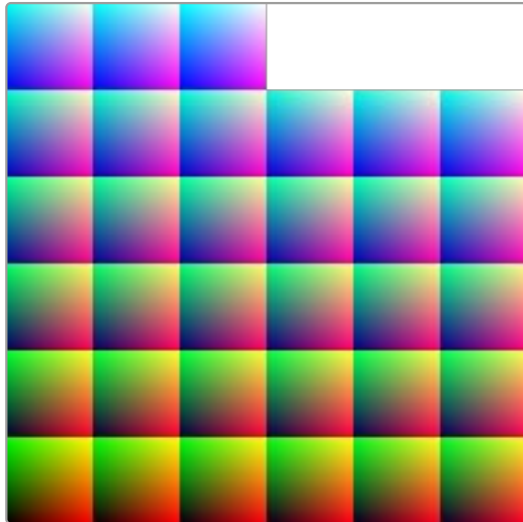
A Cube image created with the Rect type
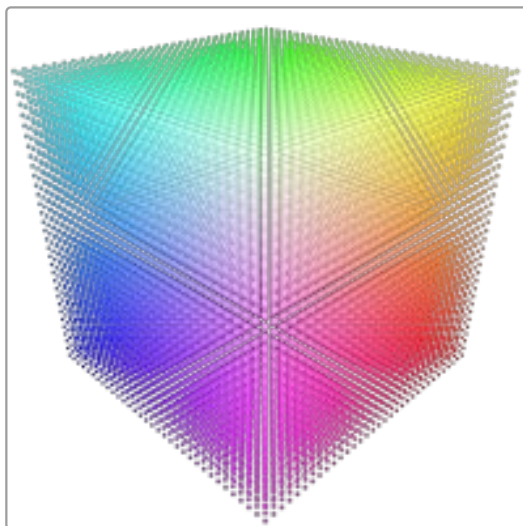
The resulting color cube

Size

Determines the resolution of the color cube.

Typical Size settings for color cubes are 33 (33 x 33 x 33) or 65 (65 x 65 x 65). These numbers are the samples on each side of the cube. A 33 x 33 x 33 cube has around 35,937 color samples.

> **NOTE:** Higher resolutions yield more accurate results but are also more memory and computationally expensive.

Settings Tab

The Settings tab in the Inspector is also duplicated in other LUT nodes. These common controls are described in the following "The Common Controls" section.

The Common Controls

LUT nodes share a number of identical controls in the Inspector. This section describes controls that are common among LUT nodes.

Inspector

Common LUT Settings Inspector

Settings Tab

The Settings tab in the Inspector can be found on every tool in the LUT category. The controls are consistent and work the same way for each tool, although some tools do include one or two individual options, which are also covered here.

Use GPU

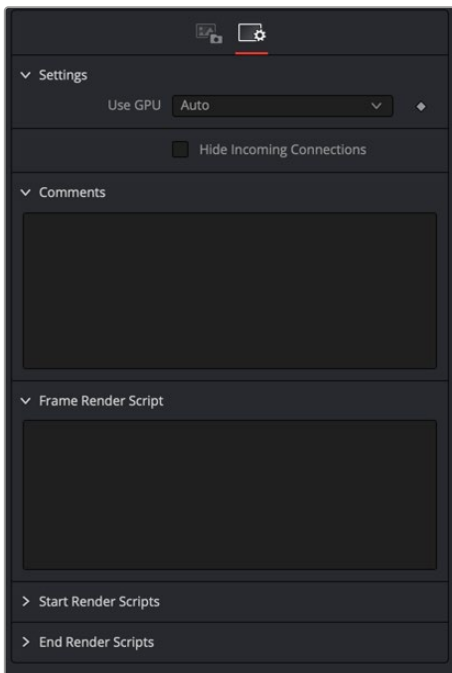The Use GPU menu has three settings. Setting the menu to Disable turns off hardware-accelerated rendering using the graphics card in your computer. Enabled uses the hardware. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node are displayed in the Inspector. Dragging a connected node from the node tree into the field hides that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line reappears.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Mask Nodes

This chapter details the Mask nodes available in Fusion.

The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Bitmap Mask [Bmp]

The Bitmap node

Bitmap Mask Node Introduction

The Bitmap mask node allows images from the node tree to act as masks for nodes and effects. Bitmap masks can be based on values from any of the color, Alpha, hue, saturation, luminance, and auxiliary coverage channels of the image. Nodes can also be masked based on the Object ID or Material ID of a 3D-rendered image (provided those channels were included when the file was rendered).

The Bitmap mask node is not required for effect masks. For effects masks, the Common Settings tab for the masked node displays controls to select which channel of the mask image is used to create the mask.

However, Bitmap mask nodes may still be required to connect to other mask inputs on some nodes, such as Garbage Mattes and Pre-Masks. Also, using a Bitmap mask node between the mask source and the target node provides additional options that would not be available when connecting directly, such as combining masks, blurring the mask, or clipping its threshold.

Inputs

The Bitmap mask node includes two inputs in the Node Editor.

— **Input:** The orange input accepts a 2D image from which the mask will be created.
— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input combines the masks. How masks are combined is handled in the Paint mode menu in the Inspector.

Basic Node Setup

The Bitmap mask node is not required for connecting an image into the effect mask input, but it does provide options that are otherwise unavailable. It allows for selecting channels other than RGBA for the mask, as well as softness and clipping. In the node tree below, the two Bitmap masks are combined using a Paint menu located in the second Bitmap mask node, which allows you to add, subtract, multiply, and perform other operations on the combined mask.
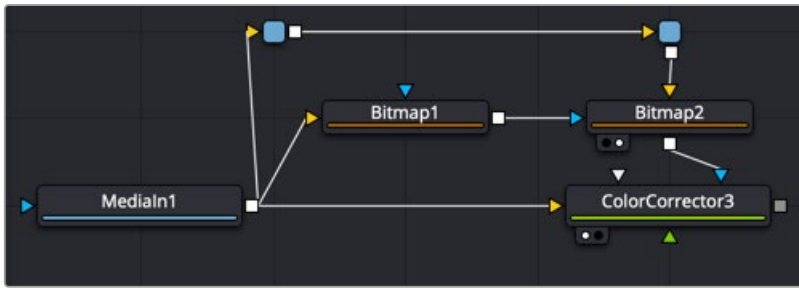
Bitmap nodes can be chained together for more advanced matte operations.

Inspector
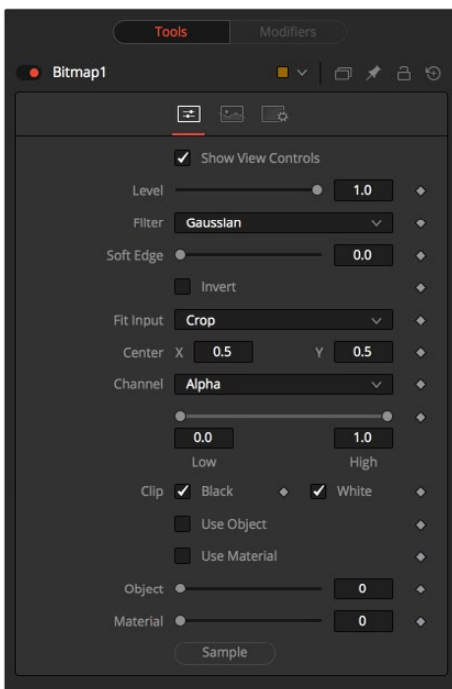
Bitmap Mask controls

Controls Tab

The Controls tab is used to refine how the image connected to the orange input converts into the Bitmap mask.

Show View Controls

The Show View Controls checkbox is used to enable/disable the display of the mask's onscreen controls in the viewer. Onscreen controls, including center position, polylines, angles, and others, do not appear when this checkbox is disabled, even when the node is selected.

Level

The Level control sets the transparency level of the pixels in the mask channel. When the value is 1.0, the mask is completely opaque (unless it has a soft edge). Lower values cause the mask to be partially transparent. The result is identical to lowering the Blend control of an effect.

> **NOTE:** Lowering the level of a mask lowers the values of all pixels covered by the mask in the mask channel. For example, if a Circle mask is placed over a Rectangle mask, lowering the level of the Circle mask lowers the values of all the pixels in the mask channel, even though the Rectangle mask beneath it is still opaque.

Filter

This control selects the filtering algorithm used when applying Soft Edge to the mask.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.

— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.

— **Multi-box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."

— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Soft Edge

Use the Soft Edge slider to blur (feather) the mask, using the selected filter. Higher values cause the edge to fade off well beyond the boundaries of the mask. A value of 0.0 creates a crisp, well-defined edge.

Paint Mode

Connecting a mask to the effect mask input displays the Paint mode menu. The Paint mode is used to determine how the incoming mask for the effect mask input and the mask created in the node are combined.

— **Merge:** Merge is the default for all masks. The new mask is merged with the input mask.

— **Add:** The mask's values add to the input mask's values.

— **Subtract:** In the intersecting areas, the new mask values subtract from the input mask's values.

— **Minimum:** Comparing the input mask's values and the new mask, this displays the lowest (minimum) value.

— **Maximum:** Comparing the input mask's values and the new mask, this displays the highest (maximum) value.

— **Average:** This calculates the average (half the sum) of the new mask and the input mask.

— **Multiply:** This multiplies the values of the input mask by the new mask's values.

— **Replace:** The new mask completely replaces the input mask wherever they intersect. Areas that are zero (completely black) in the new mask do not affect the input mask.

— **Invert:** Areas of the input mask that are covered by the new mask are inverted; white becomes black and vice versa. Gray areas in the new mask are partially inverted.

— **Copy:** This mode completely discards the input mask and uses the new mask for all values.

— **Ignore:** This mode completely discards the new mask and uses the input mask for all values.

Invert

Selecting this checkbox inverts the entire mask. Unlike the Invert Paint mode, this checkbox affects all pixels, regardless of whether the new mask covers them.

Fit Input

This menu is used to select how the image source is treated if it does not fit the dimensions of the generated mask.

In the example below, a 720 x 576 image source (yellow) is used to generate a 1920 x 1080 mask (gray).

— **Crop:** If the image source is smaller than the generated mask, it will be placed according to the X/Y controls, masking off only a portion of the mask. If the image source is larger than the generated mask, it will be placed according to the X/Y controls and cropped off at the borders of the mask.

— **Stretch:** The image source will be stretched in X and Y to accommodate the full dimensions of the generated mask. This might lead to visible distortions of the image source.

— **Inside:** The image source will be scaled uniformly until one of its dimensions (X or Y) fits the inside dimensions of the mask. Depending on the relative dimensions of the image source and mask background, either the image source's width or height may be cropped to fit the respective dimensions of the mask.

— **Width:** The image source will be scaled uniformly until its width (X) fits the width of the mask. Depending on the relative dimensions of the image source and mask, the image source's Y dimension might not fit the mask's Y dimension, resulting in either cropping of the image source in Y or the image source not covering the mask's height entirely.

— **Height:** The image source will be scaled uniformly until its height (Y) fits the height of the mask. Depending on the relative dimensions of the image source and mask, the image source's X-dimension might not fit the mask's X-dimension, resulting in either cropping of the image source in X or the image source not covering the mask's width entirely.

— **Outside:** The image source will be scaled uniformly until one of its dimensions (X or Y) fits the outside dimensions of the mask. Depending on the relative dimensions of the image source and mask, either the image source's width or height may be cropped or not fit the respective dimension of the mask.

Center X and Y

These controls adjust the position of the Bitmap mask.

Channel

The Channel menu determines the Channel of the input image used to create the mask. Choices include the red, green, blue, and alpha channels, the hue, luminance, or saturation values, or the auxiliary coverage channel of the input image (if one is provided).

Threshold Low/High

The Threshold range control can be used to clip the bitmap image. Increasing the low range control will clip pixels below the specified value to black (0.0). Decreasing the high range control will force pixels higher than the specified value to white (1.0).

Use Object/Use Material

This control has no effect unless the input image contains a Material or Object ID channel. When toggled on, the Object ID and Material ID are used to create a mask based on the selected object or material. When toggled off, the regular color channels will generate the mask.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are also duplicated in other Mask nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

B-Spline Mask [BSp]
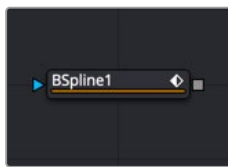
The B-Spline node

A B-Spline mask is identical to a Polygon mask in all respects except one. Where Polygon masks use Bézier splines, this mask node uses B-Splines. Where Bézier splines employ a central point and two handles to manage the smoothing of the spline segment, a B-Spline requires only a single point. This means that a B-Spline shape requires far fewer control points to create a nicely smoothed shape.

When first added to a node, the B-Spline mask consists of only Center control, which is visible onscreen. Points are added to the B-Spline by clicking in the viewer. Each new point is connected to the last one created, but instead of the spline going directly through each control point, B-Spline control points only influence the spline shape. The control point pulls the spline in its direction to create a smooth curve.

Like the Polygon mask tool, the B-Spline mask auto-animates. Adding this node to the Node Editor adds a keyframe to the current frame. Moving to a new frame and changing the shape creates a new keyframe and interpolate between the two defined shapes.

Inputs

The B-Spline mask node includes a single effect mask input.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input combines the masks. How masks are combined is handled in the Paint mode menu in the Inspector.

Basic Node Setup

The B-Spline node can be used to generate a single smooth spline shape or combined with other masks for more complex shapes. In the node tree below, the B-Spline mask is used to generate a smooth, curved shape as a garbage matte on the Delta Keyer.

A B-Spline node generates a smooth, curved shape as a garbage matte

Inspector

B-Spline Mask controls

Controls Tab

The Controls tab is used to refine how the B-Spline appears after drawing it in the viewer.

Show View Controls

The Show View Controls checkbox is used to enable/disable the display of the mask's onscreen controls in the viewer. Onscreen controls, including center position, polylines, angles, and others, do not appear when this checkbox is disabled, even when the node is selected.

Level

The Level control sets the transparency level of the pixels in the mask channel. When the value is 1.0, the mask is completely opaque (unless it has a soft edge). Lower values cause the mask to be partially transparent. The result is identical to lowering the blend control of an effect.

> **NOTE:** Lowering the level of a mask lowers the values of all pixels covered by the mask in the mask channel. For example, if a Circle mask is placed over a Rectangle mask, lowering the level of the Circle mask lowers the values of all of the pixels in the mask channel, even though the Rectangle mask beneath it is still opaque.

Filter

This control selects the filtering algorithm used when applying Soft Edge to the mask.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.

— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.

— **Multi-box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."

— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Soft Edge

Use the Soft Edge slider to blur (feather) the mask, using the selected filter. Higher values cause the edge to fade off well beyond the boundaries of the mask. A value of 0.0 creates a crisp, well-defined edge.

Border Width

The Border Width control adjusts the thickness of the mask's edge. When the solid checkbox is toggled on, the border thickens or narrows the mask. When the mask is not solid, the mask shape draws as an outline, and the width uses the Border Width setting.

Paint Mode

Connecting a mask to the effect mask input displays the Paint mode menu. The Paint mode is used to determine how the incoming mask for the effect mask input and the mask created in the node are combined.

— **Merge:** Merge is the default for all masks. The new mask is merged with the input mask.

— **Add:** The mask's values add to the input mask's values.

— **Subtract:** In the intersecting areas, the new mask values subtract from the input mask's values.

— **Minimum:** Comparing the input mask's values and the new mask, this displays the lowest (minimum) value.

— **Maximum:** Comparing the input mask's values and the new mask, this displays the highest (maximum) value.

— **Average:** This calculates the average (half the sum) of the new mask and the input mask.

— **Multiply:** This multiplies the values of the input mask by the new mask's values.

— **Replace:** The new mask completely replaces the input mask wherever they intersect. Areas that are zero (completely black) in the new mask do not affect the input mask.

— **Invert:** Areas of the input mask that are covered by the new mask are inverted; white becomes black and vice versa. Gray areas in the new mask are partially inverted.

— **Copy:** This mode completely discards the input mask and uses the new mask for all values.

— **Ignore:** This mode completely discards the new mask and uses the input mask for all values.

Invert

Selecting this checkbox inverts the entire mask. Unlike the Invert Paint mode, the checkbox affects all pixels, regardless of whether the new mask covers them or not.

Solid

When the Solid checkbox is enabled, the mask is filled to be transparent (white) unless inverted. When disabled, the spline is drawn as just an outline whose thickness is determined by the Border Width slider.

Center X and Y

These controls adjust the position of the B-Spline mask.

Size

Use the Size control to adjust the scale of the B-Spline effect mask, without affecting the relative behavior of the points that compose the mask or setting a keyframe in the mask animation.

X, Y, and Z Rotation

Use these three controls to adjust the rotation angle of the mask along any axis.

Fill Method

The Fill Method menu offers two different techniques for dealing with overlapping regions of a polyline. If overlapping segments in a mask are causing undesirable holes to appear, try switching the setting of this control from Alternate to Non Zero Winding.

Right-Click Here for Shape Animation

By default, all B-Spline masks are animated when they are created. The initial keyframe is set to the current time, and any changes to the shape at different times will create new keys.

Right-clicking on this label will display a contextual menu that offers options for removing or re-adding animation to the mask, or publishing and connecting the masks.

Adding Points

Adding Points to a B-Spline effect mask is relatively simple. Immediately after adding the node to the Node Editor, there are no points, but the tool will be in Click Append mode. Click once in the viewer wherever a point is required for the mask. Continue clicking to draw the shape of the mask.

When the shape is complete, click on the initial point again to close the mask.

When the shape is closed, the mode of the polyline changes to Insert and Modify. This allows you to add and adjust additional points on the mask by clicking the spline segments. To lock down the mask's shape and prevent accidental changes, switch the Polyline mode to Done using the Polyline toolbar or contextual menu.

Adjusting Tension on a B-Spline

The tension of the control point determines the smoothness of a B-Spline. To adjust the tension of a B-Spline's control points, select the point in the viewer, hold down the W key and drag the mouse pointer to the left and right to increase or decrease the tension of the curve through that point.
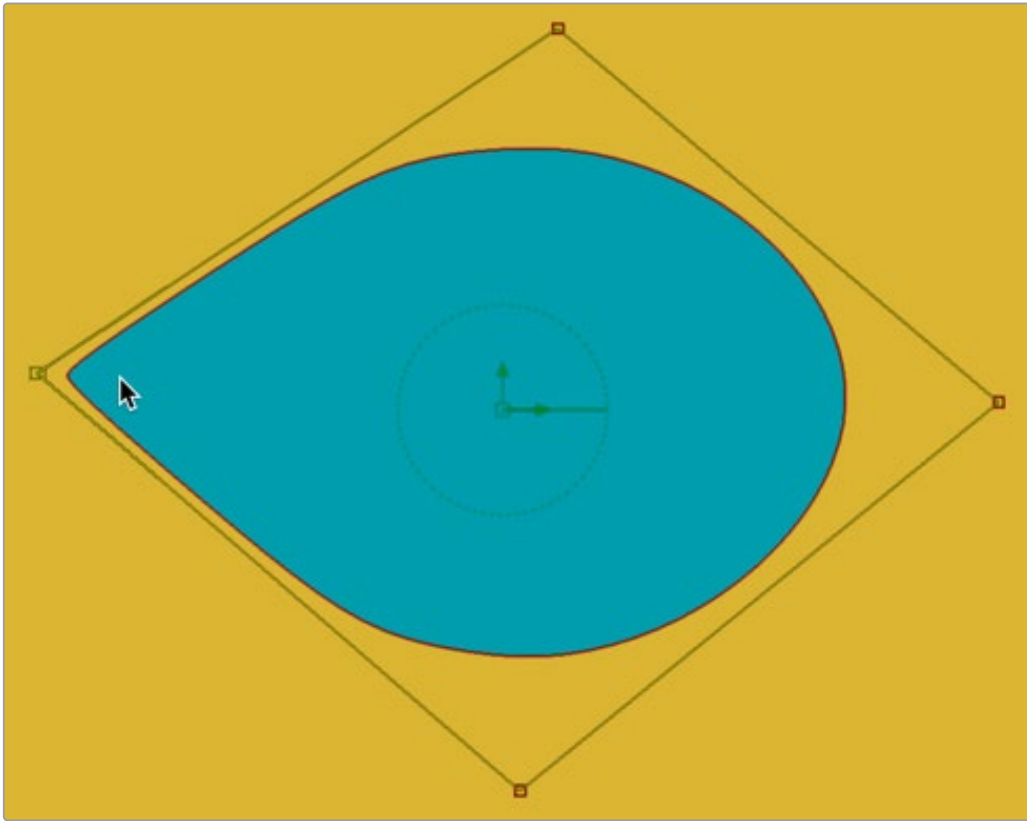
Adjusting Tension on a B-Spline

B-Spline Toolbar

When a B-Spline mask is selected in the Node Editor, a toolbar appears above the viewer with buttons for easy access to the modes. Position the pointer over any button in the toolbar to display a tooltip that describes that button's function.

B-Spline Mask Polygon toolbar

You can change the way the toolbar is displayed by right-clicking on the toolbar and selecting from the options displayed in the toolbar's contextual menu.

The functions of the buttons in this toolbar are explained in depth in the Polylines section.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are also duplicated in other mask nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Ellipse Mask [Elp]

The Ellipse node

Ellipse Mask Node Introduction

The Ellipse mask is most useful for masking round objects. It is a circle by default, but independent control is offered over the width, height, and angle, providing for a wide variety of ellipsoidal shapes.

Inputs

The Ellipse mask node includes a single effect mask input.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input combines the masks. How masks are combined is handled in the Paint mode menu in the Inspector.
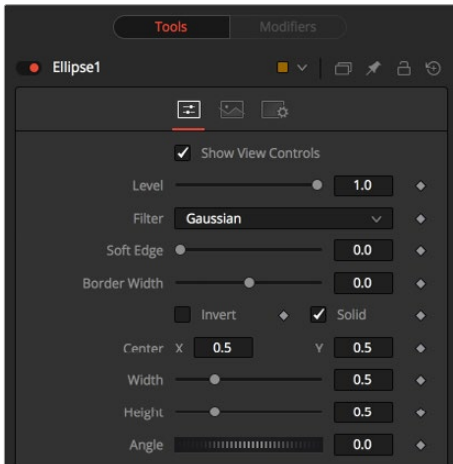
Basic Node Setup

The Ellipse mask node is useful for generating circular masks. Below, the Ellipse mask is used to generate a circular colored shape by cutting a circular shape from a background node.

Ellipse mask node creating a colored circle by masking a background node

Inspector

Ellipse Mask controls

Controls Tab

The Controls tab is used to refine how the ellipse appears after drawing it in the viewer.

Show View Controls

The Show View Controls checkbox is used to enable/disable the display of the mask's onscreen controls in the viewer. Onscreen controls, including center position, polylines, angles, and others, do not appear when this checkbox is disabled, even when the node is selected.

Level

The Level control sets the transparency level of the pixels in the mask channel. When the value is 1.0, the mask is completely opaque (unless it has a soft edge). Lower values cause the mask to be partially transparent. The result is identical to lowering the blend control of an effect.

> **NOTE:** Lowering the level of a mask lowers the values of all pixels covered by the mask in the mask channel. For example, if a Circle mask is placed over a Rectangle mask, lowering the level of the Circle mask lowers the values of all of the pixels in the mask channel, even though the Rectangle mask beneath it is still opaque.

Filter

This control selects the filtering algorithm used when applying Soft Edge to the mask.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.
— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.
— **Multi-box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."
— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Soft Edge

Use the Soft Edge slider to blur (feather) the mask, using the selected filter. Higher values cause the edge to fade off well beyond the boundaries of the mask. A value of 0.0 creates a crisp, well-defined edge.

Border Width

The Border Width control adjusts the thickness of the mask's edge. When the solid checkbox is toggled on, the border thickens or narrows the mask. When the mask is not solid, the mask shape draws as an outline, and the width uses the Border Width setting.

Paint Mode

Connecting a mask to the effect mask input displays the Paint mode menu. The Paint mode is used to determine how the incoming mask for the effect mask input and the mask created in the node are combined.

— **Merge:** Merge is the default for all masks. The new mask is merged with the input mask.

— **Add:** The mask's values add to the input mask's values.

— **Subtract:** In the intersecting areas, the new mask values subtract from the input mask's values.

— **Minimum:** Comparing the input mask's values and the new mask, this displays the lowest (minimum) value.

— **Maximum:** Comparing the input mask's values and the new mask, this displays the highest (maximum) value.

— **Average:** This calculates the average (half the sum) of the new mask and the input mask.

— **Multiply:** This multiplies the values of the input mask by the new mask's values.

— **Replace:** The new mask completely replaces the input mask wherever they intersect. Areas that are zero (completely black) in the new mask do not affect the input mask.

— **Invert:** Areas of the input mask that are covered by the new mask are inverted; white becomes black and vice versa. Gray areas in the new mask are partially inverted.

— **Copy:** This mode completely discards the input mask and uses the new mask for all values.

— **Ignore:** This mode completely discards the new mask and uses the input mask for all values.

Invert

Selecting this checkbox inverts the entire mask. Unlike the Invert Paint mode, the checkbox affects all pixels, regardless of whether the new mask covers them or not.

Solid

When the Solid checkbox is enabled, the mask is filled to be transparent (white) unless inverted. When disabled, the spline is drawn as just an outline whose thickness is determined by the Border Width slider.

Center X and Y

These controls adjust the position of the Ellipse mask.

Width

This control allows independent control of the ellipse mask's Width. In addition to the slider in the mask's controls, interactively drag the width (left or right edge) of the mask on the viewer using the pointer. Any changes will be reflected in this control.

Height

Height allows independent control of the ellipse mask's height. In addition to the slider in the mask's controls, interactively drag the height (top or bottom edge) of the mask on the view using the pointer. Any changes will be reflected in this control.

To change the mask's size without affecting the aspect ratio, drag the onscreen control between the edges (diagonal). This will modify both the width and height proportionately.

Angle

Change the rotational angle of the mask by moving the Angle control left or right. Values can be entered into the number fields provided. Alternately, use the onscreen controls by dragging the little circle at the end of the dashed angle line to interactively adjust the rotation of the ellipse.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are also duplicated in other mask nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Mask Paint [PnM]
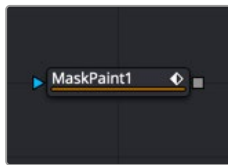
The Mask Paint node

Mask Paint Node Introduction

The Mask Paint node allows direct painting of mask images, using the pointer as if it was a paintbrush. In addition to regular paint strokes, it is possible to apply basic primitive shapes and polyline style strokes.

Each stroke can have a duration that lasts for the entire project, a single frame. or an arbitrary number of fields. The strokes can have independent durations in the Keyframes Editor for easy manipulation of time. Alternatively, Multistrokes is a faster but non-editable way for doing many mask clean up paint tasks.

Inputs

The Paint mask node includes a single effect mask input.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input combines the masks. How masks are combined is handled in the Paint mode menu in the Inspector.
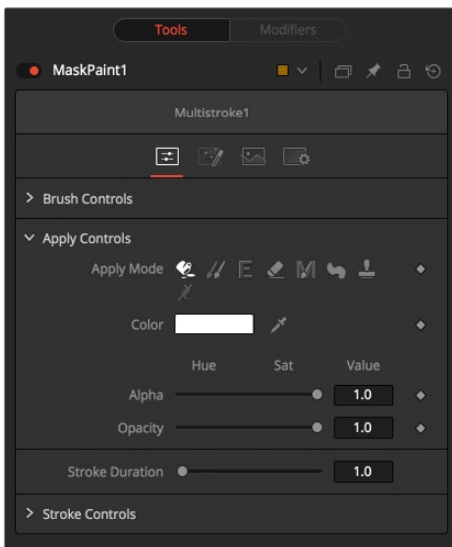
Basic Node Setup

The Mask Paint node is useful for painting masks using a more free hand, pressure sensitive style. In the node tree below, the Mask Paint node is used to patch up holes in a Bitmap mask.
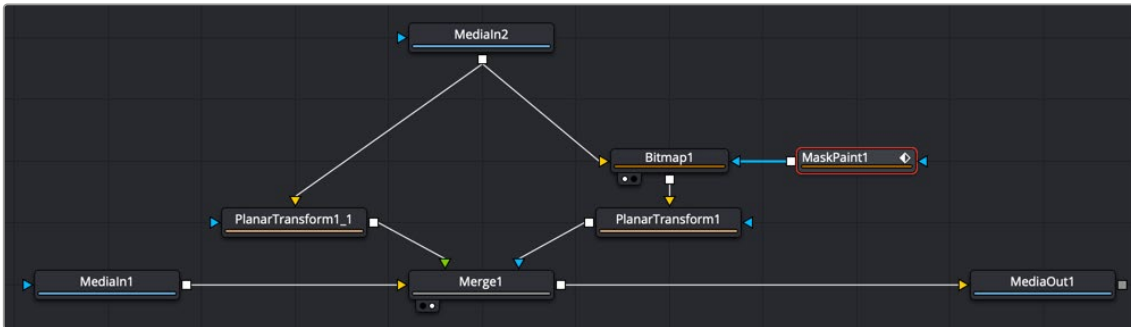
A Mask Paint node can be used to repair problematic areas of a matte.

Inspector

Mask Paint controls

As the Controls tab in the Mask Paint node is fundamentally identical to the Paint node, for more detail, *see Chapter 111, "Paint Node,"* in the DaVinci Resolve Reference Manual, or Chapter 49 in the Fusion Reference Manual. The only difference between the two nodes is that, as Mask Paint operates on single-channel mask images, there is no Channel Selector control, and all color controls have only a single Alpha value. The Mask tab, however, includes several parameters that are different from the Paint tool, so they are covered below.
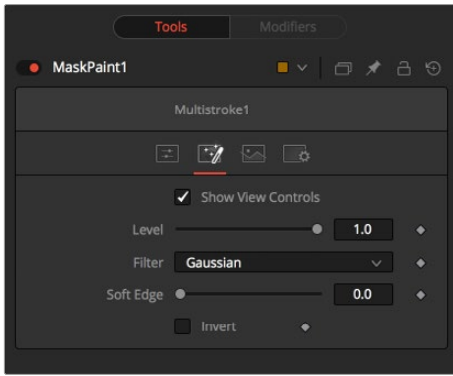
The Mask Paint tab

Mask Tab

The Mask tab is used to refine the basic mask parameters that do not fall into the category of "panting." These include how multiple masks are combined, overall softness control, and level control.

Show View Controls

The Show View Controls checkbox is used to enable/disable the display of the masks onscreen controls in the viewer. Onscreen controls including, center position, polylines, angles, and others, do not appear when this checkbox is disabled, even when the node is selected.

Level

The Level control sets the transparency level of the pixels in the mask channel. When the value is 1.0, the mask is completely opaque (unless it has a soft edge). Lower values cause the mask to be partially transparent. The result is identical to lowering the blend control of an effect.

> **NOTE:** Lowering the level of a mask lowers the values of all pixels covered by the mask in the mask channel. For example, if a Circle mask is placed over a Rectangle mask, lowering the level of the Circle mask lowers the values of all of the pixels in the mask channel, even though the Rectangle mask beneath it is still opaque.

Filter

This control selects the filtering algorithm used when applying Soft Edge to the mask.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.

— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.

— **Multi-box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."

— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Soft Edge

Use the Soft Edge slider to blur (feather) the mask, using the selected filter. Higher values cause the edge to fade off well beyond the boundaries of the mask. A value of 0.0 creates a crisp, well-defined edge.

Paint Mode

Connecting a mask to the effect mask input displays the Paint mode menu. The Paint mode is used to determine how the incoming mask for the effect mask input and the mask created in the node are combined.

— **Merge:** Merge is the default for all masks. The new mask is merged with the input mask.

— **Add:** The mask's values add to the input mask's values.

— **Subtract:** In the intersecting areas, the new mask values subtract from the input mask's values.

— **Minimum:** Comparing the input mask's values and the new mask, this displays the lowest (minimum) value.

— **Maximum:** Comparing the input mask's values and the new mask, this displays the highest (maximum) value.

— **Average:** This calculates the average (half the sum) of the new mask and the input mask.

— **Multiply:** This multiplies the values of the input mask by the new mask's values.

— **Replace:** The new mask completely replaces the input mask wherever they intersect. Areas that are zero (completely black) in the new mask do not affect the input mask.

— **Invert:** Areas of the input mask that are covered by the new mask are inverted; white becomes black and vice versa. Gray areas in the new mask are partially inverted.

— **Copy:** This mode completely discards the input mask and uses the new mask for all values.

— **Ignore:** This mode completely discards the new mask and uses the input mask for all values.

Invert

Selecting this checkbox inverts the entire mask. Unlike the Invert Paint mode, the checkbox affects all pixels, regardless of whether the new mask covers them or not.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are also duplicated in other mask nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Polygon Mask [Ply]
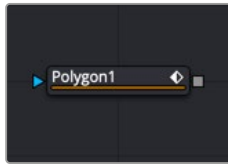
The Polygon node

Polygon Mask Node Introduction

The Polygon mask is most useful for masking objects that do not have a regular shape. When first added to a node, the Polygon mask consists of only Center and Angle controls, which are visible onscreen. Points are added to the polyline by clicking in the viewer. Each new point is connected to the last one created.

Like the B-Spline mask tool, the Polygon mask auto-animates. Adding this node to the Node Editor adds a keyframe to the current frame. Moving to a new frame and changing the shape creates a new keyframe and interpolate between the two defined shapes.

Inputs

The Polygon mask node includes a single effect mask input.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input combines the masks. How masks are combined is handled in the Paint mode menu in the Inspector.

Basic Node Setup

The Polygon node can be used to generate a detailed spline shape or combined with other masks for even more complex shapes. In the node tree below, the Polygon mask is used to generate detailed shape as a solid matte on the Delta Keyer.
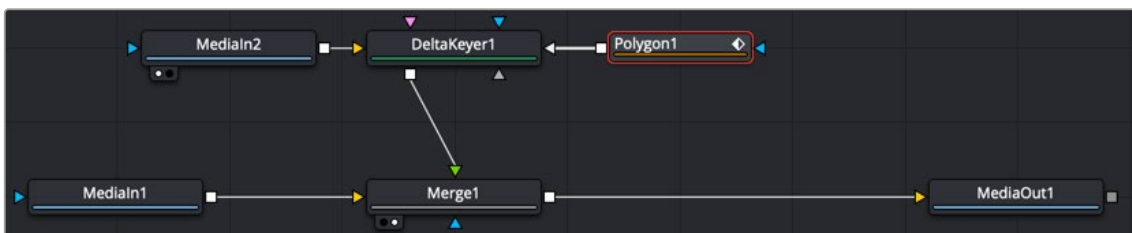
A Polygon node generates a detailed shape as a Solid matte.

Inspector

Polygon Mask controls

Controls Tab

The Controls tab is used to refine how the polyline appears after drawing it in the viewer.

Show View Controls

The Show View Controls checkbox is used to enable/disable the display of the mask's onscreen controls in the viewer. Onscreen controls, including center position, polylines, angles, and others, do not appear when this checkbox is disabled, even when the node is selected.

Level

The Level control sets the transparency level of the pixels in the mask channel. When the value is 1.0, the mask is completely opaque (unless it has a soft edge). Lower values cause the mask to be partially transparent. The result is identical to lowering the blend control of an effect.

> **NOTE:** Lowering the level of a mask lowers the values of all pixels covered by the mask in the mask channel. For example, if a Circle mask is placed over a Rectangle mask, lowering the level of the Circle mask lowers the values of all of the pixels in the mask channel, even though the Rectangle mask beneath it is still opaque.

Filter

This control selects the filtering algorithm used when applying Soft Edge to the mask.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.

— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.

— **Multi-box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."

— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Soft Edge

Use the Soft Edge slider to blur (feather) the mask, using the selected filter. Higher values cause the edge to fade off well beyond the boundaries of the mask. A value of 0.0 creates a crisp, well-defined edge.

Border Width

The Border Width control adjusts the thickness of the mask's edge. When the solid checkbox is toggled on, the border thickens or narrows the mask. When the mask is not solid, the mask shape draws as an outline, and the width uses the Border Width setting.

Paint Mode

Connecting a mask to the effect mask input displays the Paint mode menu. The Paint mode is used to determine how the incoming mask for the effect mask input and the mask created in the node are combined.

— **Merge:** Merge is the default for all masks. The new mask is merged with the input mask.

— **Add:** The mask's values add to the input mask's values.

— **Subtract:** In the intersecting areas, the new mask values subtract from the input mask's values.

— **Minimum:** Comparing the input mask's values and the new mask, this displays the lowest (minimum) value.

— **Maximum:** Comparing the input mask's values and the new mask, this displays the highest (maximum) value.

— **Average:** This calculates the average (half the sum) of the new mask and the input mask.

— **Multiply:** This multiplies the values of the input mask by the new mask's values.

— **Replace:** The new mask completely replaces the input mask wherever they intersect. Areas that are zero (completely black) in the new mask do not affect the input mask.

— **Invert:** Areas of the input mask that are covered by the new mask are inverted; white becomes black and vice versa. Gray areas in the new mask are partially inverted.

— **Copy:** This mode completely discards the input mask and uses the new mask for all values.

— **Ignore:** This mode completely discards the new mask and uses the input mask for all values.

Invert

Selecting this checkbox inverts the entire mask. Unlike the Invert Paint mode, the checkbox affects all pixels, regardless of whether the new mask covers them or not.

Solid

When the Solid checkbox is enabled, the mask is filled to be transparent (white) unless inverted. When disabled, the spline is drawn as just an outline whose thickness is determined by the Border Width slider.

Center X and Y

These controls adjust the position of the polygon spline mask.

Size

Use the Size control to adjust the scale of the polygon spline effect mask, without affecting the relative behavior of the points that compose the mask or setting a keyframe in the mask animation.

X, Y, and Z Rotation

Use these three controls to adjust the rotation angle of the mask along any axis.

Fill Method

The Fill Method menu offers two different techniques for dealing with overlapping regions of a polyline. If overlapping segments in a mask are causing undesirable holes to appear, try switching the setting of this control from Alternate to Non Zero Winding.

Right-Click Here for Shape Animation

By default, all polygon spline masks are animated when they are created. The initial keyframe is set to the current time, and any changes to the shape at different times create new keys.

Right-clicking on this label displays a contextual menu that offers options for removing or re-adding animation to the mask, or publishing and connecting the masks together.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are also duplicated in other mask nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Adding Points

Adding Points to a polygonal effect mask is relatively simple. Immediately after adding the node to the Node Editor, there are no points, but the tool will be in Click Append mode. Click once in the viewer wherever a point is required for the mask. Continue clicking to draw the shape of the mask. When the shape is complete, click on the initial point again to close the mask.

When the shape is closed, the mode of the polyline will change to Insert and Modify. This allows for the adjusting and adding of additional points to the mask by clicking on segments of the polyline. To lock down the mask's shape and prevent accidental changes, switch the Polyline mode to Done using the Polyline toolbar or contextual menu.

B-Spline Mask Polygon toolbar

When a Polygon (or B-Spline) mask is added to a node, a toolbar appears above the viewer, offering easy access to modes. Hold the pointer over any button in the toolbar to display a tooltip that describes that button's function.

— **Click:** Click is the default option when creating a polyline (or B-Spline) mask. It is a Bézier style drawing tool. Clicking sets a control point and appends the next control point when you click again in a different location.

— **Draw:** Draw is a freehand drawing tool. It creates a mask similar to drawing with a pencil on paper. You can create a new mask using the Draw tool, or you can extend an existing open spline by clicking the Draw tool and starting to draw from the last control point.

— **Insert:** Insert adds a new control point along the spline.

— **Modify:** Modify allows you to safely move or smooth any exiting point along a spline without worrying about adding new points accidentally.

— **Done:** Prevents any point along the spline from being moved or modified. Also, new points cannot be added. You can, however, move and rotate the entire spline.

— **Closed:** Closes an open spline.

— **Smooth:** Changes the selected control point from a linear to a smooth curve.

— **Linear:** Changes the selected control point from a smooth curve to linear.

— **Select All:** Selects all the control points on the spline.

— **Keys:** Shows or hides the control points along the spline.

— **Handles:** Shows or hides the Bézier handles along the polyline.

— **Shape:** Places a reshape rectangle around the selected spline shape. Using the reshape rectangle, you can deform groups of control points or entire shapes much easier than modifying each point.

— **Delete:** Deletes the selected control point(s).

— **Reduce:** Opens a Freehand precision window that can be used to reduce the number of controls points on a spline. This can make the paint stroke easier to modify, especially if it has been created using the Draw tool.

— **Publish menu:** You can use the publish menu to select between publishing the control points or the path. Publishing is a form of parameter linking, it makes the selected item available for use by other controls. It also allows you to attach a control point to a tracker.

— **Follow Points:** Allows a selected point to follow the path of a published point. The point follows the published point using an offset position.

— **Double Poly:** Allows softening part of the spline curve while keeping other portions of the curve sharp. The double polyline is composed of two shapes, an inner and outer shape. The inner shape is the original shape from the single polyline, whereas the outer shape is used to determine the spread of the softness. The further the outer shape gets from the inner shape, the softer that segment of the shape becomes. Both polylines start with exactly the same shape as the original single polyline, keeping the mask sharp to start. Any animation already applied to the shape remains. To select the outer shape, press the Tab key to cycle between the onscreen controls until the dashed outline is visible, or you can select the outer polyline using the contextual menu's Controls > Outer Polygon menu.

— **Multiframe:** Multiframe is a method of adjusting control points across multiple keyframes. The default setting of none only adjusts the control point of a spline on the current keyframe. Setting the menu to All adjusts the controls point for all keyframes. Prev settings adjust the current and previous keyframe while Next adjusts the current and next keyframe.

— **Onion Skinning:** Enabling onion skinning displays a mix in the viewer of the spline animation. It is useful when aligning spline animation and motion. Selecting Onion Skin Settings from the drop down menu allows you to set the number of overlapping frames.

— **Roto Assist:** Enable the Roto Assist button when you begin painting with the Polyline Stroke tool. The polyline points snap to the closest edge as you click to add points to the shape. A cyan outline indicates the points that have snapped to an edge. There are three main Roto Assist options selectable through the drop down menu:

  — **Multiple Points:** When enabled, a single click on a high contrast edge adds multiple points to define the entire edge, instead of having to add each point individually. This is a one time only click. The second click reverts to single point edge detection.

  — **Distance 8:** Opens a dialog where you can set the pixel range within which searching for an edge takes place.

  — **Reset:** Used for resetting the snap attribute of all snapped points. After resetting, the points become unavailable for tracking.

Change the way the toolbar is displayed by right-clicking on the toolbar and selecting from the options displayed in the toolbar's contextual menu. The functions of the buttons in this toolbar are explained in depth in the Polylines chapter.

Ranges Mask [Rng]
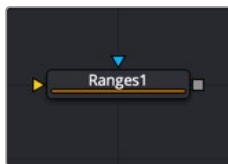
The Ranges node

Ranges Mask Node Introduction

Similar to Bitmap mask, the Ranges mask allows images from the node tree to act as masks for nodes and effects. Instead of creating a simple luminance-based mask from a given channel, Ranges allows spline-based selection of low, mid and high ranges, akin to Color Corrector.
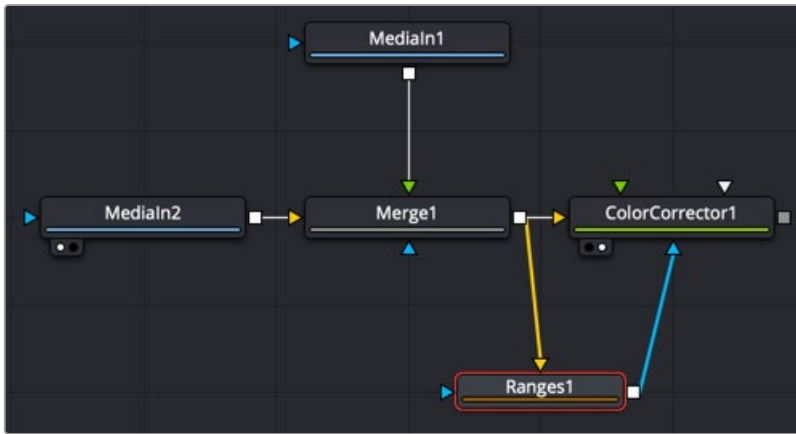
Inputs

The Ranges mask node includes two inputs in the Node Editor.

— **Input:** The orange input accepts a 2D image from which the mask will be created.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input combines the masks. How masks are combined is handled in the Paint mode menu in the Inspector.

Basic Node Setup

The Ranges node is not required for connecting an image into the effect mask input, but like the Bitmap node, it does provide options that are otherwise unavailable. It allows for selecting channels other than RGBA for the mask, as well as softness and clipping. In the node tree below, the Ranges node takes the composite out of the merge, creating a mask for the color correction.

A Ranges node selects a specific range in the image to create a mask.

Inspector
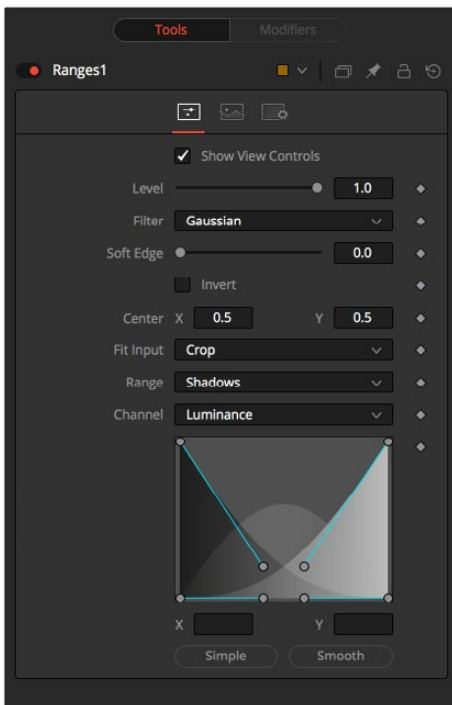
Rangers Mask controls

Controls Tab

The Controls tab is used to refine how the image connected to the orange input converts into the ranges mask.

Show View Controls

The Show View Controls checkbox is used to enable/disable the display of the mask's onscreen controls in the viewer. Onscreen controls, including center position, polylines, angles, and others, do not appear when this checkbox is disabled, even when the node is selected.

Level

The Level control sets the transparency level of the pixels in the mask channel. When the value is 1.0, the mask is completely opaque (unless it has a soft edge). Lower values cause the mask to be partially transparent. The result is identical to lowering the blend control of an effect.

> **NOTE:** Lowering the level of a mask lowers the values of all pixels covered by the mask in the mask channel. For example, if a Circle mask is placed over a Rectangle mask, lowering the level of the Circle mask lowers the values of all of the pixels in the mask channel, even though the Rectangle mask beneath it is still opaque.

Filter

This control selects the filtering algorithm used when applying Soft Edge to the mask.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.

— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.

— **Multi-box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."

— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Soft Edge

Use the Soft Edge slider to blur (feather) the mask, using the selected filter. Higher values cause the edge to fade off well beyond the boundaries of the mask. A value of 0.0 creates a crisp, well-defined edge.

Paint Mode

Connecting a mask to the effect mask input displays the Paint mode menu. The Paint mode is used to determine how the incoming mask for the effect mask input and the mask created in the node are combined.

— **Merge:** Merge is the default for all masks. The new mask is merged with the input mask.

— **Add:** The mask's values add to the input mask's values.

— **Subtract:** In the intersecting areas, the new mask values subtract from the input mask's values.

— **Minimum:** Comparing the input mask's values and the new mask, this displays the lowest (minimum) value.

— **Maximum:** Comparing the input mask's values and the new mask, this displays the highest (maximum) value.

— **Average:** This calculates the average (half the sum) of the new mask and the input mask.

— **Multiply:** This multiplies the values of the input mask by the new mask's values.

— **Replace:** The new mask completely replaces the input mask wherever they intersect. Areas that are zero (completely black) in the new mask do not affect the input mask.

— **Invert:** Areas of the input mask that are covered by the new mask are inverted; white becomes black and vice versa. Gray areas in the new mask are partially inverted.

— **Copy:** This mode completely discards the input mask and uses the new mask for all values.

— **Ignore:** This mode completely discards the new mask and uses the input mask for all values.

Invert

Selecting this checkbox inverts the entire mask. Unlike the Invert Paint mode, the checkbox affects all pixels, regardless of whether the new mask covers them or not.

Center X and Y

These controls adjust the position of the ranges mask.

Fit Input

This menu is used to select how the image source is treated if it does not fit the dimensions of the generated mask.

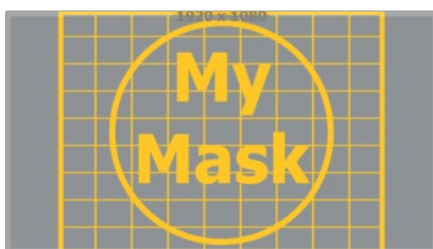For example, below, a 720 x 576 image source (yellow) is used to generate a 1920 x 1080 mask (gray).

— **Crop:** If the image source is smaller than the generated mask, it is placed according to the X/Y controls, masking off only a portion of the mask. If the image source is larger than the generated mask it is placed according to the X/Y controls and cropped off at the borders of the mask.

— **Stretch:** The image source is stretched in X and Y to accommodate the full dimensions of the generated mask. This might lead to visible distortions of the image source.

— **Inside:** The image source is scaled uniformly until one of its dimensions (X or Y) fits the inside dimensions of the mask. Depending on the relative dimensions of the image source and mask background, either the image source's width or height may be cropped to fit the respective dimension of the mask.

— **Width:** The image source is scaled uniformly until its width (X) fits the width of the mask. Depending on the relative dimensions of the image source and mask, the image source's Y dimension might not fit the mask's Y dimension, resulting in either cropping of the image source in Y or the image source not covering the mask's height entirely.

— **Height:** The image source is scaled uniformly until its height (Y) fits the height of the mask. Depending on the relative dimensions of the image source and mask, the image source's X dimension might not fit the mask's X dimension, resulting in either cropping of the image source in X or the image source not covering the mask's width entirely.

— **Outside:** The image source is scaled uniformly until one of its dimensions (X or Y) fits the outside dimensions of the mask. Depending on the relative dimensions of the image source and mask, either the image source's width or height may be cropped or not fit the respective dimension of the mask.

Channel

The Channel menu determines the Channel of the input image used to create the mask. Choices include the red, green, blue, and alpha channels; the hue, luminance, or saturation values; or the auxiliary coverage channel of the input image (if one is provided).

Shadows/Midtones/Highlights

These buttons are used to select which range is output by the node as a mask. White pixels represent pixels that are considered to be part of the range, and black pixels are not included in the range. For example, choosing Shadows would show pixels considered to be shadows as white, and pixels that are not shadows as black. Mid gray pixels are only partly in the range and do not receive the full effect of any color adjustments to that range.

Channel

The Channel selection buttons shown in this tab can be used to extract a mask from the range of a specific color channel. By default, Fusion uses the luminance channel when the color ranges are examined.

Mini Spline Editor

The extent of the ranges is selected by manipulating the spline handles. There are four spline points, each with one Bézier handle. The two handles at the top represent the start of the shadow and highlight ranges; the two handles at the bottom represent the end of the range. The Bézier handles are used to control the falloff.

The midtones range has no specific control, since its range is understood to be the space between the shadow and highlight ranges. In other words, after low and high masks have been applied, midtones are everything else.

The X and Y text controls below the Mini Spline Editor can be used to enter precise positions for the selected Bézier point or handle.

Presets

This sets the splines to two commonly-used configurations. The Simple button gives a straightforward linear-weighted selection, while the Smooth button uses a more natural falloff.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are also duplicated in other mask nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Rectangle Mask [Rec]
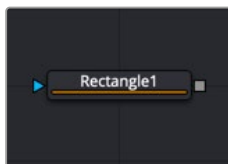
The Rectangle node

Rectangle Mask Node Introduction

The Rectangle mask creates simple square or rectangular masks. By default, it creates a rectangle in the same aspect ratio as the comp, but independent control is offered over the width, height, and angle, providing for a wide variety of rectangular shapes.

Inputs

The Rectangle mask node includes a single effect mask input.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input combines the masks. How masks are combined is handled in the Paint mode menu in the Inspector.
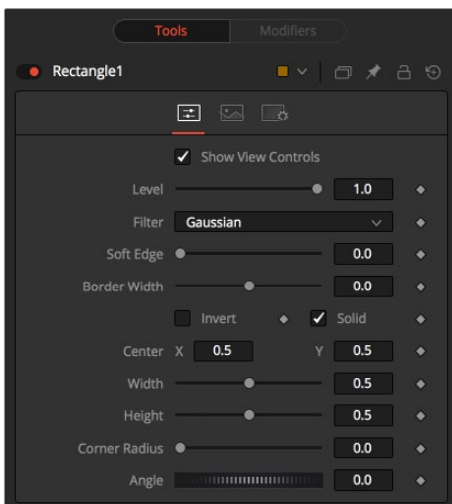
Basic Node Setup

The Rectangle mask node is useful for generating rectangular shapes. Below, the Rectangle mask is used to generate a colored square by cutting a square shape from a background node.
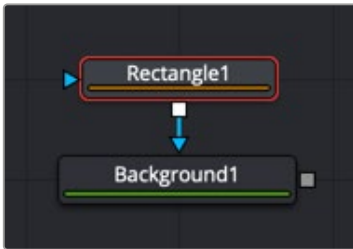
A Rectangle mask node creating a colored square by masking a Background node

Inspector

Rectangle Mask controls

Controls Tab

The Controls tab is used to refine how the rectangle appears after drawing it in the viewer.

Show View Controls

The Show View Controls checkbox is used to enable/disable the display of the mask's onscreen controls in the viewer. Onscreen controls, including center position, polylines, angles, and others, do not appear when this checkbox is disabled, even when the node is selected.

Level

The Level control sets the transparency level of the pixels in the mask channel. When the value is 1.0, the mask is completely opaque (unless it has a soft edge). Lower values cause the mask to be partially transparent. The result is identical to lowering the Blend control of an effect.

> **NOTE:** Lowering the level of a mask lowers the values of all pixels covered by the mask in the mask channel. For example, if a Circle mask is placed over a Rectangle mask, lowering the level of the Circle mask lowers the values of all the pixels in the mask channel, even though the Rectangle mask beneath it is still opaque.

Filter

This control selects the filtering algorithm used when applying Soft Edge to the mask.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.

— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.

— **Multi-box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."

— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Soft Edge

Use the Soft Edge slider to blur (feather) the mask, using the selected filter. Higher values cause the edge to fade off well beyond the boundaries of the mask. A value of 0.0 creates a crisp, well-defined edge.

Border Width

The Border Width control adjusts the thickness of the mask's edge. When the solid checkbox is toggled on, the border thickens or narrows the mask. When the mask is not solid, the mask shape draws as an outline, and the width uses the Border Width setting.

Paint Mode

Connecting a mask to the effect mask input displays the Paint mode menu. The Paint mode is used to determine how the incoming mask for the effect mask input and the mask created in the node are combined.

— **Merge:** Merge is the default for all masks. The new mask is merged with the input mask.

— **Add:** The mask's values add to the input mask's values.

— **Subtract:** In the intersecting areas, the new mask values subtract from the input mask's values.

— **Minimum:** Comparing the input mask's values and the new mask, this displays the lowest (minimum) value.

— **Maximum:** Comparing the input mask's values and the new mask, this displays the highest (maximum) value.

— **Average:** This calculates the average (half the sum) of the new mask and the input mask.

— **Multiply:** This multiplies the values of the input mask by the new mask's values.

— **Replace:** The new mask completely replaces the input mask wherever they intersect. Areas that are zero (completely black) in the new mask do not affect the input mask.

— **Invert:** Areas of the input mask that are covered by the new mask are inverted: white becomes black and vice versa. Gray areas in the new mask are partially inverted.

— **Copy:** This mode completely discards the input mask and uses the new mask for all values.

— **Ignore:** This mode completely discards the new mask and uses the input mask for all values.

Invert

Selecting this checkbox inverts the entire mask. Unlike the Invert Paint mode, this checkbox affects all pixels, regardless of whether the new mask covers them.

Solid

When the Solid checkbox is enabled, the mask is filled to be transparent (white) unless inverted. When disabled, the spline is drawn as just an outline whose thickness is determined by the Border Width slider.

Center X and Y

These controls adjust the position of the Rectangle mask.

Width and Height

Use these controls to change the X or Y scale of the rectangular effect mask independently of each other. Alternatively, drag the edges of the rectangle in the viewer to interactively adjust its size.

Corner Radius

Corner Radius allows the corners of the Rectangle mask to be rounded. A value of 0.0 is not rounding at all, which means that the rectangle has sharp corners. A value of 1.0 applies the maximum amount of rounding to the corners.

Angle

Change the rotation angle of an effect mask by moving the Angle control left or right. Values can be entered in the provided input boxes. Alternatively, use the onscreen controls by dragging the little circle at the end of the dashed angle line to interactively adjust the rotation of the ellipse.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are also duplicated in other mask nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Triangle Mask [Tri]

The Triangle node

Triangle Mask Node Introduction

The Triangle mask creates simple triangular masks. It is unique in that it has no Center, Size, or Angle control. Unlike most other types of masks, all three points of the triangle can attach to a tracker or motion path.

Inputs

The Triangle mask node includes a single effect mask input.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input combines the masks. How masks are combined is handled in the Paint mode menu in the Inspector.
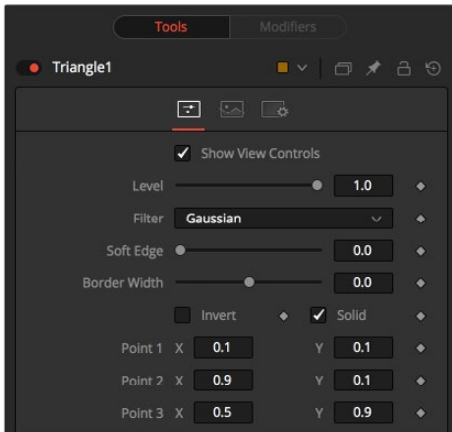
Basic Node Setup

The Triangle mask node is useful for generating triangular shapes. Below, the Triangle mask is used to generate a colored triangle by cutting a triangular shape from a background node.
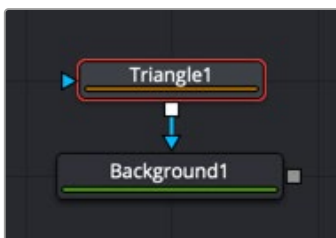
A Triangle mask node creating a colored Triangle shape by masking a Background node

Inspector

Triangle Mask controls

Controls Tab

The Controls tab is used to refine how the triangle appears after drawing it in the viewer.

Show View Controls

The Show View Controls checkbox is used to enable/disable the display of the mask's onscreen controls in the viewer. Onscreen controls, including center position, polylines, angles, and others, do not appear when this checkbox is disabled, even when the node is selected.

Level

The Level control sets the transparency level of the pixels in the mask channel. When the value is 1.0, the mask is completely opaque (unless it has a soft edge). Lower values cause the mask to be partially transparent. The result is identical to lowering the Blend control of an effect.

> **NOTE:** Lowering the level of a mask lowers the values of all pixels covered by the mask in the mask channel. For example, if a Circle mask is placed over a Rectangle mask, lowering the level of the Circle mask lowers the values of all the pixels in the mask channel, even though the Rectangle mask beneath it is still opaque.

Filter

This control selects the filtering algorithm used when applying Soft Edge to the mask.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.
— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.
— **Multi-box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."
— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Soft Edge

Use the Soft Edge slider to blur (feather) the mask, using the selected filter. Higher values cause the edge to fade off well beyond the boundaries of the mask. A value of 0.0 creates a crisp, well-defined edge.

Border Width

The Border Width control adjusts the thickness of the mask's edge. When the solid checkbox is toggled on, the border thickens or narrows the mask. When the mask is not solid, the mask shape draws as an outline, and the width uses the Border Width setting.

Paint Mode

Connecting a mask to the effect mask input displays the Paint mode menu. The Paint mode is used to determine how the incoming mask for the effect mask input and the mask created in the node are combined.

— **Merge:** Merge is the default for all masks. The new mask is merged with the input mask.

— **Add:** The mask's values add to the input mask's values.

— **Subtract:** In the intersecting areas, the new mask values subtract from the input mask's values.

— **Minimum:** Comparing the input mask's values and the new mask, this displays the lowest (minimum) value.

— **Maximum:** Comparing the input mask's values and the new mask, this displays the highest (maximum) value.

— **Average:** This calculates the average (half the sum) of the new mask and the input mask.

— **Multiply:** This multiplies the values of the input mask by the new mask's values.

— **Replace:** The new mask completely replaces the input mask wherever they intersect. Areas that are zero (completely black) in the new mask do not affect the input mask.

— **Invert:** Areas of the input mask that are covered by the new mask are inverted: white becomes black and vice versa. Gray areas in the new mask are partially inverted.

— **Copy:** This mode completely discards the input mask and uses the new mask for all values.

— **Ignore:** This mode completely discards the new mask and uses the input mask for all values.

Invert

Selecting this checkbox inverts the entire mask. Unlike the Invert Paint mode, this checkbox affects all pixels, regardless of whether the new mask covers them.

Solid

When the Solid checkbox is enabled, the mask is filled to be transparent (white) unless inverted. When disabled, the spline is drawn as just an outline whose thickness is determined by the Border Width slider.

Point 1, Point 2, Point 3

These controls show the position coordinates of the three corners of the triangle. Each point can be published, connected to other controls, animated with a path, or attached to trackers. These tasks are performed by right-clicking the Position control or directly on the point in the viewer.

Common Controls

Image and Settings Tabs

The Image and Settings tabs in the Inspector are also duplicated in other mask nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Wand Mask [Wnd]
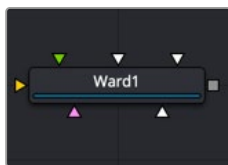
The Wand node

Wand Mask Node Introduction

The Wand mask masks an image based on a wand-style selection, similar to the Magic Wand tool found in Adobe Photoshop. As with a Bitmap mask, any image in the composition can be the source of the mask. Generally, the default is most useful, where the source image is the input of the node to which the mask is applied.

When adding a Wand mask to a node, a crosshair appears in the viewers. This crosshair should be positioned in the image to select the color used to create the Wand mask. The mask is created by examining the pixel color beneath the selection point and adding that color to the mask. The mask then expands to examine the pixels surrounding the selection point. Surrounding pixels are added to the mask if they are the same color. The mask stops expanding when no connecting pixels fall within the color range of the mask.

Inputs

The Wand mask node includes two inputs in the Node Editor.

— **Input:** The orange input accepts a 2D image from which the mask is created.
— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input combines the masks. How masks are combined is handled in the Paint mode menu in the Inspector.

Basic Node Setup

The Wand mask node is not required for connecting an image into the effect mask input, but like the Bitmap node, it does provide options that are otherwise unavailable. It allows for selecting channels other than RGBA for the mask, as well as softness and clipping. In the node tree below, the Wand node takes the composite out of the merge, creating a mask for the color correction.

A Wand node selects a specific area in the image to create a mask.

Inspector
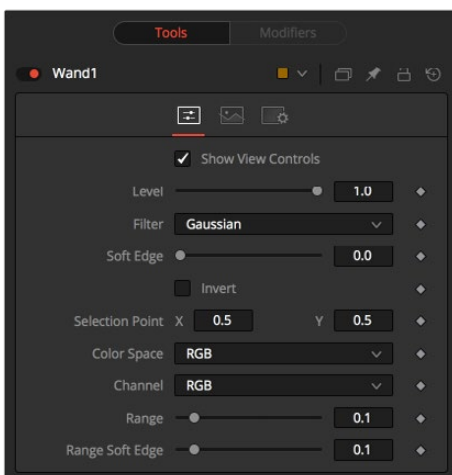
Wand Mask controls

Controls Tab

The Controls tab is used to refine how the mask appears after the Wand makes a selection in the viewer.

Show View Controls

The Show View Controls checkbox is used to enable/disable the display of the mask's onscreen controls in the viewer. Onscreen controls, including center position, polylines, angles, and others, do not appear when this checkbox is disabled, even when the node is selected.

Level

The Level control sets the transparency level of the pixels in the mask channel. When the value is 1.0, the mask is completely opaque (unless it has a soft edge). Lower values cause the mask to be partially transparent. The result is identical to lowering the Blend control of an effect.

> **NOTE:** Lowering the level of a mask lowers the values of all pixels covered by the mask in the mask channel. For example, if a Circle mask is placed over a Rectangle mask, lowering the level of the Circle mask lowers the values of all the pixels in the mask channel, even though the Rectangle mask beneath it is still opaque.

Filter

This control selects the filtering algorithm used when applying Soft Edge to the mask.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.

— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.

— **Multi-box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."

— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Soft Edge

Use the Soft Edge slider to blur (feather) the mask, using the selected filter. Higher values cause the edge to fade off well beyond the boundaries of the mask. A value of 0.0 creates a crisp, well-defined edge.

Paint Mode

Connecting a mask to the effect mask input displays the Paint mode menu. The Paint mode is used to determine how the incoming mask for the effect mask input and the mask created in the node are combined.

— **Merge:** Merge is the default for all masks. The new mask is merged with the input mask.

— **Add:** The mask's values add to the input mask's values.

— **Subtract:** In the intersecting areas, the new mask values subtract from the input mask's values.

— **Minimum:** Comparing the input mask's values and the new mask, this displays the lowest (minimum) value.

— **Maximum:** Comparing the input mask's values and the new mask, this displays the highest (maximum) value.

— **Average:** This calculates the average (half the sum) of the new mask and the input mask.

— **Multiply:** This multiplies the values of the input mask by the new mask's values.

— **Replace:** The new mask completely replaces the input mask wherever they intersect. Areas that are zero (completely black) in the new mask do not affect the input mask.

— **Invert:** Areas of the input mask that are covered by the new mask are inverted; white becomes black and vice versa. Gray areas in the new mask are partially inverted.

— **Copy:** This mode completely discards the input mask and uses the new mask for all values.

— **Ignore:** This mode completely discards the new mask and uses the input mask for all values.

Invert

Selecting this checkbox inverts the entire mask. Unlike the Invert Paint mode, this checkbox affects all pixels, regardless of whether the new mask covers them.

Selection Point

The Selection Point is a pair of X and Y coordinates that determines where in the source image the Wand mask derives its initial color sample. This control is also seen as a crosshair in the viewers. The selection point can be positioned manually, connected to a tracker, path, or other expressions.

Color Space

The Color Space button group determines the color space used when selecting the source color for the mask. The Wand mask can operate in RGB, YUV, HLS, or LAB color spaces.

Channel

The Channel button group is used to select whether the color that is masked comes from all three color channels of the image, the alpha channel, or an individual channel only.

The exact labels of the buttons depend on the color space selected for the Wand mask operation. If the color space is RGB, the options are R, G, or B. If YUV is the color space, the options are Y, U, or V.

Range

The Range slider controls the range of colors around the source color that are included in the mask. If the value is left at 0.0, only pixels of the same color as the source are considered part of the mask. The higher the value, the more that similar colors in the source are considered to be wholly part of the mask.

Range Soft Edge

The Range Soft Edge determines the falloff range of the colors selected. Any pixel within the range defined above are treated as 100% within the mask. If the soft range is set to 0.0, no other pixels are considered for the mask. Increasing the soft range increases the number of colors close to, but not quite within, the range included in the mask. These pixels are semitransparent in the mask

The Common Controls

Nodes that create masks share several identical controls in the Inspector. This section describes controls that are common among mask nodes.

Inspector
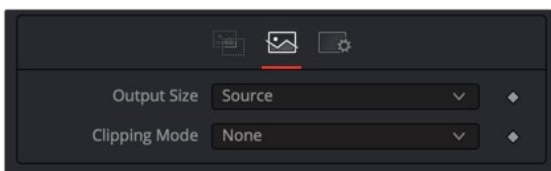
Mask node Image tab

Image Tab

The controls in this tab set the resolution and clipping method used by the generated mask.

Output Size

The Output size menu sets the resolution of the mask node's output. The three options include the default resolution of the comp, the source input's resolution on nodes that have an input, or a custom resolution.

Custom

When selecting Custom from the Output Size menu, the width, height, and pixel aspect of the mask created are locked to values defined in the composition's Frame Format preferences. If the Frame Format preferences change, the resolution of the mask produced is changed to match. Disabling this option can be useful for building a composition at a different resolution than the eventual target resolution for the final render.
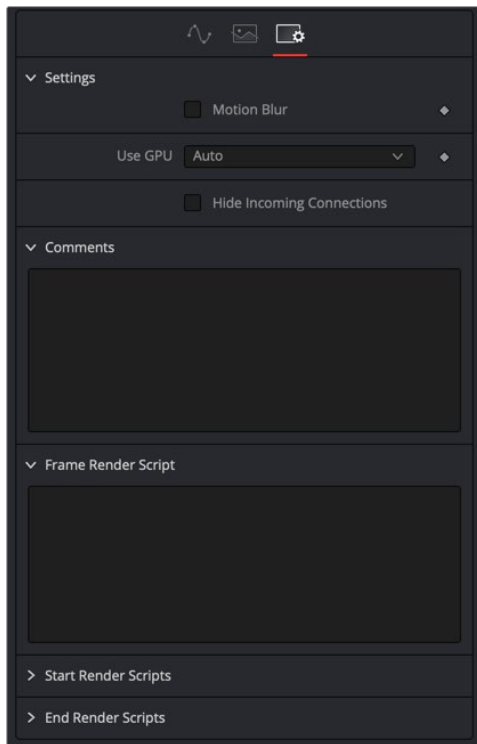
— **Width and Height:** This pair of controls is used to set the Width and Height dimensions of the mask to be created.

— **Pixel Aspect:** This control is used to specify the Pixel Aspect ratio of the created mask. An aspect ratio of 1:1 would generate a square pixel with the same dimensions on either side (like a computer monitor), and an aspect of 0.91 would create a slightly rectangular pixel (like an NTSC monitor).

— **Depth:** The Depth drop-down menu is used to set the pixel color depth of the image created by the mask. 32-bit pixels require four times the memory of 8-bit pixels but have far greater accuracy. Float pixels allow high dynamic range values outside the normal 0..1 range, for representing colors that are brighter than white or darker than black.

> **NOTE:** Right-click on the Width, Height, or Pixel Aspect controls to display a menu listing the file formats defined in the preferences Frame Format tab. Selecting any of the listed options sets the width, height, and pixel aspect to the values for that format.

Clipping Mode

This option determines how the domain of definition rendering handles edges. The Clipping mode is most important when blur or softness is applied, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **None:** Setting this option to None does not perform any source image clipping. Any data required to process the node's effect that would usually be outside the upstream DoD is treated as black/transparent.

Common Mask Settings controls

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Mask category. The Settings controls are even found on third-party plugin tools. The controls are consistent and work the same way for each tool, although some tools do include one or two individual options, which are also covered here.

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 causes Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one full frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows for the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off hardware-accelerated rendering using the graphics card in your computer. Enabled uses the hardware, and Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Matte Nodes

This chapter details the Matte nodes available in Fusion.

> The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.
>
> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Alpha Divide [ADv]

The Alpha Divide node

Alpha Divide Node Introduction

As the name gives away, the Alpha Divide's sole purpose is to divide an incoming image's color channels by its alpha channel. When you color correct an image that contains a premultiplied alpha channel, first apply an Alpha Divide node before any color correction node to create a non-premultiplied image. Then you can perform the color correction. After the color correction, add an Alpha Multiply node to return the image to its premultiplied state.

Inputs

The Alpha Divide node includes two inputs in the Node Editor.

— **Input:** The orange input accepts a 2D image with a premultiplied Alpha.
— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input limits the pixels where the Alpha divide occurs. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Alpha Divide node is placed before any color correction is done to an image with a premultiplied Alpha channel. Here the MediaIn node is assumed to have a premultiplied Alpha channel. The Alpha Divide node is inserted, and then color correction nodes operate on the "straight" Alpha. An Alpha Multiply node is placed at the end of the chain to premultiply the Alpha channel again. If only a single color correction node is used, then the Pre-Divide/Post-Multiply checkbox on the Options tab can be used in place of the Alpha Divide/Alpha Multiple nodes.
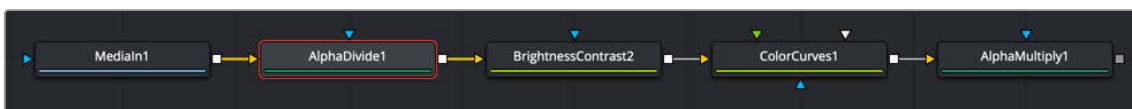
An Alpha Divide node is inserted before color correcting an image with premultiplied alpha.

Inspector

This node has no controls.

Alpha Multiply [AMl]

The Alpha Multiply node

Alpha Multiply Node Introduction

As the name gives away, the Alpha Multiply's sole purpose is to multiply an image's color channels by its alpha channel. When you color correct an image that contains a premultiplied alpha channel, first apply an Alpha Divide node before any color correction node to create a non-premultiplied image. Then you can perform the color correction. After the color correction, add an Alpha Multiply node to return the image to its premultiplied state.

Inputs

The Alpha Multiply node includes two inputs in the Node Editor.

— **Input:** The orange input accepts a 2D image with a "straight" or non-premultiplied alpha.
— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input limits the pixels where the Alpha multiply occurs. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Alpha Multiply node is placed after color correction is done to an image with a premultiplied Alpha channel. Here the MediaIn node is assumed to have a premultiplied Alpha channel. The Alpha Divide node is inserted, and then color correction nodes operate on the "straight" Alpha. An Alpha Multiply node is placed at the end of the chain to premultiply the Alpha channel again. If only a single color correction node is used, then the Pre-Divide/Post-Multiply checkbox on the Options tab can be used in place of the Alpha Divide/Alpha Multiple nodes.
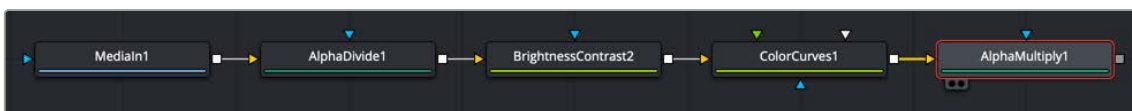
An Alpha Multiply node is inserted after color correcting an image with premultiplied alpha.

Inspector

This node has no controls.

Chroma Keyer [CKy]

The Chroma Keyer node

Chroma Keyer Node Introduction

The Chroma Keyer node creates an alpha channel (matte) for an image by removing selected colors from the scene. Unlike the Delta Keyer or Primatte, which use specific optimizations for keying from blue and green colors, the Chroma Keyer works equally well with any color.

> **NOTE:** When working with blue- or green-screen shots, it is best to use the Delta Keyer or Primatte node, rather than the more general purpose Chroma Keyer node.

Inputs

The Chroma Keyer node includes four inputs in the Node Editor.

— **Input:** The orange input accepts a 2D image that contains the color you want to be keyed for transparency.

— **Garbage Matte:** The gray garbage matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be made transparent. The garbage matte is applied directly to the alpha channel of the image.

— **Solid Matte:** The white solid matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be fully opaque.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input limits the pixels where the alpha multiply occurs. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Chroma Keyer node takes in a 2D image like the MediaIn node below and removes a color that you identify by dragging over it in the viewer. The result is that the selected color is replaced with transparency, allowing you to composite the image as the foreground in a Merge node.
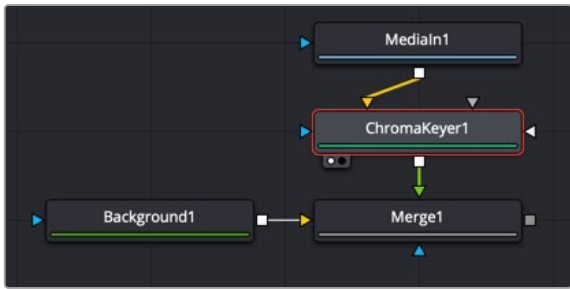
A Chomra Keyer node creating transparency on the MediaIn node.

Inspector
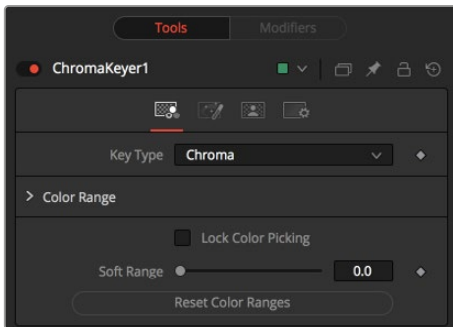
The Chroma Key tab

Chroma Key Tab

The Chroma Key tab is used to make the initial selection of color for keying.

Key Type

The Key Type menu determines the selection method used for the matte creation.

— **Chroma:** The Chroma method creates a matte based on the RGB values of the selected color range.

— **Color:** The Color method creates a matte based on the hue of the selected color range.

Color Range

Colors are made transparent by selecting the Chroma Keyer node in the node tree, and then dragging a selection around the colors in the viewer. The range controls update automatically to represent the current color selection. You can tweak the range sliders slightly, although most often selecting colors in the displays is all that is required.

Lock Color Picking

When this checkbox is activated, selecting colors from the viewer is disabled to prevent accidental addition to the range. It is a good idea to activate this checkbox once you make the color selection for the matte. All other controls in the node remain editable.

Soft Range

This control softens the selected color range, adding additional colors into the matte.

Reset Color Ranges

Clicking this button resets the Chroma Keyer's range controls, discarding all color selections. All other sliders and controls maintain their values.
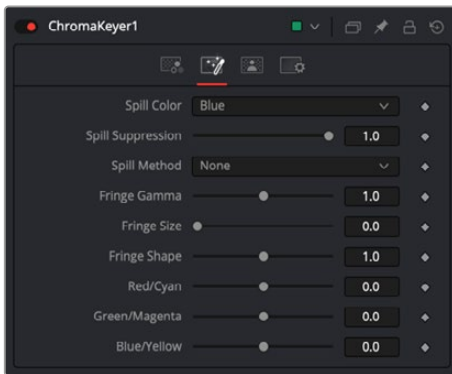
The Chroma Key Image tab

Image Tab

The Image tab primarily handles removing spill color on the foreground subject. Color spill occurs when light containing the color you are removing is reflected onto the foreground subject.

Spill Color

This menu selects the color used as the base for all spill suppression techniques.

Spill Suppression

This slider sets the amount of spill suppression applied to the foreground subject.

When this slider is set to 0, no spill suppression is applied.

Spill Method

This menu selects the strength of the algorithm used to apply spill suppression to the image.

— **None:** None is selected when no spill suppression is required.
— **Rare:** This removes very little of the spill color and is the lightest of all methods.
— **Medium:** This works best for green screens.
— **Well Done:** This works best for blue screens.
— **Burnt:** This works best for blue. Use this mode only for very troublesome shots. Most likely you will have to add strong color correction after the key to get, for example, your skin tones back.

Fringe Gamma

This control is used to adjust the brightness of the fringe or halo that surrounds the keyed image.

Fringe Size

This expands and contracts the size of the fringe or halo surrounding the keyed image.

Fringe Shape

Fringe Shape forces the fringe toward the external edge of the image or toward the inner edge of the fringe. Its effect is most noticeable while the Fringe Size slider's value is large.

Cyan/Red, Magenta/Green, and Yellow/Blue

Use these three controls to color correct the fringe of the image. This is useful for correcting semitransparent pixels that still contain color from the original background to match the new background.
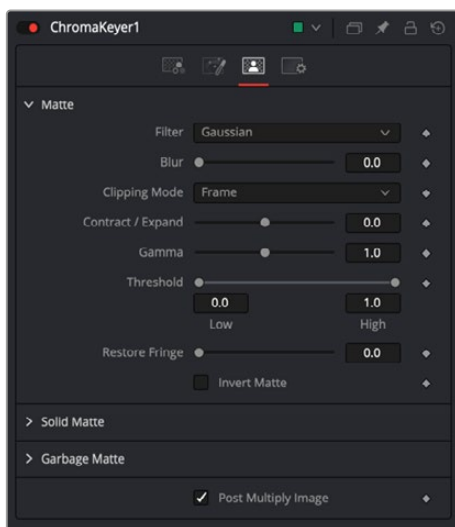
The Chroma Key Matte tab

Matte Tab

The Matte tab refines the softness, density, and overall fit of the resulting matte.

Filter

This control selects the filtering algorithm used when applying blur to the matte.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.

— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.

— **Multi-box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."

— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Blur

Matte Blur blurs the edge of the matte based on the Filter menu setting. A value of zero results in a sharp, cutout-like hard edge. The higher the value, the more blur applied to the matte.

Clipping Mode

This option determines how edges are handled when performing domain of definition rendering. This is profoundly important when blurring the matte, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame will be treated as black/transparent.

— **Domain:** Setting this option to Domain will respect the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None will not perform any source image clipping at all. This means that any data required to process the node's effect that would normally be outside the upstream DoD will be treated as black/transparent.

Contract/Expand

This slider shrinks or grows the semitransparent areas of the matte. Values above 0.0 expand the matte, while values below 0.0 contract it.

This control is usually used in conjunction with the Matte Blur to take the hard edge of a matte and reduce fringing. Since this control affects only semitransparent areas, it will have no effect on a matte's hard edge.

Gamma

Matte Gamma raises or lowers the values of the matte in the semitransparent areas. Higher values cause the gray areas to become more opaque, and lower values cause the gray areas to become more transparent. Completely black or white regions of the matte remain unaffected.

Since this control affects only semitransparent areas, it will have no effect on a matte's hard edge.

Threshold

This range slider sets the lower threshold using the handle on the left and sets the upper threshold using the handle on the right.

Any value below the lower threshold setting becomes black or transparent in the matte.

Any value above the upper threshold setting becomes white or opaque in the matte. All values within the range maintain their relative transparency values.

This control is often used to reject salt and pepper noise in the matte.

Restore Fringe

This restores the edge of the matte around the keyed subject. Often when keying, the edge of the subject where you have hair is clipped out. Restore Fringe brings back that edge while keeping the matte solid.

Invert Matte

When this checkbox is selected, the alpha channel created by the keyer is inverted, causing all transparent areas to be opaque and all opaque areas to be transparent.

Solid Matte

Solid Mattes are mask nodes or images connected to the solid matte input on the node. The solid matte is applied directly to the alpha channel of the image. Generally, solid mattes are used to hold out keying in areas you want to remain opaque, such as someone with blue eyes against a blue screen.

Enabling Invert will invert the solid matte, before it is combined with the source alpha.

Garbage Matte

Garbage mattes are mask nodes or images connected to the garbage matte input on the node. The garbage matte is applied directly to the alpha channel of the image. Generally, garbage mattes are used to remove unwanted elements that cannot be keyed, such as microphones and booms. They are also used to fill in areas that contain the color being keyed but that you wish to maintain.

Garbage mattes of different modes cannot be mixed within a single tool. A Matte Control node is often used after a Keyer node to add a garbage matte with the opposite effect of the matte applied to the keyer.

Enabling Invert will invert the garbage matte, before it is combined with the source alpha.

Post-Multiply Image

Select this option to cause the keyer to multiply the color channels of the image against the alpha channel it creates for the image. This option is usually enabled and is on by default.

Deselect this checkbox and the image can no longer be considered premultiplied for purposes of merging it with other images. Use the Subtractive option of the Merge node instead of the Additive option.

For more information on these Merge node settings, *see Chapter 95, "Composite Nodes,"* in the DaVinci Resolve Reference Manual, or Chapter 33 in the Fusion Reference Manual.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other matte nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.
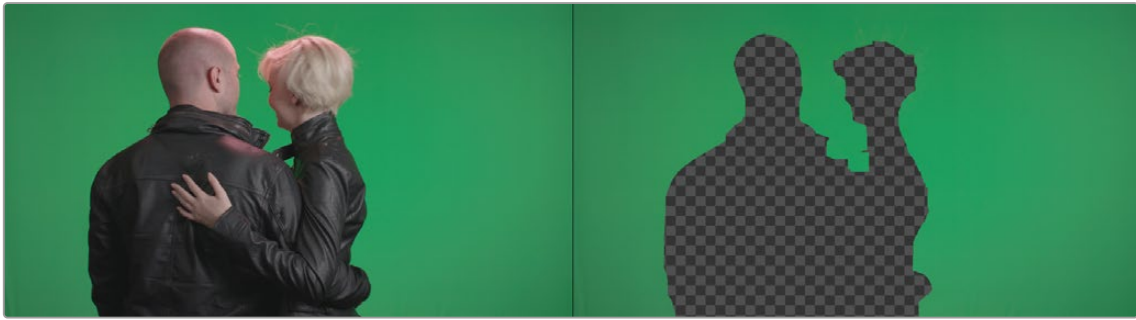
Clean Plate
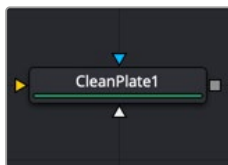
The Clean Plate node

Clean Plate Node Introduction

The Clean Plate tool is a pre-keying node used to generate an image of the green or blue color screen to smooth out the lighting differences. The output of the Clean Plate is later connected to the Clean Plate input on the Delta Keyer so it can key fine detail without choking or clipping the matte.

How to Create a Clean Plate

Creating a clean plate is the opposite of creating a key. When keying, you try to remove the green or blue color. When creating a clean plate, you try to keep as much of the blue- or green-screen as possible. By box selecting areas of the screen color in the viewer, you end up with an image of the green/blue screen. A transparent cutout represents everything that is not part of the blue or green screen.

Once you have the selection, the Erode control expands the pre-matte, removing any small pixels of non-green/blue screen around the edges. Then, growing the pre-matte fills in the holes until you have a solid blue or green image.

Inputs

The Clean Plate node includes three inputs in the Node Editor.

— **Input:** The orange input accepts a 2D image that contains the green or blue screen.

— **Garbage Matte:** The white garbage matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be excluded from the clean plate. For a clean plate, garbage mattes should contain areas that are not part of the blue or green screen.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input limits the pixels where the clean plate is generated. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Clean Plate node and the Delta Keyer are two separate branches stemming from the main image you want to key. The green-screen or blue-screen clip is breached to connect to both the orange image input on the Clean Plate and the orange image input on the Delta Keyer. The output of the clean plate is then connected to the magenta clean plate input on the Delta Keyer. The output of the Delta Keyer is then used as the foreground to a Merge.
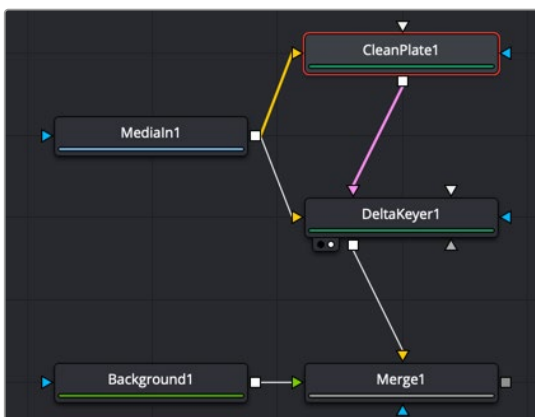
A Clean Plate connected to a Delta Keyer

Inspector
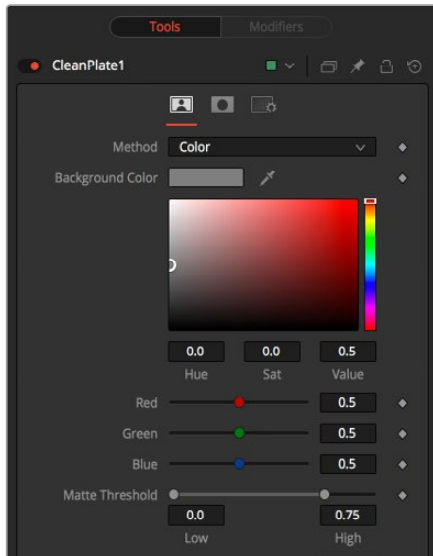
The Clean Plate tab

Plate Tab

The Plate tab contains the primary tools for creating a clean plate. Using this tab, you drag over the areas in the viewer, and then use the Erode and Grow Edges sliders to create the clean plate.

Method

The Method menu selects the type of color selection you use when sampling colors in the viewer.

— **Color:** Color uses a difference method to separate the background color. This works well on screen colors that are even.

— **Ranges:** Ranges uses a chroma range method to separate the background color. This is a better option for shadowed screen or screens that have different colors.

Matte Threshold

This range slider sets the lower threshold using the handle on the left and sets the upper threshold using the handle on the right.

Any value below the lower threshold becomes black or transparent in the matte.

Any value above the upper threshold becomes white or opaque in the matte. All values within the range maintain their relative transparency values. This control is often used to reject salt and pepper noise in the matte.

Erode

The Erode slider decreases the size of the screen area. It is used to eat away at small non-screen color pixels that may interfere with creating a smooth green- or blue-screen clean plate.

Crop

Crop trims in from the edges of the image.

Grow Edges
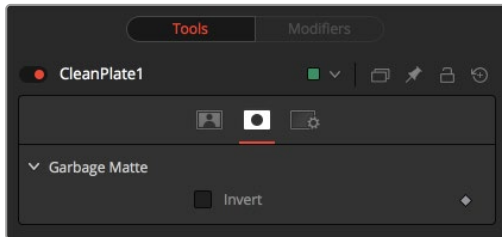
The Grow Edges slider expands the color of the edges to fill in holes until fully green or blue screen is created.

Fill

The Fill checkbox fills in remaining holes with color from the surrounding screen color.

Time Mode

— **Sequence:** Generates a new clean plate every frame.

— **Hold Frame:** Holds the clean plate at a single frame.

The Clean Plate Mask tab

Mask Tab

The Mask tab is used to invert the mask connected to the garbage mask input on the node.
The garbage mask can be applied to clear areas before growing edges or filling remaining holes.
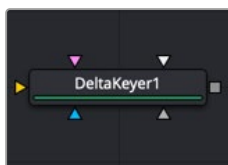
Invert

Invert uses the transparent parts of the mask to clear the image.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other matte nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Delta Keyer

The Delta Keyer node

Delta Keyer Node Introduction

The Delta Keyer is an advanced color difference keyer, with many features and controls for tuning the matte and separating the subject from a blue or green screen.

It contains several keying systems: the Key tab is the master difference keyer and Pre-Matte is a built-in clean plate to smooth out screen color. Tuning, Fringe, and Matte finish the keying process. The tabs are laid out in the rough order you tend to use them.

Inputs

The Delta Keyer node includes five inputs in the Node Editor.

— **Input:** The orange input accepts a 2D image that contains the color you want to be keyed for transparency.

— **Garbage Matte:** The gray garbage matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be made transparent. The garbage matte is applied directly to the alpha channel of the image.

— **Solid Matte:** The white solid matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be fully opaque.

— **Clean Plate:** Accepts the resulting image from the Clean Plate node.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input limits the pixels where the keying occurs. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

A single Delta keyer rarely gets perfect results because most green- or blue-screen shots have problems that the keyer is not made to handle. Keyers often need the help of garbage mattes or solid mattes created with a Polygon or B-Spline node. Shots can also require more than just one keyer to achieve perfect results. That is shown in one of the examples below, where one Delta Keyer is used to create a hard inner white matte, while a second Delta Keyer is used to capture the soft edges. The core matte below is then connected into the solid matte input on the second Delta Keyer. Color correction, including spill suppression, is best handled as a separate branch in your node tree. Separating your matte from your color correction using underlays makes it clear what branch of nodes is performing what operation.
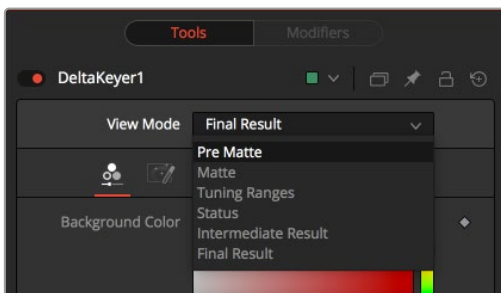

Starting point for a Delta Keyer setup
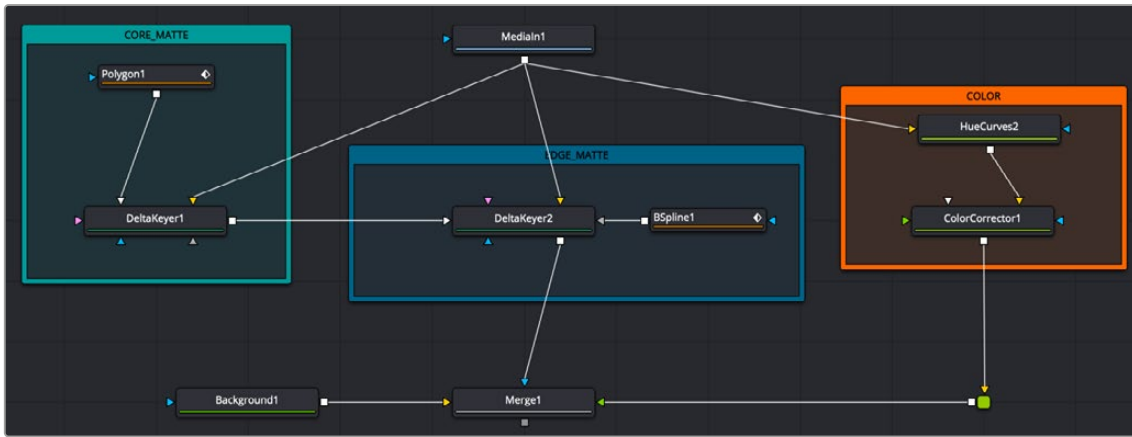
An expanded key setup with two Delta Keyers, a Polygon and B-Spline
nodes for mattes, and a branch for color correction

Inspector

The Delta Keyer View Mode menu

Key Tab View Mode

At the top of the Inspector is the View Mode menu. The default selection shows the final result. You
can change the view to see various intermediate stages of the keying process.

— **Pre Matte:** Displays the output of the Pre Matte key.

— **Matte:** Displays the alpha of the key before being combined with solid and garbage masks.
When displaying the matte, set the viewer to show the alpha channel.

— **Tuning Ranges:** Displays a false color for Shadow, Midtone, and Highlight ranges of the image.
Shadows are in the red channel, Midtones are in the green channel, and Highlights are in
the blue channel.

— **Status:** Displays information to indicate areas that are solid, transparent, or in between. It also
displays areas that have been affected by matte adjustments, such as thresholding or erode/dilate
and areas affected by the solid mask.

— **Intermediate Result:** The untouched source image color channels combined with the final matte.
Optionally, combine this output with additional Delta Keyer nodes.

— **Final Result:** The final keyed image with spill suppression, ready to merge onto a scene.
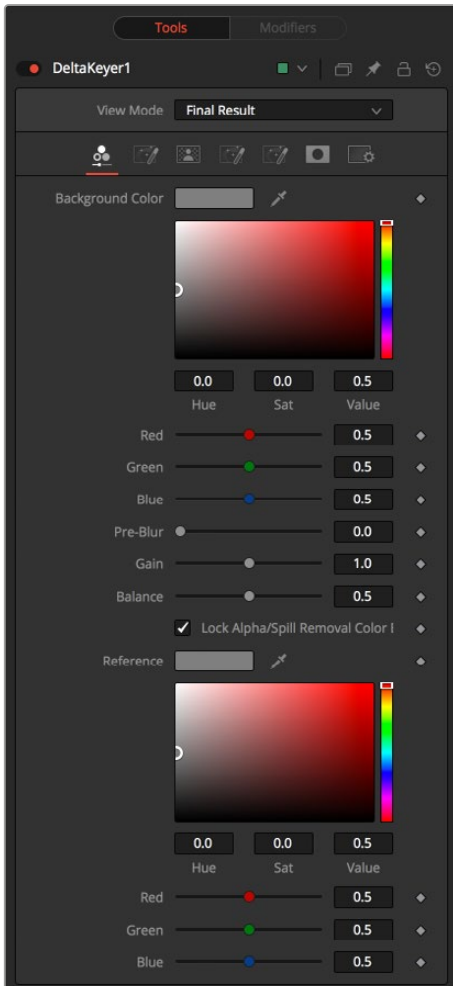
The Delta Keyer Key tab

Key Tab

The Key tab is where most keying begins. It is used to select the screen color.

Background Color

This is the color of the blue or green screen, sometimes called the screen color. To create the key with the Delta Keyer, use the background color Eyedropper to select the screen color from the image.

Pre-Blur

Applies a blur before generating the alpha. This can help with certain types of noise, edge enhancements, and artifacts in the source image.

Gain

Gain increases the influence of the screen color, causing those areas to become more transparent.

Balance

A color difference keyer, like the Delta Keyer, compares the differences between the dominant channel determined by the selected background color and the other two channels. Adjusting balance determines the proportions of the other two channels. A value of 0 uses the minimum of the other two channels, where a value of 1 uses the maximum. A value of 0.5 uses half of each.

Lock Alpha/Spill Removal Color Balance Reference

Unlocking this allows you to use different color references when generating the alpha and when determining how much of the background color to subtract from the image. When enabled, spill and color are combined.

Color Balance Reference

The Color balance reference controls adjust for lighting or white balance that might be reducing background color purity and saturation. A correction is applied based on the reference of a neutral-colored object when generating the key and determining the amount of background color subtraction, without altering the background color that is subtracted.
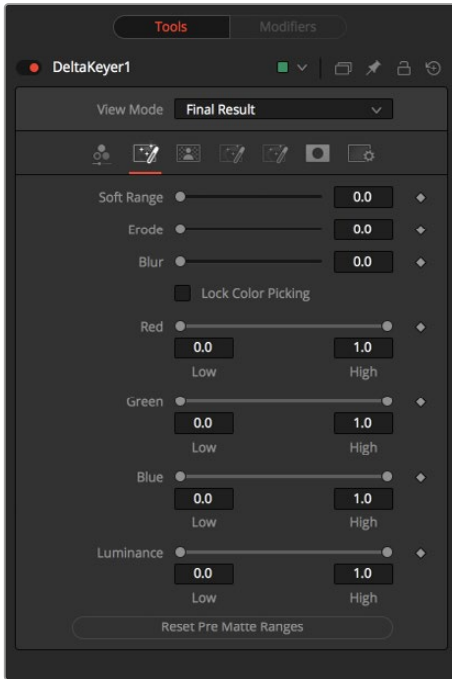
The Delta Keyer Pre Matte tab

Pre Matte Tab

The Pre Matte tab is in some ways a garbage matte keyer that occurs before the delta keyer process. It is used to even out the screen color before a more delicate key is done. After selecting the initial screen color, you can switch to the Pre Matte tab and view the PreMatte Out from view mode menu at the top of the Inspector. The Pre Matte works on ranges of color, so you start by dragging a bounding box around areas where lighting is different from the initial screen color pick. Use the Soft range to expand, and Erode to cut away the edges.

Once you have a more even screen selection, you can move to the Matte tab.

Soft Range

The Soft Range extends the range of selected color and rolloff of the screen color.

Erode

Erode contracts the edge of the pre matte, so the edge detail does not clip.

Blur

This softens the edges of the pre matte.

Pre Matte Range

These controls update automatically to represent the current color selection. The controls tweak the selection slightly, although selecting colors in the viewer is all that is required.

Lock Color Picking

When this checkbox is activated, it prevents the accidental growth of the selected range by selecting more colors from the view. It is a good idea to activate this checkbox after selecting the color for the matte. All other controls in the node remain editable.

Reset Pre Matte Ranges

This button discards all color selection by resetting the ranges but maintains all other slider and control values.
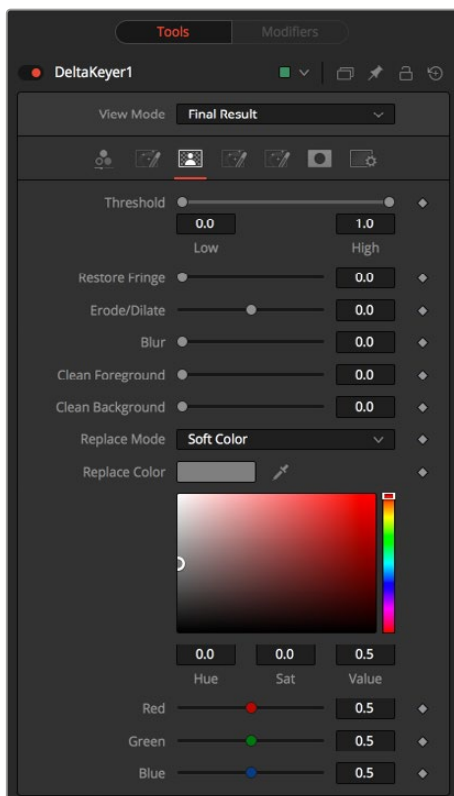
The Delta Keyer Matte tab

Matte Tab

The Matte tab refines the alpha of the key, combined with any solid and garbage masks connected to the node. When using the matte tab, set the viewer to display the alpha channel of the Delta Keyer's final output.

Threshold

This range slider sets the lower threshold using the handle on the left and sets the upper threshold using the handle on the right.

Any value below the lower threshold setting becomes black or transparent in the matte.

Any value above the upper threshold setting becomes white or opaque in the matte. All values within the range maintain their relative transparency values.

Restore Fringe

This restores the edge of the matte around the keyed subject. Often when keying, clipping occurs around the edge of the subject where there is hair. Restore Fringe brings back that edge while keeping the matte solid.

Erode/Dilate

Expands or contracts the matte.

Blur

Softens the matte.

Clean Foreground

Fills slightly transparent (light gray) areas of the matte.

**Clean Background**

Clips the bottom dark range of the matte.

Replace Mode

Determines how matte adjustments restore color to the image.

— **None:** No color replacement. Matte processing does not affect the color.
— **Source:** The color from the original image.
— **Hard Color:** A solid color.
— **Soft Color:** A solid color weighted by how much background color was originally removed.

Replace Color

The color used with the Hard Color and Soft Color replace modes.
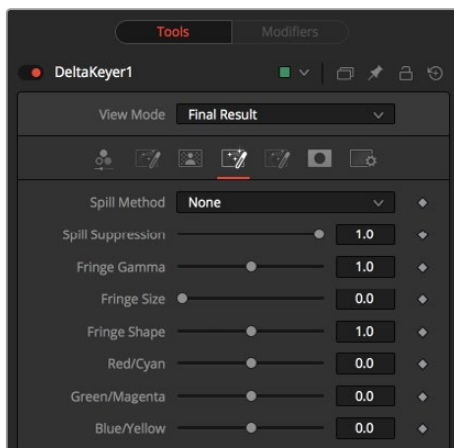
The Delta Keyer Fringe tab

Fringe Tab

The Fringe tab handles the majority of spill suppression in the Delta Keyer. Spill suppression is a form of color correction that attempts to remove the screen color from the fringe of the matte.

Spill is the transmission of the screen color through the semitransparent areas of the alpha channel. In the case of blue- or green-screen keying, this usually causes the color of the background to become apparent in the edges of the foreground subject.

Spill Method

This selects the strength of the algorithm used to apply spill suppression to the image.

— **None:** None is selected when no spill suppression is required.

— **Rare:** This removes very little of the spill color and is the lightest of all methods.

— **Medium:** This works best for green screens.

— **Well Done:** This works best for blue screens.

— **Burnt:** This works best for blue screens. Use this mode only for very troublesome shots.

Spill Suppression

When this slider is set to 0, no spill suppression is applied to the image. Increasing the slider increases the strength of the spill method.

Fringe Gamma

This control can be used to adjust the brightness of the fringe or halo that surrounds the keyed image.

Fringe Size

This expands and contracts the size of the fringe or halo surrounding the keyed image.

Fringe Shape

Fringe Shape presses the fringe toward the external edge of the image or pulls it toward the inner edge of the fringe. Its effect is most noticeable while the Fringe Size value is large.

Cyan/Red, Magenta/Green, and Yellow/Blue

Use these three controls to color correct the fringe of the image.

This is useful for correcting semitransparent pixels that still contain color from the original background to match the new background.
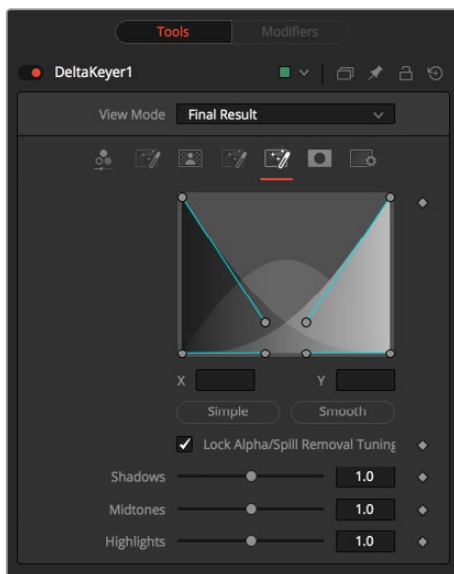
The Delta Keyer Tuning tab

Tuning Tab

The Tuning tab is an advanced tab that allows you to determine the size of the shadow, midtone, and highlight ranges. By modifying the ranges, you can select the strength of the matte and spill suppression based on tonal values.

Range Controls

The range controls define how much color is considered to fall into the shadows, midtones, and highlights areas of the image. The spline controls allow for easy adjusting of the tonal ranges of each Shadow and Highlight tonal map.

Simple/Smooth

The Simple button sets the range to be linear. The Smooth button sets a smooth tonal gradient for the ranges.

Lock Alpha/Spill Removal Tuning

When this checkbox is disabled, separate tuning controls are used when generating the alpha and when determining how much of the background color to subtract from the image.

— **Shadows:** Adjusts the strength of the key in darker areas of the background.
— **Midtones:** Adjusts the strength of the key in midtone areas of the background.
— **Highlights:** Adjusts the strength of the key in brighter areas of the background.
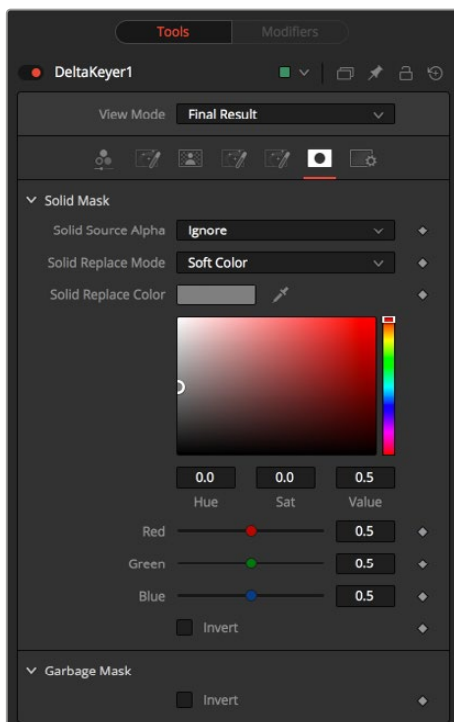
The Delta Keyer Mask tab

Mask Tab

The Mask tab determines how the solid and garbage mattes are applied to the key.

Solid Source Alpha

Used to combine the existing alpha from the source image with the solid mask.

— **Ignore:** Does not combine the alpha from the source image.
— **Add:** Solid areas of the source image alpha are made solid in the solid mask.
— **Subtract:** Transparent areas of the source image alpha are made transparent in the solid mask.

Solid Replace Mode

This determines how the solid mask restores color to the image.

— **None:** No color replacement. The solid mask does not affect the color.

— **Source:** The color from the original image.

— **Hard Color:** A solid color.

— **Soft Color:** A solid color weighted by how much background color was originally removed.

Solid Replace Color

The color used with the Hard Color and Soft Color replace modes.

— **Invert:** Inverts the solid mask, before it is combined with the source alpha.
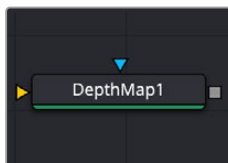
Garbage Mask

— **Invert:** Normally, solid areas of a garbage mask remove the image. When inverted, the transparent areas of the mask remove the image.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other matte nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Depth Map [DMp]

The Depth Map node

Depth Map Node Introduction

Depth Map creates an Alpha channel based on the perceived distance of objects in your clip. By being able to isolate a specific depth region, the opportunities to manipulate the resulting image are greatly expanded. For example, combined with a Blur node you could simulate a depth of field effect, but confined only to the distant background of your shot. You could enhance a foreground object, like a person, increasing contrast, saturation, and sharpness, similar to using a qualifier. You can use a Depth Map to fix video issues, for example fixing a color temperature problem by isolating a far window that was shot in daylight, causing background actors to be tinted blue, while leaving the foreground subject that was shot with studio lighting unaffected.

The resulting Depth Map Alpha channel is visualized as a black and white image, with white being the area that is affected by the resulting changes and black areas remaining unchanged.

Inputs

The Depth Map node includes two inputs in the Node Editor.

— **Input:** The yellow input accepts a 2D image that contains the shot you wish to analyze for depth.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input limits the pixels where the difference matte occurs. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Depth Map is designed to take an input from an image source and create an Alpha channel based on the perceived distance from the camera. Then it will feed that Alpha to downstream nodes. In the example below, the Depth Map is used to isolate the background of the image and apply a color grade to just the background, while leaving the foreground actors unaffected as they move through the frame.
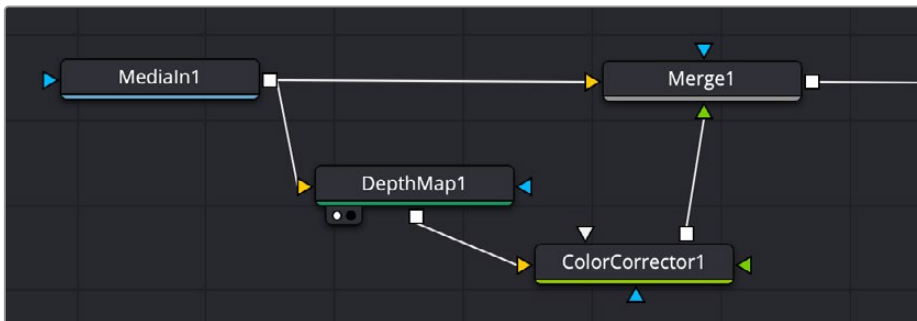
A sample Depth Map node structure for isolating subjects based on distance from the camera

Inspector
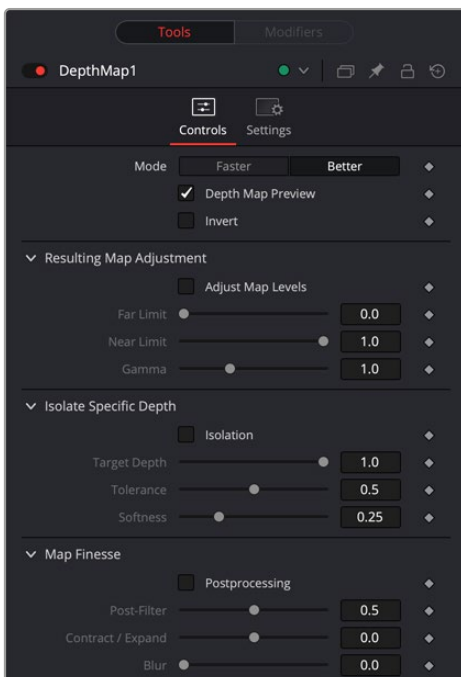
The Depth Map Controls tab

Controls Tab

The Controls tab in the Depth Map contains all the parameters for adjusting the quality of the resulting matte.

— **Mode:** Depth Map is a very computationally intensive effect. The quality setting allows a Faster mode to speed up the responsiveness to adjustments, while the default Better mode gives the best results and should be turned on when the adjustments are finished.

— **Depth Map Preview:** By default this box is checked and shows you the current Depth Map for making adjustments. When this check box is disabled, the resulting Alpha can then be used for grading on other nodes.

— **Invert:** Checking this box reverses the Depth Map, switching its transparent and opaque regions.

Resulting Map Adjustment

These controls let you determine how the Depth Map's contrast is adjusted.

— **Adjust Map Levels:** When deselected (default), all scaling is turned off, allowing you to adjust the full range of the Depth Map. When enabled, this option clips the Depth Map's levels to 0 and 1. This functions as a preview of what will happen to the Depth Map when used as an Alpha channel where the values are always clipped to 0 and 1. Checking this box also activates the tools below.

— **Far Limit:** This control adjusts the black levels of the Depth Map.

— **Near Limit:** This control adjusts the white levels of the Depth Map.

— **Gamma:** This control adjusts the intermediate depth values to be brighter or dimmer compared to the fixed black and white levels.

Isolate Specific Depth

These controls allow you to sweep backwards and forwards in the scene by depth, allowing you to isolate a specific depth range to adjust.

— **Isolation:** This checkbox turns the depth isolation tools on or off.

— **Target Depth:** This controls the specific depth you want to isolate. 1 is fully in the foreground, while 0 is fully in the background.

— **Tolerance:** Sets the range to either side of the Target Depth to include in the Depth Map.

— **Softness:** This sets a subtle ramp-in and ramp-out of the selected range, giving it a more organic selection.

Map Finesse

These controls modify the resulting Depth Map's Alpha channel for use in grading.

— **Post Processing:** This control turns the Map Finesse tools on or off.

— **Post-Filter:** This control blends the map to the smooth areas and edges of the image. It is used to prevent later grading effects from visibly varying within the region.

— **Contract / Expand:** This control dilates or erodes the overall shape at the edges; useful for fine tuning the boundary between the affected and unaffected regions of the map.

— **Blur:** This control softens the boundary of the map, allowing it to blend more smoothly into the resulting image.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Matte nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Difference Keyer [DfK]
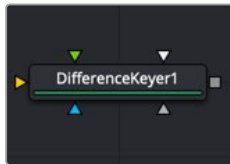
The Difference Keyer node

Difference Keyer Node Introduction

Difference keying is a process that produces a matte based on the differences between two images. A difference key uses two input images: one containing the subject with the background and another containing the background without the subject.

Although the process sounds reasonable at first glance, subtle variations in the camera position from shot to shot usually make it difficult to get clean results. Think of the futile attempt of trying to key smoke in front of a brick wall and using a clean plate of the brick wall as your difference input. Part of the wall's structure is always visible in this keying method. Instead, a Difference Keyer is often used to produce a rough matte that is combined with other nodes to produce a more detailed matte.
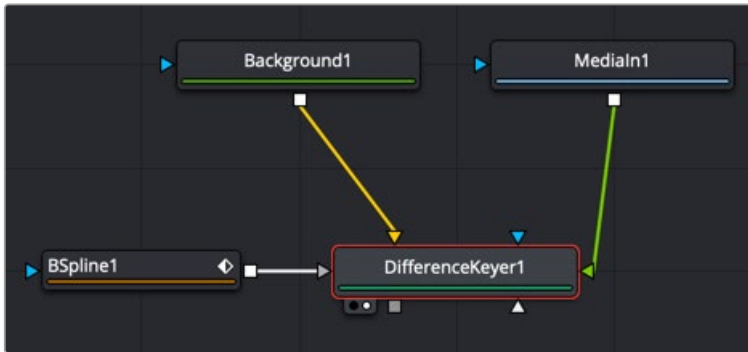
Inputs

The Difference Keyer node includes four inputs in the Node Editor.

— **Background:** The orange background input accepts a 2D image that contains just the set without your subject.

— **Foreground:** The green foreground input accepts a 2D image that contains the shot with your subject in the frame.

— **Garbage Matte:** The gray garbage matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be made transparent.

— **Solid Matte:** The white solid matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be fully opaque.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input limits the pixels where the difference matte occurs. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

When you do not have content shot on a blue or green screen, the Difference Keyer can be one node in a chain of many used to extract an object from the background. The example below has the MediaIn1 as the main subject and a clean background shot without the subject (Background). A B-Spline is used to limit the area the Difference Keyer must deal with for extraction. The result is a matte that can be used to help but not solve the key.

A Difference Keyer with two inputs: one of the subject on a background and the other of just the background

Inspector
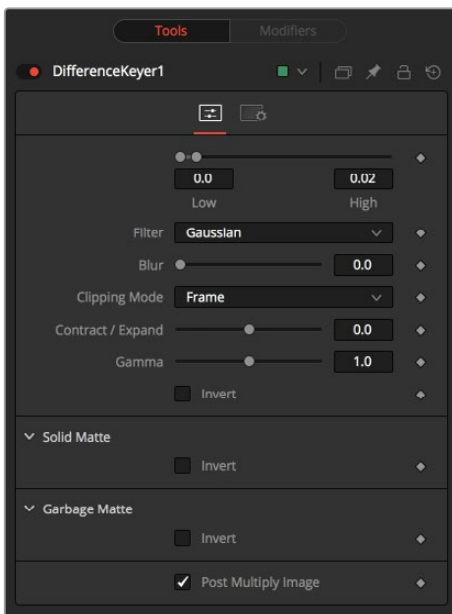
The Difference Keyer Controls tab

Controls Tab

The Controls tab in the Difference Keyer contains all the parameters for adjusting the quality of the matte.

Threshold

This range slider sets the lower threshold using the handle on the left and sets the upper threshold using the handle on the right. Adjusting them defines a range of difference values between the images to create a matte.

A difference below the lower threshold setting becomes black or transparent in the matte.

Any difference above the upper threshold setting becomes white or opaque in the matte.

The difference values in the range in between create a grayscale matte.

Filter

This control selects the filtering algorithm used when applying a blur to the matte.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.
— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.
— **Multi-box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."
— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Blur

This blurs the edge of the matte using the method selected in the Filter menu. A value of zero results in a sharp, cutout-like hard edge. The higher the value, the more blur.

Clipping Mode

This option determines how edges are handled when performing domain of definition rendering. This is profoundly important when blurring the matte, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.
— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.
— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that would usually be outside the upstream DoD is treated as black/transparent.

Contract/Expand

This slider shrinks or grows the semitransparent areas of the matte. Values above 0.0 expand the matte, while values below 0.0 contract it.

This control is usually used in conjunction with the blur to take the hard edge of a matte and reduce fringing. Since this control affects only semitransparent areas, it has no effect on a matte's hard edge.

Gamma

Matte Gamma raises or lowers the values of the matte in the semitransparent areas. Higher values cause the gray areas to be more opaque, and lower values cause the gray areas to be more transparent. Wholly black or white regions of the matte remain unaffected.

Invert

Selecting this checkbox inverts the matte, causing all transparent areas to be opaque and all opaque areas to be transparent.

Solid Matte

Solid Mattes are mask nodes or images connected to the solid matte input on the node. The solid matte is applied directly to the alpha channel of the image. Generally, solid mattes are used to hold out keying in areas you want to remain opaque, such as someone with blue eyes against a blue screen.

Enabling Invert, inverts the solid matte before it is combined with the source alpha.

Garbage Matte

Garbage mattes are mask nodes or images connected to the garbage matte input on the node. The garbage matte is applied directly to the alpha channel of the image. Generally, garbage mattes are used to remove unwanted elements that cannot be keyed, such as microphones and booms. They are also used to fill in areas that contain the color being keyed but that you wish to maintain.

Garbage mattes of different modes cannot be mixed within a single tool. A Matte Control node is often used after a Keyer node to add a garbage matte with the opposite effect of the matte applied to the keyer.

Enabling Invert inverts the garbage matte before it is combined with the source alpha.

Post-Multiply Image

Select this option to cause the keyer to multiply the color channels of the image against the alpha channel it creates for the image. This option is usually enabled and is on by default.

Deselect this checkbox and the image can no longer be considered premultiplied for purposes of merging it with other images. Use the Subtractive option of the Merge node instead of the Additive option.

For more information on these Merge node settings, *see Chapter 95, "Composite Nodes,"* in the DaVinci Resolve Reference Manual, or Chapter 33 in the Fusion Reference Manual.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Matte nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Luma Keyer [LKy]

The Luma Keyer node

Luma Keyer Node Introduction

The Luma Keyer node uses the overall luminance of an image to create an Alpha channel. The label of this node may seem misleading since it allows pulling mattes from almost any channel. In some respects, it is more accurate to call this node an all-purpose channel keyer, but its primary purpose is for extracting alpha channels based on luminance

Inputs

The Luma Keyer node includes four inputs in the Node Editor.

— **Input:** The orange input accepts a 2D image that contains the luminance values you want to be keyed for transparency.

— **Garbage Matte:** The gray garbage matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be made transparent. The garbage matte is applied directly to the alpha channel of the image.

— **Solid Matte:** The white solid matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be fully opaque.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input limits the pixels where the luminance key occurs. An effects mask is applied to the tool after the tool is processed.
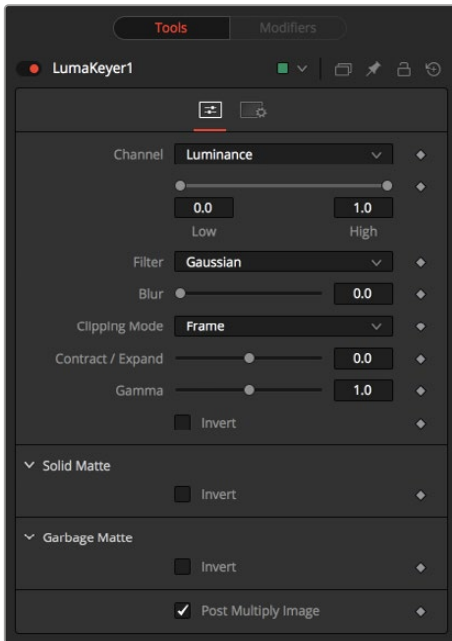
Basic Node Setup

The Luma Keyer takes an input from an image with significant luminance difference to extract a key. You can then use the output of the Luma Keyer into any mask input.
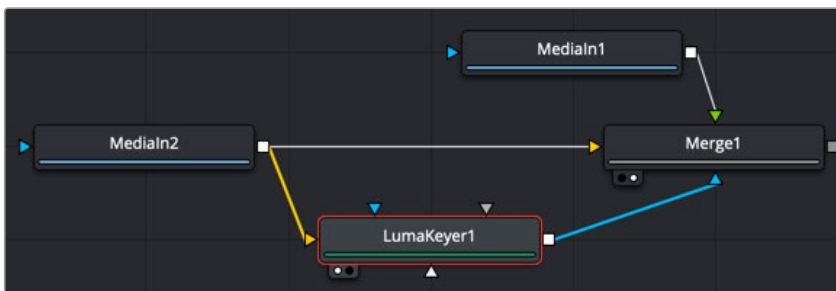
A Luma Keyer output connecting into an effect mask on a Merge node

Inspector

The Luma Keyer Controls tab

Controls Tab

The Controls tab in the Luma Keyer contains all the parameters for adjusting the quality of the matte.

Channel

This menu selects the color channel used for creating the matte. Select from the Red, Green, Blue, Alpha, Hue, Luminance, Saturation, and Depth (Z-buffer) channels.

Threshold

This range slider sets the lower threshold using the handle on the left and sets the upper threshold using the handle on the right. Adjusting them defines a range of luminance values to create a matte.

A value below the lower threshold setting becomes black or transparent in the matte.

Any value above the upper threshold setting becomes white or opaque in the matte.

The values within the range create a grayscale matte.

Filter

This control selects the filtering algorithm used when applying a blur to the matte.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.
— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.
— **Multi-box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."
— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Blur

This blurs the edge of the matte using the method selected in the Filter menu. A value of zero results in a sharp, cutout-like hard edge. The higher the value, the more blur.

Clipping Mode

This option determines how edges are handled when performing domain of definition rendering. This is profoundly important when blurring the matte, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that is usually outside the upstream DoD is treated as black/transparent.

Contract/Expand

This slider shrinks or grows the semitransparent areas of the matte. Values above 0.0 expand the matte, while values below 0.0 contract it.

This control is usually used in conjunction with the blur to take the hard edge of a matte and reduce fringing. Since this control affects only semitransparent areas, it has no effect on a matte's hard edge.

Gamma

Matte Gamma raises or lowers the values of the matte in the semitransparent areas. Higher values cause the gray areas to be more opaque, and lower values cause the gray areas to be more transparent. Wholly black or white regions of the matte remain unaffected.

Invert

Selecting this checkbox inverts the matte, causing all transparent areas to be opaque and all opaque areas to be transparent.

Solid Matte

Solid mattes are mask nodes or images connected to the solid matte input on the node. The solid matte is applied directly to the alpha channel of the image. Generally, solid mattes are used to hold out keying in areas you want to remain opaque, such as someone with blue eyes against a blue screen.

Enabling Invert inverts the solid matte before it is combined with the source alpha.

Garbage Matte

Garbage mattes are mask nodes or images connected to the garbage matte input on the node. The garbage matte is applied directly to the alpha channel of the image. Generally, garbage mattes are used to remove unwanted elements that cannot be keyed, such as microphones and booms. They are also used to fill in areas that contain the color being keyed but that you wish to maintain.

Garbage mattes of different modes cannot be mixed within a single tool. A Matte Control node is often used after a Keyer node to add a garbage matte with the opposite effect of the matte applied to the keyer.

Enabling Invert inverts the garbage matte before it is combined with the source alpha.

Post-Multiply Image

Select this option to cause the keyer to multiply the color channels of the image against the alpha channel it creates for the image. This option is usually enabled and is on by default.

Deselect this checkbox and the image can no longer be considered premultiplied for purposes of merging it with other images. Use the Subtractive option of the Merge node instead of the Additive option.

For more information on these Merge node settings, *see Chapter 95, "Composite Nodes,"* in the DaVinci Resolve Reference Manual, or Chapter 33 in the Fusion Reference Manual.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Matte nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Magic Mask [MagM]
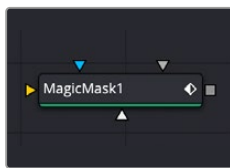
The Magic Mask node

Magic Mask Node Introduction

The Magic Mask palette uses the DaVinci Neural Engine to automagically create a mask to isolate an object in the frame, guided by user-applied paint strokes to identify the subject for isolation. Masks can be generated for either an entire object or just a part of it. This allows you to create complicated masks of complex shapes that may be difficult to isolate with other tools. For example, using the Magic Mask to modify only the wood grain section of this guitar.

(Left) Multiple strokes isolating the wood grain of the guitar, while ignoring the musician's arm; (Right) The finished shot, with a warmer wood grain

> **NOTE:** The Magic Mask tool in Fusion is based on the Object mode of the Magic Mask tool in the Color page and is designed to be used for any subject that is not a person. Subjects like cars, pets, and food are all good candidates for Object mode. If you are trying to isolate a human being, using Magic Mask's Person mode in the Color page will likely give you better results.

Drawing Strokes in the Viewer

The Magic Mask tool works by drawing lines directly into the viewer. Once a stroke is drawn, the viewer will automatically show the mask you've created. You can draw multiple strokes to create a mask in an additive fashion and use negative strokes to remove areas from the mask. It is best to choose a frame to draw on where the object is completely in view and unobstructed.

— **To draw a positive stroke to add an object to a mask:** Load the Magic Mask node in a viewer, and left-click and drag the stroke across the object you want to choose. Alternatively you can change the Stroke Mode to Add in the Inspector and left-click and drag to draw a new positive stroke. Positive strokes are colored blue.

— **To draw a negative stroke to remove an object from a mask:** Load the Magic Mask node in a viewer, and Option-left-click and drag the stroke across the object you want to remove. Alternatively you can change the Stroke Mode to Subtract in the Inspector and left-click and drag to draw a new negative stroke. Negative strokes are colored red.

— **To delete a stroke (or group of strokes) to remove them from the mask:** Shift-drag around a stroke or group of strokes to select them (green), and press the delete key to remove them. Alternatively you can change the Stroke Mode to Select in the Inspector and left-click and drag a selection window to select one or more strokes. Clicking the delete button removes them.

Drawing positive strokes will select areas of similar contrast and color, allowing you to link complex shapes together. Generally you will need more strokes to actively define a complex object, due to the greater variety of the shapes involved. Stroke position is usually more important than stroke length.

Drawing negative strokes removes areas from the object that you don't want to isolate. This can be something simple, like removing the wheels of a car from a mask, or more complicated like removing specific books from a mask of a bookshelf. Stroke position is usually more important than stroke length.

(Left) Multiple strokes isolating the car's body, while removing the wheels and cabin; (Right) The finished shot, fed through a color corrector node to change the color of just the car's body

Inputs

The Magic Mask node includes four inputs in the Node Editor.

— **Input:** The orange input accepts a 2D image that contains the luminance values you want to be keyed for transparency.

— **Garbage Matte:** The gray garbage matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be made transparent. The garbage matte is applied directly to the Alpha channel of the image.

— **Solid Matte:** The white solid matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be fully opaque.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input limits the pixels where the luminance key occurs. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Magic Mask takes RGB inputs from other nodes and lets you isolate specific objects in the frame to use as a mask. Tracking the mask as it changes over time is built into the tool. You can then use the output of the Magic Mask into any mask input.
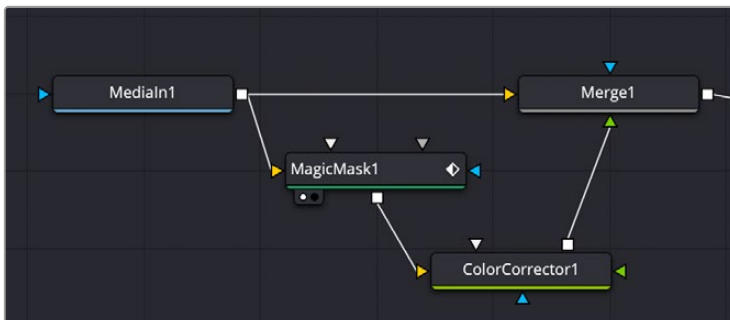
A Magic Mask node, feeding a matte to a Color Corrector
connected to the foreground of a Merge node

Inspector

The Magic Mask Tracking tab

Tracking Tab

You can think of each stroke you've drawn as a persistent eyedropper that samples the image that overlaps it. The mask that results from all strokes' collective analysis of the image is generated live over each frame of the clip. This means that if the camera or the subject moves, you need to motion track or otherwise adjust the position of each stroke to follow along with the motion, so the subject continues to be correctly identified. You also need to make sure that each stroke is able to analyze what it's supposed to, and turn off strokes that can't for whatever reason.

— **Tracking Controls:** These buttons control the tracking direction, from left to right:

  — **Track Reverse:** Continuously tracks from the current frame all the way to the beginning of the clip.

  — **Track Reverse One Frame:** Tracks one frame backwards and stops. Useful if you're tracking frame-by-frame to watch the progress of a particularly complicated bit of motion. If something goes wrong, you can back up to the last frame where the stroke was able to properly track the subject, and drag the stroke to a better location using the pointer to make it follow the subject properly. If necessary, you can go a frame at a time, dragging the stroke to a better position every time it fails to follow the feature you're using it to isolate.

  — **Stop Tracking:** Stops tracking, in cases where there's a problem with the track, and you want to make a change.

  — **Track Forward Then Reverse:** Tracks from the current frame all the way to the end of the clip, then returns to the original tracking point and tracks backwards to the beginning of the clip.

  — **Track Forward One Frame:** Tracks one frame forward and stops. Useful if you're tracking frame-by-frame to watch the progress of a particularly complicated bit of motion. If necessary, you can go a frame at a time, dragging the stroke to a better position every time it fails to follow the feature you're using it to isolate.

  — **Track Forward:** Continuously tracks from the current frame all the way to the end of the clip.

— **Go To Frame:** These buttons snap the playhead to the selected frame, from left to right:

  — **First Frame of Tracked Area:** Moves the playhead to the first tracked frame of a range of tracked frames in preparation for tracking backwards if there are untracked frames at the beginning of the clip.

  — **Reference Frame:** Moves the playhead to the frame on which you initially drew the strokes.

  — **Last Frame of Tracked Area:** Moves the playhead to the last tracked frame of a range of tracked frames in preparation for tracking forwards if there are untracked frames at the end of the clip.

— **Stroke Mode:** These buttons let you change and modify the strokes drawn in the Viewer

  — **Add:** Lets you add an additional stroke (blue) that determines what in the frame will be included in the mask.

  — **Subtract:** Lets you add an additional stroke (red) that determines what in the frame will be excluded from the mask.

  — **Select:** Lets you draw a selection rectangle around single or multiple strokes to select them (green).

  — **Delete:** Deletes all selected strokes (green).

— **Clear Strokes:** These buttons remove all strokes from the selected range.
  — **Current:** Removes strokes from the frame under the playhead.
  — **Range:** Removes strokes from the selected render range.
  — **All:** Removes all strokes from the entire clip.

— **Disk Cache:** These buttons allow you to control what's stored in the disk cache.
  — **Regenerate All:** Rebuilds the disk cache.
  — **Clear:** Deletes the frames in the current cache.

— **Reference Time:** The Reference Time determines the frame where the initial strokes are drawn. It is also the time from which tracking begins. The reference frame cannot be changed once it has been set without destroying all pre-existing tracking information.
— **Processed Frames:** Displays the range of frames that have already been tracked. Start is the earliest frame tracked and end is the last frame tracked. These fields are not user editable.
— **Mode:** Two options let you choose a tradeoff between quality and performance. Faster lets you generate a lower quality mask more quickly that's suitable for garbage matting. Better generates a higher quality mask with more detail that's more processor-intensive.

Post-Multiply Image

Select this option to cause the keyer to multiply the color channels of the image against the Alpha channel it creates for the image. This option is usually enabled and is on by default.

Deselect this checkbox and the image can no longer be considered premultiplied for purposes of merging it with other images. Use the Subtractive option of the Merge node instead of the Additive option.

For more information on these Merge node settings, *see Chapter 95, "Composite Nodes,"* in the DaVinci Resolve Reference Manual, or Chapter 33 in the Fusion Reference Manual.

Matte Tab

The Matte tab refines the alpha of the key, combined with any solid and garbage masks connected to the node. When using the Matte tab, set the viewer to display the Alpha channel of Magic Mask's final output.

— **Filter:** Selects the Filter that is used when blurring the matte.
  — **Box Blur:** This option applies a Box Blur effect to the whole image. This method is faster than the Gaussian blur but produces a lower-quality result.
  — **Bartlett:** Bartlett applies a more subtle, anti-aliased blur filter.
  — **Multi-Box:** Multi-Box uses a box filter layered in multiple passes to approximate a Gaussian shape. With a moderate number of passes (e.g., four), a high-quality blur can be obtained, often faster than the Gaussian filter and without any ringing.
  — **Gaussian:** Gaussian applies a smooth, symmetrical blur filter, using a sophisticated constant-time Gaussian approximation algorithm.
  — **Fast Gaussian:** Gaussian applies a smooth, symmetrical blur filter, using a sophisticated constant-time Gaussian approximation algorithm. This mode is the default filter method.

— **Blur:** This blurs the edge of the matte using a standard, constant speed Gaussian blur. A value of zero results in a sharp, cutout-like hard edge. The higher the value, the more blur is applied to the matte.

— **Erode/Dilate:** This shrinks or grows the matte. Eroding the matte reveals more of the foreground input, while dilating the matte reveals more of the background input. Values above 0.0 dilate the matte and values below 0.0 erode it.

— **Gamma:** This raises or lowers the values of the matte in the semitransparent areas. Higher values cause the gray areas to become more opaque, and lower values cause the gray areas to become more transparent. Completely black or white regions of the matte remain unaffected.

— **Threshold:** Any value below the lower threshold becomes black or transparent in the matte. Any value above the upper threshold becomes white or opaque in the matte. All values within the range maintain their relative transparency values.

— **Restore Fringe:** This restores the edge of the matte around the keyed subject. Often when keying, the edge of the subject where you have hair is clipped out. Restore Fringe brings back that edge while keeping the matte solid.

— **Invert Matte:** When this checkbox is selected, the Alpha channel of the image is inverted, causing all transparent areas to be opaque and all opaque areas to be transparent.

Solid Matte

Solid mattes are mask nodes or images connected to the solid matte input on the node. The solid matte is applied directly to the Alpha channel of the image. Generally, solid mattes are used to hold out keying in areas you want to remain opaque, such as someone with blue eyes against a blue screen.

Enabling Invert inverts the solid matte before it is combined with the source alpha.

Garbage Matte

Garbage mattes are mask nodes or images connected to the garbage matte input on the node. The garbage matte is applied directly to the Alpha channel of the image. Generally, garbage mattes are used to remove unwanted elements that cannot be keyed, such as microphones and booms. They are also used to fill in areas that contain the color being keyed but that you wish to maintain.

Garbage mattes of different modes cannot be mixed within a single tool. A Matte Control node is often used after a Keyer node to add a garbage matte with the opposite effect of the matte applied to the keyer.

Enabling Invert inverts the garbage matte before it is combined with the source alpha.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Matte nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Matte Control [Mat]

The Matte Control node

Matte Control Node Introduction

Keyer nodes are used to create an alpha channel on an image that does not have one. The Matte Control node is used to combine and manipulate the alpha channels embedded in images as well as masks created by masking tools.

Typically, you add this node to copy a color channel or alpha channel from the foreground input to the background input, or to combine alpha channels from the two images.
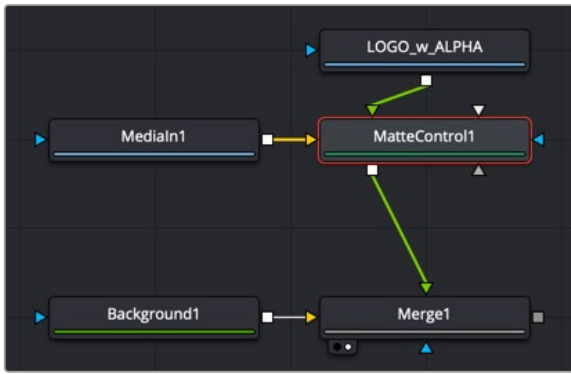
Inputs

The Matte Control node includes four inputs in the Node Editor.

— **Background:** The orange background input accepts a 2D image that receives the foreground image alpha channel (or some other channel you want to copy to the background).

— **Foreground:** The green foreground input accepts a 2D image that contains an alpha channel (or some other channel) you want to be applied to the background image.

— **Garbage Matte:** The gray garbage matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the foreground/background combination that fall within the matte to be made transparent.

— **Solid Matte:** The white solid matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the foreground/background combination that fall within the matte to be fully opaque.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input limits the pixels where the matte control occurs. An effects mask is applied to the tool after the tool is processed.
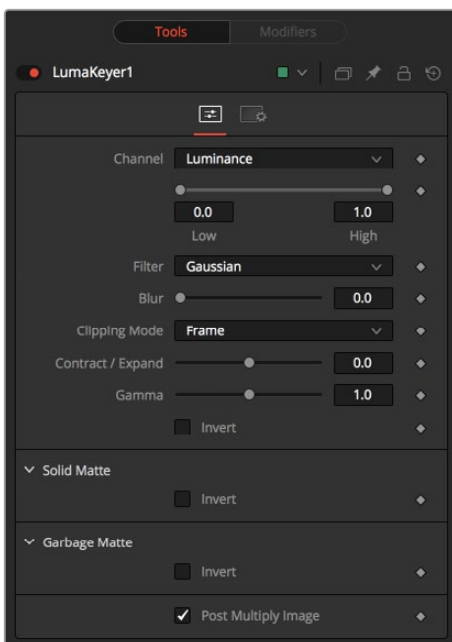
Basic Node Setup

Below, the Matte Control node is set up to copy the foreground (green) input's alpha channel into the background (orange) input. The output of the Matte Control is then an image with an alpha channel used as the foreground composite in the Merge node.

A Matte Control embedded an alpha from the foreground input to the background input

Inspector

The Matte Control Matte tab

Matte Tab

The Matte tab combines and modifies alpha or color channels from an image in the foreground input with the background image.

Combine

Use this menu to select which operation is applied. The default is set to None for no operation.

— **None:** This causes the foreground image to be ignored.

— **Combine Red:** This combines the foreground red channel to the background alpha channel.

— **Combine Green:** This combines the foreground green channel to the background alpha channel.

— **Combine Blue:** This combines the foreground blue channel with the background alpha channel.

— **Combine Alpha:** This combines the foreground alpha channel with the background alpha channel.

— **Solid:** This causes the background alpha channel to become completely opaque.

— **Clear:** This causes the background alpha channel to become completely transparent.

Combine Operation

Use this menu to select the method used to combine the foreground channel with the background.

— **Copy:** This copies the foreground source over the background alpha, overwriting any existing alpha in the background.

— **Add:** This adds the foreground source to the background alpha.

— **Subtract:** This subtracts the foreground source from the background alpha.

— **Inverse Subtract:** This subtracts the background alpha from the foreground source.

— **Maximum:** This compares the foreground source and the background alpha and takes the value from the pixel with the highest value.

— **Minimum:** This compares the foreground source and the background alpha and takes the value from the pixel with the lowest value.

— **And:** This performs a logical AND on the two values.

— **Or:** This performs a logical OR on the values.

— **Merge Over:** This merges the foreground source channel over the background alpha channel.

— **Merge Under:** This merges the foreground source channel under the background alpha channel.

Filter

Selects the Filter that is used when blurring the matte.

— **Box Blur:** This option applies a Box Blur effect to the whole image. This method is faster than the Gaussian blur but produces a lower-quality result.

— **Bartlett:** Bartlett applies a more subtle, anti-aliased blur filter.

— **Multi-Box:** Multi-Box uses a box filter layered in multiple passes to approximate a Gaussian shape. With a moderate number of passes (e.g., four), a high-quality blur can be obtained, often faster than the Gaussian filter and without any ringing.

— **Gaussian:** Gaussian applies a smooth, symmetrical blur filter, using a sophisticated constant-time Gaussian approximation algorithm. In extreme cases, this algorithm may exhibit ringing; see below for a discussion of this. This mode is the default filter method.

Blur

This blurs the edge of the matte using a standard, constant speed Gaussian blur. A value of zero results in a sharp, cutout-like hard edge. The higher the value, the more blur is applied to the matte.

Clipping Mode

This option determines how edges are handled when performing domain of definition rendering. This is profoundly important when blurring the matte, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that is usually outside the upstream DoD is treated as black/transparent.

Contract/Expand

This shrinks or grows the matte similar to an Erode Dilate node. Contracting the matte reveals more of the foreground input, while expanding the matte reveals more of the background input. Values above 0.0 expand the matte, and values below 0.0 contract it.

Gamma

This raises or lowers the values of the matte in the semitransparent areas. Higher values cause the gray areas to become more opaque, and lower values cause the gray areas to become more transparent. Completely black or white regions of the matte remain unaffected.

Threshold

Any value below the lower threshold becomes black or transparent in the matte. Any value above the upper threshold becomes white or opaque in the matte. All values within the range maintain their relative transparency values.

Restore Fringe

This restores the edge of the matte around the keyed subject. Often when keying, the edge of the subject where you have hair is clipped out. Restore Fringe brings back that edge while keeping the matte solid.

Invert Matte

When this checkbox is selected, the alpha channel of the image is inverted, causing all transparent areas to be opaque and all opaque areas to be transparent.

Solid Matte

Solid mattes are mask nodes or images connected to the solid matte input on the node. The solid matte is applied directly to the alpha channel of the image. Generally, solid mattes are used to hold out areas you want to remain opaque, such as someone with blue eyes against a blue screen.

Enabling Invert inverts the solid matte before it is combined with the source alpha.

Garbage Matte

Garbage mattes are mask nodes or images connected to the garbage matte input on the node. The garbage matte is applied directly to the alpha channel of the image. Generally, garbage mattes are used to remove unwanted elements that cannot be keyed, such as microphones and booms. They are also used to fill in areas that contain the color being keyed but that you wish to maintain.

Garbage mattes of different modes cannot be mixed within a single tool. A Matte Control node is often used after a Keyer node to add a garbage matte with the opposite effect of the matte applied to the keyer.

Enabling Invert inverts the garbage matte before it is combined with the source alpha.

Post-Multiply Image

Selecting this option multiplies the color channels of the image against the alpha channel it creates for the image. This option is usually enabled and is on by default.

Deselect this checkbox and the image can no longer be considered premultiplied for purposes of merging it with other images. Use the Subtractive option of the Merge node instead of the Additive option.

For more information on these Merge node settings, *see Chapter 95, "Composite Nodes,"* in the DaVinci Resolve Reference Manual, or Chapter 33 in the Fusion Reference Manual.
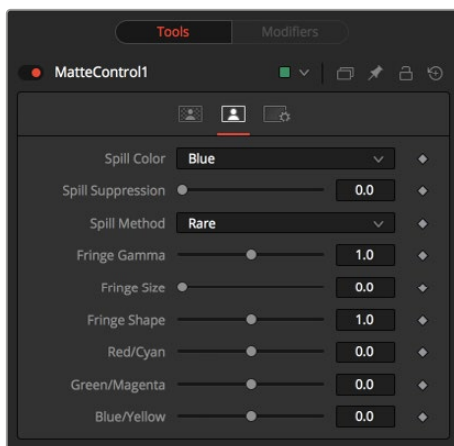
The Matte Control Spill tab

Spill Tab

The Spill tab handles spill suppression in the Matte Control. Spill suppression is a form of color correction that attempts to remove the screen color from the fringe of the matte.

Spill is the transmission of the screen color through the semitransparent areas of the alpha channel. In the case of blue- or green-screen keying, this usually causes the color of the background to become apparent in the edges of the foreground subject.

Spill Color

This menu selects the color used as the base for all spill suppression techniques.

Spill Suppression

When this slider is set to 0, no spill suppression is applied to the image. Increasing the slider increases the strength of the spill method.

Spill Method

This selects the strength of the algorithm used to apply spill suppression to the image.

— **None:** None is selected when no spill suppression is required.
— **Rare:** This removes very little of the spill color and is the lightest of all methods.
— **Medium:** This works best for green screens.
— **Well Done:** This works best for blue screens.
— **Burnt:** This works best for blue screen. Use this mode only for very troublesome shots.

Fringe Gamma

This control can be used to adjust the brightness of the fringe or halo that surrounds the keyed image.

Fringe Size

This expands and contracts the size of the fringe or halo surrounding the keyed image.

Fringe Shape

Fringe Shape presses the fringe toward the external edge of the image or pulls it toward the inner edge of the fringe. Its effect is most noticeable while the Fringe Size value is large.

Cyan/Red, Magenta/Green, and Yellow/Blue

Use these three controls to color correct the fringe of the image.

This is useful for correcting semitransparent pixels that still contain color from the original background to match the new background.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Matte nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Primatte [Pri]
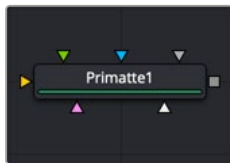
The Primatte node

> **NOTE:** Primatte is only available in Fusion Studio.

Primatte Node Introduction

Primatte is an advanced keying tool for Fusion Studio. To use Primatte effectively, you must understand how it works. Using a series of selection buttons, Primatte assigns RGB pixels into one of the four specific zones.

— **Zone 1:** Complete background image.
— **Zone 2:** Foreground image with spill suppression and transparency.
— **Zone 3:** Foreground image with spill suppression only.
— **Zone 4:** Complete foreground image.

Depending on the type of blue- or green-screen content, you may find that the Delta Keyer or the Primatte keyer handles the specific keying task better. There is no one-solution-fits-all when it comes to keying, and in some cases, the combination of the two keyers may prove to be the best solution.

> **NOTE:** Primatte is distributed and licensed by IMAGICA Corp. of America, Los Angeles, CA, USA. Primatte was developed by and is a trademark of IMAGICA Corp., Tokyo, Japan.

Inputs

The Primatte node includes six inputs in the Node Editor. Unlike every other tool in Fusion, the primary orange input is labeled as the Foreground input, since it accepts the green-screen or blue-screen image. The background input on the Primatte node is the green input; this is an optional input that allows Primatte to create the final merged composite.

— **Foreground Input:** The orange input accepts a 2D image that contains blue or green screen.
— **Background Input:** The green (optional) input accepts a 2D image layered as the background in the composite. If no image is connected, Primatte outputs the keyed foreground. Connecting an image to the background input activates Primatte's advanced edge blending options.
— **Replacement Image:** The magenta (optional) input accepts a 2D image used as a source of Primatte's spill suppression color correction.
— **Garbage Matte:** The gray garbage matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be made transparent. The garbage matte is applied directly to the alpha channel of the image.
— **Solid Matte:** The white solid matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be fully opaque.
— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input limits the pixels where the keying occurs. An effects mask is applied to the tool after the tool is processed.

> **NOTE:** Connecting the background input without connecting the replacement image input uses the background image as the replacement image surf spill suppression.

Basic Node Setup

A single Primatte keyer can rarely get perfect results because most green- or blue-screen shots have problems the keyer is not made to handle. Keyers often need the help of garbage mattes or solid mattes created with a Polygon or B-Spline node. Shots can also require more than just one keyer to achieve perfect results. Below, the Primatte node has the blue-screen content connected to the orange input. Unlike other Fusion nodes, the foreground gets connected to the orange input. The result is an image with alpha that can then be connected into the foreground of a Merge node.
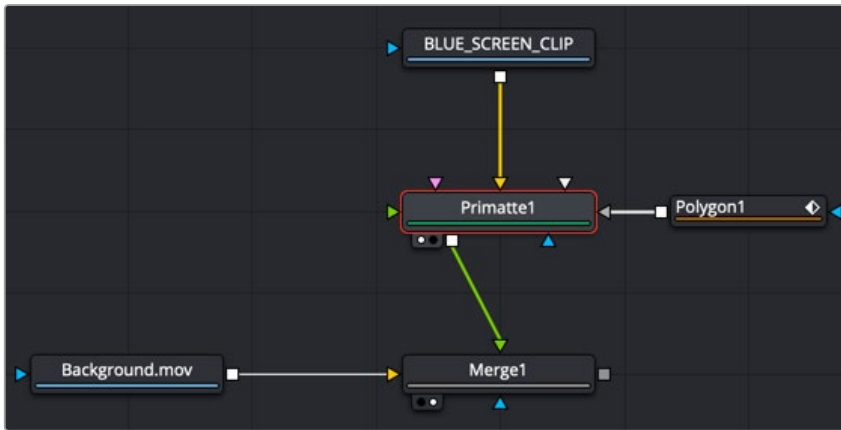
A Primatte node combined with a polygon matte as a garbage
matte and connected to the foreground of a Merge

Primatte Tab View Mode

At the top of the Inspector is the View Mode menu. The default selection shows the final Composite
result. You can change the view to see various intermediate stages of the keying process.
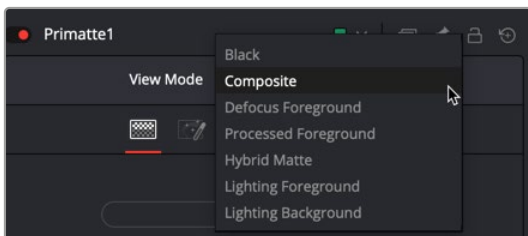
The Primatte tab View mode

— **Black:** Displays the foreground subject on a black or transparent background.
— **Composite:** The final keyed image with spill suppression, composited over the image connected
to the green Background Input on the node.
— **Defocus Foreground:** Displays the output of the Pre Matte key.
— **Processed Foreground:** Displays the alpha of the key before being combined with solid and
garbage masks. When displaying the matte, set the viewer to show the alpha channel.
— **Hybrid Matte:** Displays the matte generated when the Hybrid Rendering checkbox is enabled.
Best viewed when adjusting the Hybrid Blur and Hybrid Erode sliders.
— **Lighting Foreground:** Displays the foreground subject over the optimized artificial backing
screen that the Adjust Lighting mode creates.
— **Lighting Background:** Displays the optimized artificial backing screen that the Adjust Lighting
mode creates.
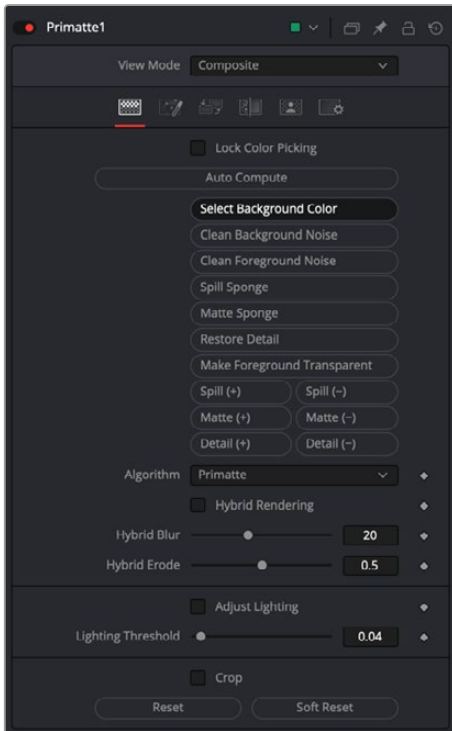
The Primatte tab Inspector

Primatte Tab

The core functionality for Primatte is found in the Primatte tab. The basic workflow is based on selecting one of the operational mode buttons and then scrubbing over areas in the viewer.

Lock Color Picking

Activate this button once you finished adjusting your key to prevent making accidental changes in the viewer.

Auto Compute

The Auto Compute button is likely the first button pressed when starting to key your footage. Primatte automagically analyzes the original foreground image, determines the backing color, and sets it as the central backing color. Then, using that information, another analysis determines the foreground areas. A Clean FG Noise operation is performed using the newly determined foreground areas, and Primatte renders the composite.

> **NOTE:** The Auto Compute button may make the next three buttons—Select Background Color, Clean Background Noise, and Clean Foreground Noise—unnecessary and make your keying operation much more straightforward. Clicking Auto Compute automatically senses the backing screen color, eliminates it, and even gets rid of some foreground and background noise. If you get good results, then jump ahead to the Spill Removal tools. If you don't get satisfactory results, continue from this point using the three buttons described below.

Select Background Color

Clicking the Select Background Color button allows you to select the screen color by scrubbing in the viewer. It uses the traditional Primatte method of taking the sampled backing screen color, projecting a line in the opposite direction on the hue wheel, and generating artificial pixels that may represent the FG object. Then, using the artificially generated foreground pixels, it internally does the Clean FG Noise operation and creates the shape of the middle and outer polyhedrons. It then renders the composite using these generated polyhedrons. This does not automatically use the Adjust Lighting functionality, as it must be selected in a separate operation.

Clean Background Noise

Clicking this button helps to remove any white regions in the dark screen area ("noise"), or shades of the screen color that did not get picked up on the first sample. Once you click the button, scrub the mouse pointer over areas in the viewer to sample white-ish noise regions.

Clean Foreground Noise

If there are dark transparent regions in the middle of the mostly white opaque foreground object, click the Clean Foreground Noise button and scrub over the dark pixels in the foreground area until that area is as white as possible.

Spill Sponge

The Spill Sponge is the quickest method for removing color spill on your subject. Click the Spill Sponge button and scrub the mouse pointer over a screen color pixel, and the screen color disappears from the selected color region and is replaced by a complementary color, a selected color, or a color from a replacement image. These options are set in the Replace tab. Additionally, use the tools under the Fine Tuning tab or use the Spill(+) and Split(-) features to adjust the spill.

Matte Sponge

Sometimes in the Primatte operation, a 100% opaque, foreground area (all white) becomes slightly transparent (gray). To clean those transparent areas, click the Matte Sponge button and scrub over the transparent pixels. All the spill-suppression information remains intact.

Restore Detail

Clicking Restore Detail and scrubbing over background regions in the viewer turns completely transparent areas translucent. This operation is useful for restoring lost hair details, thin wisps of smoke, and the like.

Make Foreground Transparent

When this button is selected, the opaque foreground color region sampled in the viewer becomes slightly translucent. This operation is useful for the subtle tuning of foreground subjects, which are otherwise 100 percent covered with smoke or clouds. It can be used only one time on a particular color. For a more flexible way to thin out a color region, and to be able to take multiple samples, use the Matte(-) tool.

Spill(+)

Clicking the Spill(+) button returns the color spill to the sampled pixel color (and all colors like it) in the amount of one Primatte increment. This tool can be used to move the sampled color more in the direction of the color in the original foreground image. It can be used to nullify a Spill(-) step.

Spill(-)

Clicking the Spill(-) button removes from the sampled pixel color (and all colors like it) in the amount of one Primatte increment. If spill color remains, another click using this operational mode tool removes more of the color spill. Continue using this tool until all color spill has been removed from the sampled color region.

Matte(+)

Clicking the Matte(+) button makes the matte more opaque for the sampled pixel color (and all colors like it) in the amount of one Primatte increment. If the matte is still too translucent or thin, another click using this operational mode tool makes the sampled color region even more opaque. This can be used to thicken smoke or make a shadow darker to match shadows in the background imagery. It can only make these adjustments to the density of the color region on the original foreground image. It can be used to nullify a Matte(-) step.

Matte(-)

Clicking the Matte(+) button makes the matte more translucent for the sampled pixel color (and all colors like it) in the amount of one Primatte increment. If the matte is still too opaque, another click using this operational mode tool makes the sampled color region even more translucent. This can be used to thin out smoke or make a shadow thinner to match shadows in the background imagery.

Detail(+)

When this button is selected, the foreground detail becomes less visible for the sampled pixel color (and all colors like it) in the amount of one Primatte increment. If there is still too much detail, another click using this operational mode tool makes more of it disappear. This can be used to remove smoke or wisps of hair from the composite. Sample where detail is visible, and it disappears. This is for moving color regions into the 100% background region. It can be used to nullify a Detail(-) step.

Detail(-)

When this button is selected, foreground detail becomes more visible for the sampled pixel color (and all colors like it) in the amount of one Primatte increment. If detail is still missing, another click using this operational mode tool makes detail more visible. This can be used to restore lost smoke or wisps of hair. Sample where the smoke or hair just disappears and it returns to visibility. Use this for restoring color regions that were moved into the 100% background region. It may start to bring in background noise if shooting conditions were not ideal on the foreground image.

Algorithms

There are three keying algorithms available in the Primatte keyer:

— **Primatte:** The Primatte algorithm mode delivers the best results and supports both the Solid Color and the Complement Color spill suppression methods. This algorithm uses three multifaceted polyhedrons (as described later in this section) to separate the 3D RGB colorspace. It is also the default algorithm mode and, because it is computationally intensive, it may take the longest to render.

— **Primatte RT:** Primatte RT is the simplest algorithm and therefore the fastest. It uses only a single planar surface to separate the 3D RGB colorspace (as described later in this section) and, as a result, does not separate the foreground from the backing screen as carefully as the above Primatte algorithm. Another disadvantage of the Primatte RT algorithm is that it does not work well with less saturated backing screen colors, and it does not support the Complement Color spill suppression method.

— **Primatte RT+:** Primatte RT+ is in between the above two options. It uses a six planar surface color separation algorithm (as described later in this section) and delivers results in between the other two options in both quality and performance. Another disadvantage of the Primatte RT+ algorithm is that it does not work well with less saturated backing screen colors, and it does not support the Complement Color spill suppression method.

Hybrid Rendering

After sampling the backing screen color and producing acceptable edges around the foreground object, you sometimes find a transparent area within the foreground subject. This can occur when the foreground subject contains a color that is close to the backing screen color. Removing this transparency with the Clean FG Noise mode can cause the edge of the foreground subject to pick up a fringe that is close to the backing screen color. Removing the fringe is very difficult without sacrificing quality somewhere else on the image. The Hybrid Render mode internally creates two keying operations: Body and Edge. The optimized Edge operation gets the best edge around the foreground subject without any fringe effect. The Body operation deals with transparency within the foreground subject. The resultant matte is created by combining these two mattes, and then blurring and eroding the foreground subject in the Body matte and combining it with the edge matte.

To use Hybrid Rendering, start by keying the main foreground area using the Select Background Color mode (or any of the other Primatte backing screen detection methods). Activate the Hybrid Rendering checkbox. Lastly, select the Clean FG Noise button and scrub over the transparent area. The Hybrid Render mode performs the "Body/Edge" operation. The result is a final composite with perfect edges around the foreground subject with a solid foreground subject.

Hybrid Blur

Blurs the Body matte that has been automatically generated when Hybrid Rendering is activated.

Hybrid Erode

This slider dilates or erodes the Hybrid matte. You can view the results by selecting Hybrid matte in the View Mode menu.

Adjust Lighting

Before applying the Adjust Lighting operation, it is necessary to determine the backing screen color using Auto Compute or Select Background Color. After performing one of those operations, click on the Adjust Lighting button. Primatte generates an artificial clean plate and uses it to generate an evenly lit backing screen behind the foreground object. The default setting should detect all the areas that contain foreground pixels and deliver a smooth backing screen for the keying.

Lighting Threshold

Should Adjust Lighting fail to produce a smoother backing screen, adjust the Lighting Threshold slider while viewing the Lighting Background setting in the View Mode menu. This displays the optimized artificial backing screen that the Adjust Lighting mode creates.

Crop

This button reveals the Crop sliders to create a rectangular garbage matte with the Primatte node. As opposed to Fusion's Crop tool, this does not change the actual image size.

Reset

Resets all the Primatte key control data back to a blue- or green-screen.

Soft Reset

Resets just the Primatte parameters used since the Select Background Color operation
was last completed.
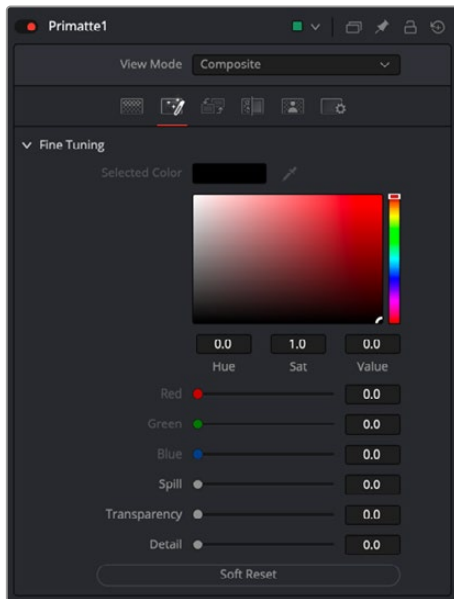
The Primatte Fine Tuning tab

Fine Tuning Tab

The Fine Tuning tab can make refined adjustments to the spill suppression, density of the matte, and semitransparent areas. These sliders provide a bit more granularity over the Spill(+)(-), Matte(+)(-) and Detail(+)(-) buttons in the Primatte tab.

Selected Color

This shows the color selected (or registered) by the scrubbing in the viewer while the Fine Tuning tab is selected.

Fine Tuning Sliders

The color of the scrubbed pixel is registered as a reference color for fine tuning. It is displayed in the Color swatch. To perform the tuning operation, sample a color region on the image, and adjust one of the Fine Tuning sliders to achieve the desired effect.

Spill

The Spill slider can be used to remove spill from the selected color region. The more to the right the slider moves, the more spill is removed. The more to the left the slider moves, the closer the color component of the selected region is to the color in the original foreground image. If moving the slider to the right does not remove the spill, resample the color region and move the slider again.

These slider operations are additive. The result achieved by moving the slider to the right can also be achieved by clicking on the color region using the Spill(-) operational mode.

Transparency

The Transparency slider makes the matte more translucent in the selected color region. Moving this slider to the right makes the selected color region more transparent. Moving the slider to the left makes the matte more opaque. If moving the slider to the right does not make the color region

translucent enough, resample the color region and again move the slider to the right. These slider operations are additive. The result achieved by moving the slider to the right can also be achieved by clicking on the color region using the Matte(-) operational mode.

Detail

The Detail slider can be used to restore lost detail. After selecting a color region, moving this slider to the left makes the selected color region more visible. Moving the slider to the right makes the color region less visible. If moving the slider to the left does not make the color region visible enough, resample the color region and again move the slider to the left.

These slider operations are additive. This result achieved by moving the slider to the left can also be achieved by clicking on the color region using the Detail(-) operational mode.
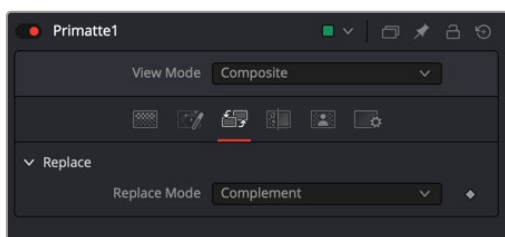
The Primatte Replace tab

Replace Tab

The Replace tab allows you to choose between the three methods of color spill replacement as covered in detail in the Spill Sponge section above. There are three options for the replacement color when removing the spill. These options are selected from the Replace mode menu.

Replace Mode

— **Complement:** Replaces the spill color with the complement of the screen color. This mode maintains fine foreground detail and delivers the best-quality results. If foreground spill is not a significant problem, this mode is the one that should be used. However, If the spill intensity on the foreground image is rather significant, this mode may often introduce serious noise in the resultant composite.

— **Image:** Replaces the spill color with colors from a defocused version of the background image or the Replace image, if one is connected to the Replace input (magenta) on the node. This mode results in a good color tone on the foreground subject even with a high-contrast background. On the negative side, the Image mode occasionally loses the fine edge detail of the foreground subjects. Another problem can occur if you later change the size of the foreground image against the background. Since the background/foreground alignment would change, the applied color tone from the defocused image might not match the new alignment.

— **Color:** Replaces the spill color with a solid color. When this option is selected, a color swatch and R,G,B sliders are displayed for selecting the color. Changing the palette color for the solid replacement, you can select a good spill replacement that matches the composite background. Its strength is that it works fine with even severe spill conditions. On the negative side, when using the Solid Color Replacement mode, fine detail on the foreground edge tends to be lost. The single palette color sometimes cannot make a good color tone if the background image has some high-contrast color areas.
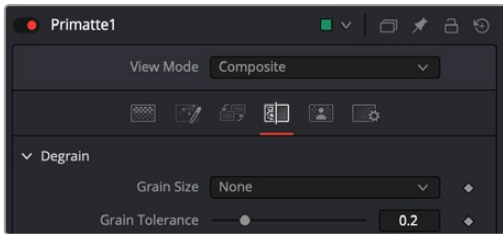
The Primatte Degrain tab

Degrain Tab

The Degrain tab is used when a foreground image is highly compromised by film grain. As a result of the grain, when backing screen noise is completely removed, the edges of the foreground object often become harsh and jagged, leading to a poor key.

Grain Size

The Grain Size selector provides a range of grain removal from Small to Large. If the foreground image has a large amount of film grain-induced pixel noise, you may lose a good edge to the foreground object when trying to clean all the grain noise with the Clean Background Noise Operation Mode. These tools clean up the grain noise without affecting the quality of the key.

— **None:** No degraining is performed.
— **Small:** The average color of a small region of the area around the sampled pixel. This should be used when the grain is very dense.
— **Medium:** The average color of a medium-sized region of the area around the sampled pixel. This should be used when the grain is less dense.
— **Large:** The average color of a larger region of the area around the sampled pixel. This should be used when the grain is very loose.

Grain Tolerance

Adjusting this slider increases the effect of the Clean Background Noise tool without changing the edge of the foreground object.
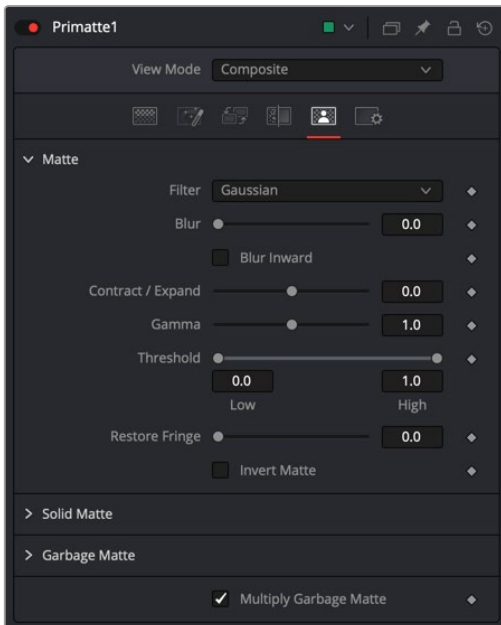
The Primatte Matte tab

Matte Tab

The Matte tab refines the alpha of the key, combined with any solid and garbage masks connected to the node. When using the Matte tab, set the viewer to display the alpha channel of Primatte's final output.

Filter

This control selects the filtering algorithm used when applying blur to the matte.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.

— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.

— **Multi-Box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."

— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Blur

Matte Blur blurs the edge of the matte based on the Filter menu setting. A value of zero results in a sharp, cutout-like hard edge. The higher the value, the more blur applied to the matte.

Blur Inward

Activating the Blur Inward checkbox generates the blur toward the center of the foreground subject. Conventional blurring or defocus affects the matte edges in both directions (inward and outward) and sometimes introduces a halo artifact around the edge in the composite view. Blur Inward functions only in the inward direction of the foreground subject (toward the center of the white area). The final result removes small and dark noise in the screen area without picking them up again in the Clean Background Noise mode. It can sometimes result in softer, cleaner edges on the foreground objects.

Contract/Expand

This slider shrinks or grows the semitransparent areas of the matte. Values above 0.0 expand the matte, while values below 0.0 contract it.

This control is usually used in conjunction with the Matte Blur to take the hard edge of a matte and reduce fringing. Since this control affects only semitransparent areas, it will have no effect on a matte's hard edge.

Gamma

Matte Gamma raises or lowers the values of the matte in the semitransparent areas. Higher values cause the gray areas to become more opaque, and lower values cause the gray areas to become more transparent. Completely black or white regions of the matte remain unaffected.

Since this control affects only semitransparent areas, it will have no effect on a matte's hard edge.

Threshold

This range slider sets the lower threshold using the handle on the left and sets the upper threshold using the handle on the right.

Any value below the lower threshold setting becomes black or transparent in the matte.

Any value above the upper threshold setting becomes white or opaque in the matte. All values within the range maintain their relative transparency values.

This control is often used to reject salt and pepper noise in the matte.

Restore Fringe

This restores the edge of the matte around the keyed subject. Often when keying, the edge of the subject where you have hair is clipped out. Restore Fringe brings back that edge while keeping the matte solid.

Invert Matte

When this checkbox is selected, the alpha channel created by the keyer is inverted, causing all transparent areas to be opaque and all opaque areas to be transparent.

Solid Matte

Solid mattes are mask nodes or images connected to the solid matte input on the node. The solid matte is applied directly to the alpha channel of the image. Generally, solid mattes are used to hold out keying in areas you want to remain opaque, such as someone with blue eyes against a blue screen.

Enabling Invert inverts the solid matte before it is combined with the source alpha.

Garbage Matte

Garbage mattes are mask nodes or images connected to the garbage matte input on the node. The garbage matte is applied directly to the alpha channel of the image. Generally, garbage mattes are used to remove unwanted elements that cannot be keyed, such as microphones and booms. They are also used to fill in areas that contain the color being keyed but that you wish to maintain.

Garbage mattes of different modes cannot be mixed within a single tool. A Matte Control node is often used after a Keyer node to add a garbage matte with the opposite effect of the matte applied to the keyer.

Enabling Invert inverts the garbage matte before it is combined with the source alpha.

Post-Multiply Image

Select this option to cause the keyer to multiply the color channels of the image against the alpha channel it creates for the image. This option is usually enabled and is on by default.

Deselect this checkbox and the image can no longer be considered premultiplied for purposes of merging it with other images. Use the Subtractive option of the Merge node instead of the Additive option.

For more information on these Merge node settings, *see Chapter 95, "Composite Nodes,"* in the DaVinci Resolve Reference Manual, or Chapter 33 in the Fusion Reference Manual.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Matte nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

How to Key with Primatte

You begin keying with Primatte by connecting the blue- or green-screen shot to the orange foreground input on the Primatte node and the background shot for the composite into the green background input. Once the connections are made, there are four main steps to using the Primatte:

1 Select Background Color.

2 Clean the Background Noise.

3 Clean the Foreground Noise.

4 Remove Spill.

Selecting Background Color

— In the Inspector, click the Select Background Color button.

— Position the mouse pointer over the blue/green-screen area in the viewer, somewhere near the foreground subject.

— Drag over the background color.

Primatte averages the pixels to get a single color. Sometimes Primatte works best when only a single pixel is sampled instead of a range of pixels.

Should you have difficulties with your keying, try the Select Background Color operation again with a single dark screen pixel or single light screen pixel.

Instead, If you want to make a rectangular selection, use the Box button in the top left-hand corner of the viewer. The Median button is the same as Line selection, except that each point sampled is the result of a 3 x 3 region based on where you click and then apply a median filter. This can potentially reduce any noisy pixels.
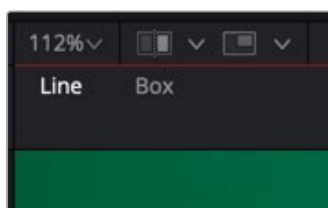
Primatte viewer buttons

If the foreground image has a shadow in it that you want to keep in the composite, do not select any of the dark screen pixels in the shadow. This keeps the shadow with the rest of the foreground image.

Clean Background Noise

If there are any white or light gray regions in the dark screen area, this is referred to as "noise." Technically, it is varying shades of the screen color that did not get picked up on the first sample and should be removed. You remove background noise using the Clean Background Noise button.

1   From the View Mode menu in the Inspector, select Black.

2   Above the viewer, click the Alpha Channel/RGB button.

The image displayed changes to a black and white "matte" view of the image.

3   Click the Clean Background Noise button.

4   Drag the mouse pointer through these white or light gray regions that should be pure black.

Primatte processes the selection and eliminates the noise.

5   Repeat this procedure as often as necessary to clear the noise from the background areas.

Selecting Gain/Gamma from the viewer's Options menu to increase the brightness or gamma allows you to see noise that would otherwise be invisible.
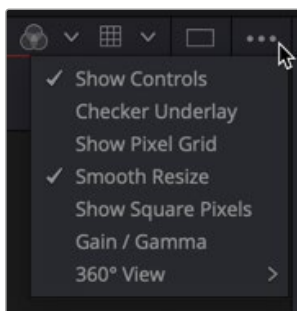
Primatte viewer Options menu

You do not need to remove every single white pixel to get good results. Most pixels displayed as a dark color close to black in a key image are considered transparent and virtually allow the background to be the final output in that area. Consequently, there is no need to eliminate all noise in the screen portions of the image. In particular, if an attempt is made to remove noise around the foreground subject meticulously, a smooth composite image is often difficult to generate.

> **TIP:** When clearing noise from around loose, flying hair or any background/ foreground transitional area, be careful not to select any of the areas near the edge of the hair. Leave a little noise around the hair as this can be cleaned up later using the Fine Tuning tools.

Clean Foreground Noise

If there are dark regions in the middle of the mostly white foreground subject, the key is not 100% in those areas. Using Clean Foreground Noise can make the matte as white as possible.

1   Keep the View Mode menu set to Black and the viewer set to the Alpha Channel.
2   Click the Clean Foreground Noise button.
3   Drag the mouse pointer through these dark pixels in the foreground that should be pure white. Primatte processes the selection and eliminates the noise.
4   Repeat this procedure as often as necessary to clear the noise from the foreground areas.
5   If enabled, disable Gain/Gamma from the viewer's Options menu to return to a regular viewer.

Removing Spill

The first three sections created a clean matte. At this point, the foreground can be composited onto any background image. However, if there is color spill on the foreground subject, a final operation is necessary to remove that screen spill for a more natural-looking composite.

1   From the View Mode menu, select Composite.
2   Above the viewer, click the Alpha/RGB toggle button to see the RGB image.
    There are two ways in Primatte to remove the spill color:

Spill Sponge

The quickest method is to select the Spill Sponge button and then sample the spill areas away. Additional spill removal can be done using the tools under the Fine Tuning tab or by using the Spill(-) button.

Fine Tuning Tab

To use the Fine Tuning tab for spill, first scrub over the spill color in the viewer. For most images, adjusting the Spill slider is all that is required to remove any remaining spill.

> **NOTE:** When using the slider in the Fine Tuning tab to remove spill, spill color replacement is replaced based on the setting of the Spill Replacement options.

You can use the other two sliders in the same way for different key adjustments. The Detail slider controls the matte softness for the color that is closest to the background color. For example, you can recover lost rarefied smoke in the foreground by selecting the Fine Tuning mode, clicking on the area of the image where the smoke starts to disappear and moving the Detail slider to the left. The Transparency slider controls the matte softness for the color that is closest to the foreground color. For example, if you have thick and opaque smoke in the foreground, you can make it semitransparent by moving the Transparency slider to the right after selecting the pixels in the Fine Tuning mode.

Ultra Keyer [UKy]

The Ultra Keyer node

Ultra Keyer Node Introduction

Like the newer Delta Keyer, the Ultra Keyer node has two keyers built in to it: a pre-matte keyer acts as a garbage matte creator and the color difference keyer that extracts fine detail and transparency. Generally, you start with the Delta Keyer as your first keyer of choice. If you do not get good results, try Primatte if you are using Fusion Studio. A good third choice is to try the Ultra Keyer.

Inputs

The Ultra Keyer node includes four inputs in the Node Editor.

— **Input:** The orange input accepts a 2D image that contains the color you want to be keyed for transparency.

— **Garbage Matte:** The gray garbage matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be made transparent. The garbage matte is applied directly to the alpha channel of the image.

— **Solid Matte:** The white solid matte input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input causes areas of the image that fall within the matte to be fully opaque.

— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps masks. Connecting a mask to this input limits the pixels where the keying occurs. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

A single keyer can rarely get perfect results because most green- or blue-screen shots have problems the keyer is not made to handle. Keyers often need the help of garbage mattes or solid mattes created with a Polygon or B-Spline node. Shots can also require more than just one keyer to achieve perfect results. Below, the Ultra Keyer node has the blue-screen content connected to the orange input. The result is an image with alpha that can then be connected into the foreground of a Merge node.
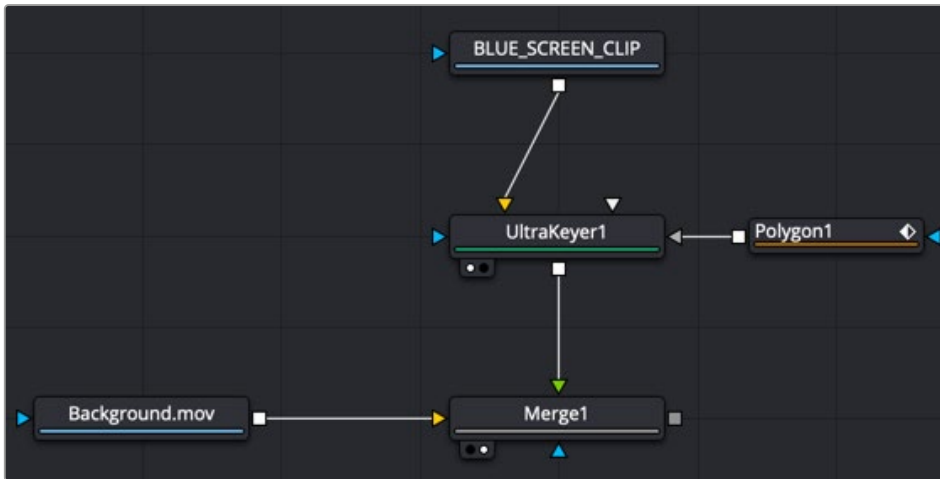
An Ultra Keyer node combined with polygon matte as a
garbage matte and connected to the foreground of a Merge

Inspector

The Ultra Keyer Pre-Matte tab

Pre-Matte Tab

The Pre-Matte tab is where most keying begins. It is used to select the screen color and smooth out
the color of the screen.

Background Color

The Background Color is used to select the color of the blue or green screen of the images. It is good practice to select the screen color close to the subject to be separated from the screen background.

Red Level, Green Level, Blue Level

These color sliders tune the level of the difference channels to help separate the color. When the background color is green, Red and Blue level options are provided. When the background color is blue, Red and Green level options are provided.

Background Correction

Depending on the background color selected above, the keyer iteratively merges the pre-keyed image over either a blue or green background before processing it further.

In some instances, this leads to better, more subtle edges.

Matte Separation

Matte Separation performs a pre-process on the image to help separate the foreground from the background before color selection. Generally, increase this control while viewing the alpha to eliminate the bulk of the background, but stop just before it starts cutting holes in the subject or eroding fine detail on the edges of the matte.

Pre-Matte Range

These R,G,B, and Luminance range controls update automatically to represent the current color selection. Colors are selected by selecting the Ultra Keyer node's tile in the node tree and dragging the Eyedropper into the viewer to select the colors to be used to create the matte. These range controls can be used to tweak the selection slightly, although selecting colors in the viewer is all that is required.

Lock Color Picking

This checkbox prevents accidentally selecting more colors from the view. It is a good idea to activate this checkbox once the color selection is made for the matte. All other controls in the node remain editable.

Pre Matte Size

The Pre Matte Size control can be used to soften the general area around the keyed image. This is used to close holes in the matte often caused by spill in semitransparent areas of the subject. This can cause a small halo around the subject, which can be removed using the Matte Contract tools found later in the tool.

Reset Pre Matte Ranges

This discards all color selection by resetting the ranges but maintains all other slider and control values.
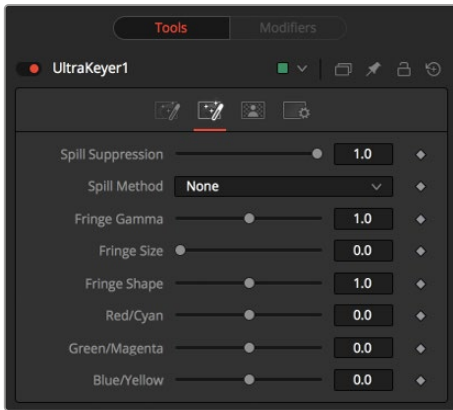
The Ultra Keyer Image tab

Image Tab

The Image tab handles the majority of spill suppression in the Ultra Keyer. Spill suppression is a form of color correction that attempts to remove the screen color from the fringe of the matte.

Spill is the transmission of the screen color through the semitransparent areas of the alpha channel. In the case of blue- or green-screen keying, this usually causes the color of the background to become apparent in the edges of the foreground subject.

Spill Suppression

When this slider is set to 0, no spill suppression is applied to the image. Increasing the slider increases the strength of the spill method.

Spill Method

This selects the strength of the algorithm used to apply spill suppression to the image.

— **None:** None is selected when no spill suppression is required.
— **Rare:** This removes very little of the spill color and is the lightest of all methods.
— **Medium:** This works best for green screens.
— **Well Done:** This works best for blue screens.
— **Burnt:** This works best for blue screens. Use this mode only for very troublesome shots.

Fringe Gamma

This control can be used to adjust the brightness of the fringe or halo that surrounds the keyed image.

Fringe Size

This expands and contracts the size of the fringe or halo surrounding the keyed image.

Fringe Shape

Fringe Shape presses the fringe toward the external edge of the image or pulls it toward the inner edge of the fringe. Its effect is most noticeable while the Fringe Size value is large.

Cyan/Red, Magenta/Green, and Yellow/Blue

Use these three controls to color correct the fringe of the image.

This is useful for correcting semitransparent pixels that still contain color from the original background to match the new background.

The Ultra Keyer Matte tab

Matte Tab

The Matte tab refines the alpha of the key, combined with any solid and garbage masks connected to the node. When using the Matte tab, set the viewer to display the alpha channel of the Delta Keyer's final output.

Filter

This control selects the filtering algorithm used when applying blur to the matte.

— **Box:** This is the fastest method but at reduced quality. Box is best suited for minimal amounts of blur.

— **Bartlett:** Otherwise known as a Pyramid filter, Bartlett makes a good compromise between speed and quality.

— **Multi-Box:** When selecting this filter, the Num Passes slider appears and lets you control the quality. At 1 and 2 passes, results are identical to Box and Bartlett, respectively. At 4 passes and above, results are usually as good as Gaussian, in less time and with no edge "ringing."

— **Gaussian:** The Gaussian filter uses a true Gaussian approximation and gives excellent results, but it is a little slower than the other filters. In some cases, it can produce an extremely slight edge "ringing" on floating-point pixels.

Blur

Matte Blur blurs the edge of the matte based on the Filter menu setting. A value of zero results in a sharp, cutout-like hard edge. The higher the value, the more blur applied to the matte.

Clipping Mode

This option determines how edges are handled when performing domain of definition rendering. This is profoundly important when blurring the matte, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining areas in the frame are treated as black/transparent.

— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that would normally be outside the upstream DoD is treated as black/transparent.

Contract/Expand

This slider shrinks or grows the semitransparent areas of the matte. Values above 0.0 expand the matte, while values below 0.0 contract it.

This control is usually used in conjunction with the Matte Blur to take the hard edge of a matte and reduce fringing. Since this control affects only semitransparent areas, it has no effect on a matte's hard edge.

Gamma

Matte Gamma raises or lowers the values of the matte in the semitransparent areas. Higher values cause the gray areas to become more opaque, and lower values cause the gray areas to become more transparent. Completely black or white regions of the matte remain unaffected.

Since this control affects only semitransparent areas, it will have no effect on a matte's hard edge.

Threshold

This range slider sets the lower threshold using the handle on the left and sets the upper threshold using the handle on the right.

Any value below the lower threshold setting becomes black or transparent in the matte.

Any value above the upper threshold setting becomes white or opaque in the matte. All values within the range maintain their relative transparency values.

This control is often used to reject salt and pepper noise in the matte.

Restore Fringe

This restores the edge of the matte around the keyed subject. Often when keying, the edge of the subject where you have hair is clipped out. Restore Fringe brings back that edge while keeping the matte solid.

Invert Matte

When this checkbox is selected, the alpha channel created by the keyer is inverted, causing all transparent areas to be opaque and all opaque areas to be transparent.

Solid Matte

Solid mattes are mask nodes or images connected to the solid matte input on the node. The solid matte is applied directly to the alpha channel of the image. Generally, solid mattes are used to hold out keying in areas you want to remain opaque, such as someone with blue eyes against a blue screen.

Enabling Invert inverts the solid matte before it is combined with the source alpha.

Garbage Matte

Garbage mattes are mask nodes or images connected to the garbage matte input on the node. The garbage matte is applied directly to the alpha channel of the image. Generally, garbage mattes are used to remove unwanted elements that cannot be keyed, such as microphones and booms. They are also used to fill in areas that contain the color being keyed but that you wish to maintain.

Garbage mattes of different modes cannot be mixed within a single tool. A Matte Control node is often used after a Keyer node to add a garbage matte with the opposite effect of the matte applied to the keyer.

Enabling Invert inverts the garbage matte before it is combined with the source alpha.

Post-Multiply Image

Select this option to cause the keyer to multiply the color channels of the image against the alpha channel it creates for the image. This option is usually enabled and is on by default.

Deselect this checkbox and the image can no longer be considered premultiplied for purposes of merging it with other images. Use the Subtractive option of the Merge node instead of the Additive option.

For more information on these Merge node settings, *see Chapter 95, "Composite Nodes,"* in the DaVinci Resolve Reference Manual, or Chapter 33 in the Fusion Reference Manual.

Subtract Background

This option color corrects the edges when the screen color is removed and anti-aliased to a black background. By enabling this option, the edges potentially become darker. Disabling this option allows you to pass on the color of the screen to use in other processes down the line.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other matte nodes. These common controls are described in detail in the following "The Common Controls" section.

The Common Controls

Nodes that handle matte operations share a number of identical controls in the Inspector. This section describes controls that are common among matte nodes.

Inspector
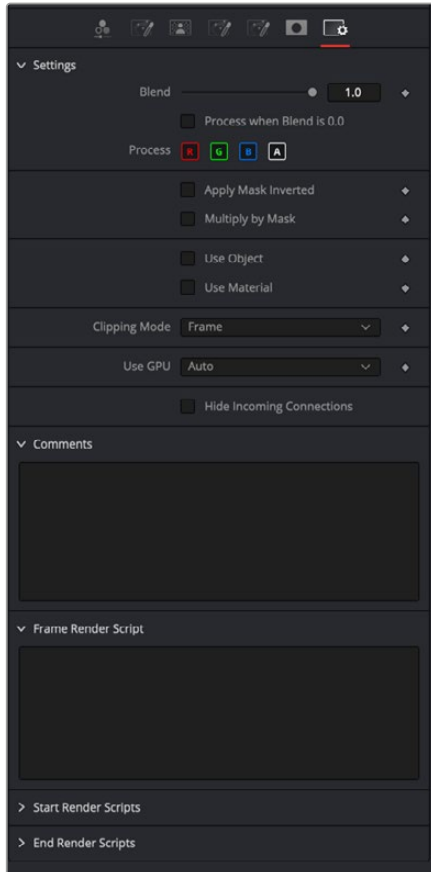
Common Matte settings Inspector

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Matte category. The controls are consistent and work the same way for each tool.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Usually, this causes the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if this node is scripted to trigger another task, but the value is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the Red button on a Blur tool is deselected, the blur is first applied to the image, and then the red channel from the original input is copied back over the red channel of the result.

There are some exceptions, such as tools where deselecting these channels causes the tool to skip processing that channel entirely. In that case, there are a set of RGBA buttons on the Controls tab in the tool. The buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option causes the RGB values of the masked image to be multiplied by the mask channel's values. This causes all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object ID and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels are used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID is used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the view. The image or sequence must have been rendered from a 3D software package with those channels included.

Clipping Mode

This option determines how edges are handled when performing domain of definition rendering. This is profoundly important when blurring the matte, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame is treated as black/transparent.

— **Domain:** Setting this option to Domain respects the upstream domain of definition when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that would normally be outside the upstream DoD is treated as black/transparent.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off GPU hardware-accelerated rendering. Enabled uses the GPU hardware for rendering the node. Auto uses a capable GPU if one is available, and falls back to software rendering when a capable GPU is not available

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node are displayed in the Inspector. Dragging a connected node from the node tree into the field hides that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line reappears.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Chapter 108

Metadata Nodes

This chapter details the Metadata nodes available in Fusion.

> The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.
>
> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Copy Metadata [Meta]

The Copy Metadata node

Copy Metadata Node Introduction

Copy Metadata combines, replaces, or clears the metadata in your image. Metadata can be viewed in a subview of the viewer.

Inputs

The two inputs on the Copy Metadata node are used to connect two 2D images.

— **Background Input:** The orange background input is used for the primary 2D image that is output from the node.

— **Foreground Input:** The green foreground input is used for the secondary 2D image that contains metadata you want merge or overwrite onto the background image.

Basic Node Setup

The Copy Metadata node takes metadata from the foreground input (green) and copies it into the background input (orange). The output is the background input with modified metadata.

A Copy Metadata node copies metadata from the foreground and embeds it into the background clip

Inspector
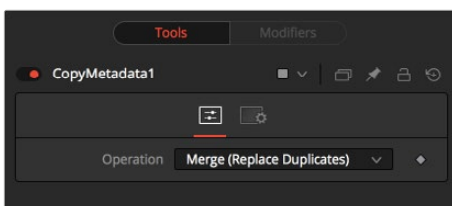
The Copy Metadata Controls tab

Controls Tab

The Controls tab configures how metadata coming from the foreground input image gets added to the background input image.

Operation

The Operation menu determines how the metadata of the foreground and background inputs are treated.

— **Merge (Replace Duplicates):** All values are merged, but values with duplicate names are taken from the foreground input.

— **Merge (Preserve Duplicates):** All values are merged, but values with duplicate names are taken from the background input.

— **Replace:** The metadata in the foreground replaces the entire metadata in the background.

— **Clear:** All metadata is discarded.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Metadata nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Set Metadata [SMeta]

The Set Metadata node

Set Metadata Node Introduction

Set Metadata allows you to create new Name = Value pairs in the metadata. Metadata can be viewed in a subview of the viewer.

Inputs

The single input on the Set Metadata node is used to connect a 2D image that gets metadata added.

— **Background Input:** The orange background input is used for the primary 2D image that is output from the node with the new metadata.

Basic Node Setup

The Set Metadata node embeds new metadata into the background input (orange). The output is the background input with new metadata.

A Set Metadata node creates new metadata and embeds it into the background clip.

Inspector
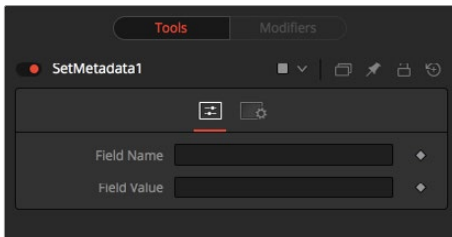
The Set Metadata Controls tab

Controls Tab

The Controls tab is where you set up the name of the metadata field and the value or information regarding the metadata.

Field Name

The name of the metadata value. Do not use spaces.

Field Value

The value assigned to the name above.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Metadata nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Set Timecode [TcMeta]
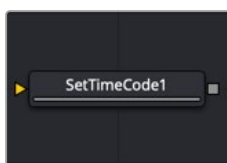
The Set Timecode node

Set Timecode Node Introduction

Set Timecode inserts dynamic timecode values into the metadata table based on the FPS settings.

Inputs

The single input on the Set Timecode node is used to connect a 2D image that gets timecode added.

— **Background Input:** The orange background input is used for the primary 2D image that is output from the node with the new timecode.

Basic Node Setup

The Set Timecode node embeds new timecode metadata into the background input (orange). The output is the background input with updated timecode.

A Set Timecode node inserts new timecode metadata into the background clip.

Inspector
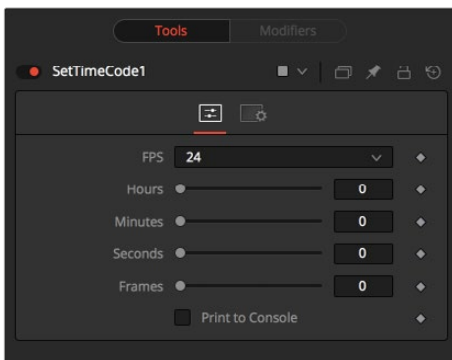
The Set Timecode Controls tab

Controls Tab

The Controls tab sets the clip's starting timecode metadata based on FPS, hours, minutes, seconds, and frames.

FPS

You can choose from a variety of settings for frames per second.

Since this is a Fuse, you can easily adapt the settings to your needs by editing the appropriate piece of code for the buttons:

```
MBTNC_StretchToFit = true,
{ MBTNC_AddButton = "24" },
{ MBTNC_AddButton = "25" },
{ MBTNC_AddButton = "30" },
{ MBTNC_AddButton = "48" },
{ MBTNC_AddButton = "50" },
```

```
    { MBTNC_AddButton = "60" },
    })
```

as well as for the actual values:

```
local rates = { 24, 25, 30, 48, 50, 60 }
```

Hours/Minutes/Seconds/Frames Sliders

Define an offset from the starting frame of the current comp.

Print to Console

Verbose output of the Timecode/Frame value in the Console.

The Timecode/Frames conversion is done according to the FPS settings. The result might look like this:

```
TimeCode: 00:00:08:15
Frames:   207
```

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Metadata nodes. These common controls are described in detail in the following "The Common Controls" section.

The Common Controls

Nodes that handle metadata operations share several identical controls in the Inspector. This section describes controls that are common among Metadata nodes.

Inspector
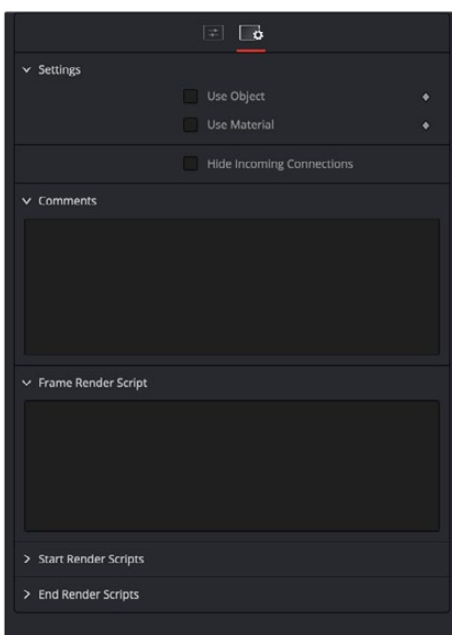
The Common Metadata Settings tab

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Metadata category. The controls are consistent and work the same way for each tool.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object ID and material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels are used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID is used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the view. The image or sequence must have been rendered from a 3D software package with those channels included.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Miscellaneous Nodes

This chapter details miscellaneous nodes within Fusion.

> The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.
>
> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Auto Domain [ADoD]

The Auto Domain node

Auto Domain Node Introduction

The Auto Domain node automatically sets the image's domain of definition (DoD) based on bounds of the input image's background Canvas color. It does not change the image's physical dimensions. Some EXR images come with optimized DoDs already set, but other formats do not. For formats other than EXR, this node can speed up compositions by optimizing the DoD based on the content rather than the frame's dimensions.

For example, a CG character rarely takes up the entire frame of an image. With this type of image, the Auto Domain node sets the DoD to a rectangular region by comparing image pixels with the Canvas color. The Canvas color indicates what color the pixels are outside the DoD. By default, unless a Canvas color is set using the Set Canvas Color node, the color is set to black. This default works well when an image has a premultiplied alpha channel. The result is a DoD that encompasses the portion of the clip that contains only the character. The DoD is updated on each frame to accommodate changes, such as a character walking closer to the camera. However, if a clip does not contain an alpha channel, the Set Canvas Color node can be used to define the Canvas color as solid alpha with a color that matches the solid background.

For more detail on the Set Canvas Color node, *see Chapter 94, "Color Nodes,"* in the DaVinci Resolve Reference Manual, or Chapter 32 in the Fusion Reference Manual..

> **NOTE:** The Domain of Definition is a bounding box that encompasses pixels that have a nonzero value. The DoD is used to limit image-processing calculations and speeds up rendering.

Inputs

The single input on the Auto Domain node is used to connect a 2D image and an effect mask, which can be used to limit the blurred area.

— **Input:** The orange input is used for the primary 2D image that is blurred.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the blur to only those pixels within the mask.

Basic Node Setup

The example below assumes that the image does not contain an alpha channel but a CG character rendered against a black background. This is common for different render passes like specular or shadows, for example. The image is connected to a Set Canvas Color node, which sets the Canvas color to black with a solid alpha. The Set Canvas Color then connects to the Auto Domain node, which detects the pixels and sets the DoD. If the original image contained a premultiplied alpha channel, the Set Canvas Color would not be needed, and the image could be connected directly into the Auto Domain node.
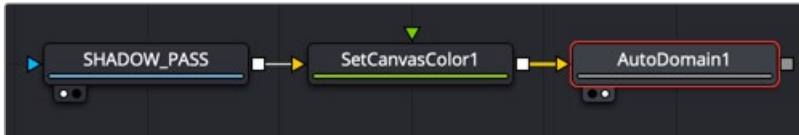
An Auto Domain node automatically limits the area of image processing.
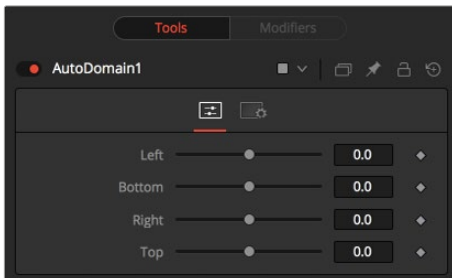
Inspector

The Auto Domain Controls tab

Controls Tab

In most cases, the Auto Domain node automatically calculates the DoD bounding box; however, the rectangular shape can be modified using the Controls tab in the Inspector.

Left

Defines the left border of the search area of the ADoD. Higher values on this slider move the left border toward the right, excluding more data from the left margin.

1 represents the right border of the image; 0 represents the left border. The slider defaults to 0 (left border).

Bottom

Defines the bottom border of the search area of the ADoD. Higher values on this slider move the bottom border toward the top, excluding more data from the bottom margin.

1 represents the top border of the image; 0 represents the bottom border. The slider defaults to 0 (bottom border).

Right

Defines the right border of the search area of the ADoD. Higher values on this slider move the right border toward the left, excluding more data from the right margin.

1 represents the right border of the image; 0 represents the left border. The slider defaults to 1 (right border).

Top

Defines the top border of the search area of the ADoD. Higher values on this slider move the top border toward the bottom, excluding more data from the top margin.

1 represents the top border of the image; 0 represents the bottom border. The slider defaults to 1 (top border).

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other miscellaneous nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Change Depth [CD]
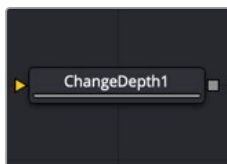
The Change Depth node

Change Depth Node Introduction

The Change Depth node has one simple use, and that is to change the bits per color channel used to process a node. This node is often used after color correcting 32-bit floating-point image files, converting them from float processing to 16-bit per channel to preserve memory and performance.

It can also be useful if, from a certain point in your node tree, you feel the need to process your images in a higher bit depth than their original one or to reduce the bit depth to save memory.

Inputs

The single input on the Change Depth node is used to connect a 2D image and an effect mask, which can be used to limit the blurred area.

— **Input:** The orange input is used for the primary 2D image to be converted.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes or bitmaps from other tools. Connecting a mask to this input limits the blur to only those pixels within the mask.

Basic Node Setup

Below, a Loader node in Fusion Studio is color corrected and then down converted from a floating-point image to a 16-bit image to save image-processing time and memory.

A Change Depth node placed after color correction is done on a floating-point image.

Inspector
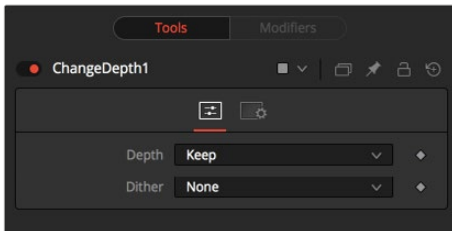
The Change Depth Controls tab

Controls Tab

The two controls for this node are the Depth menu and the Dither menu. These two menus are used to convert and adjust the color depth of the image.

Depth

The Keep setting doesn't do anything to the image but instead keeps the input depth. The other options change the bit depth of the image to the respective value.

Dither

When down converting from a higher bit depth, it can be useful to add Error Diffusion or Additive Noise to camouflage artifacts that result from problematic (high-contrast) areas.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other miscellaneous nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Custom Tool [CT]

The Custom Tool node

Custom Tool Node Introduction

The Custom Tool node is the most complex and the most powerful node in Fusion. It is used to create custom expressions and filters to modify an image. In addition to providing three image inputs, the Custom Tool node allows for the connection of up to eight numeric inputs and as many as four XY position values from other controls and parameters in the node tree.

Per-pixel calculations can be performed on the Red, Green, Blue, Alpha, Z, Z-Coverage, UV texture coords, XYZ Normals, RGBA background color, and XY motion vector channels of the images.

You should be moderately experienced with scripting, or C++ programming, to understand the structure and terminology used by the Custom Tool node.

Inputs

The Custom Tool node has three image inputs, a matte input, and an effect mask input.

— **Input:** The orange, green, and magenta inputs combine 2D images to make your composite. When entering them into the Custom Tool fields, they are referred to as c1, c2 and c3 (c standard for all three R, G, B channels)

— **Matte Input:** The white input is for a matte created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a matte to this input allows a matte to be combined into any equation. When entering the matte into the Custom Tool fields, it is referred to as m1.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Custom Tool effect to only those pixels within the mask.

Basic Node Setup

The Custom Tool below takes two image inputs and a matte input, and then combines them using some calculation. The result can be output to a Merge or other image-processing nodes.
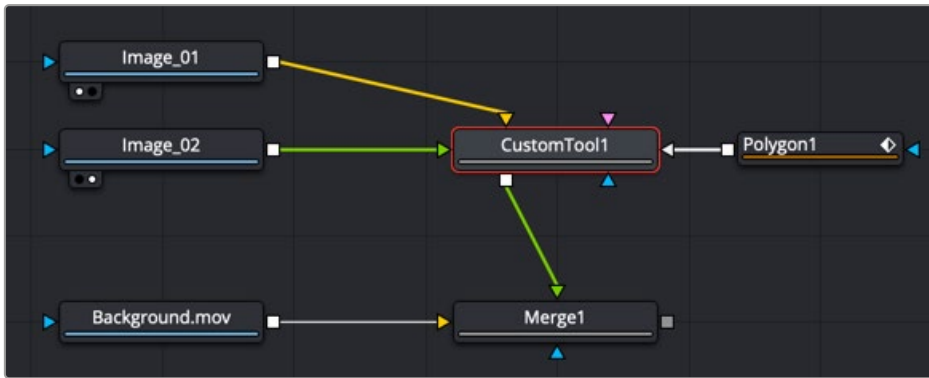
A Custom Tool is used to build your own effects using C++ and scripting.

Inspector
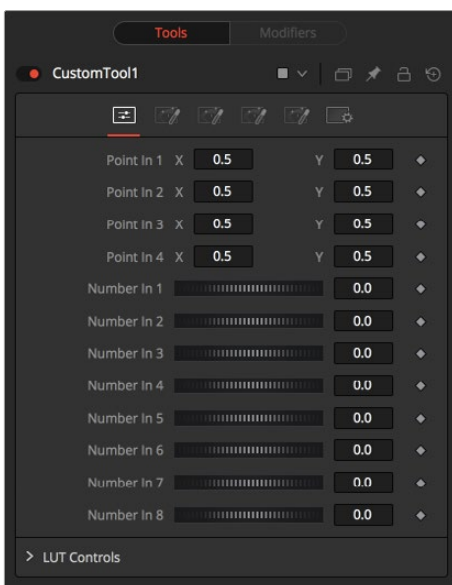
The Custom Tool Controls tab

Controls Tab

Point in 1-4, X and Y

These four controls are 2D X and Y center controls that are available to expressions entered in the Setup, Intermediate, and Channels tabs as variables p1x, p1y, …., p4x, p4y. They are normal positional controls and can be animated or connected to modifiers as any other node might.

Number in 1-8

The values of these controls are available to expressions entered in the Setup, Intermediate, and Channels tabs as variables n1, n2, n3, …, n8. They are normal slider controls and can be animated or connected to modifiers exactly as any other node might.

LUT in 1-4

The Custom Tool node provides 4 LUT splines. The values of these controls are available to expressions entered in the Setup, Intermediate, and Channels tabs using the getlut# function. For example, setting the R, G, B, and A expressions to getlut1(r1), getlut2(g1), getlut3(b1), and getlut4(a1), respectively, would cause the Custom Tool node to mimic the Color Curves node.

These controls can be renamed using the options in the Config tab to make their meanings more apparent, but expressions still see the values as n1, n2, ..., n8.

Custom Tool Setup Tab
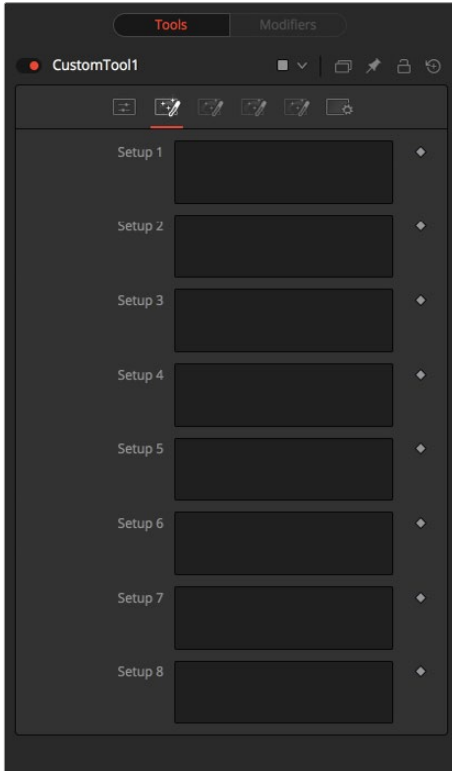
The Custom Tool Setup tab

Setup 1-4

Up to four separate expressions can be calculated in the Setup tab of the Custom Tool node. The Setup expressions are evaluated once per frame before any other calculations are performed. The results are then made available to the other expressions in the Custom Tool node as variables s1, s2, s3, and s4.

> **NOTE:** Because these expressions are evaluated once per frame only and not for each pixel, it makes no sense to use per-pixel variables like X and Y or channel variables like r1, g1, b1. Allowable values include constants, variables such as n1..n8, time, W and H, and functions like sin() or getr1d().

Custom Tool Intermediate Tab
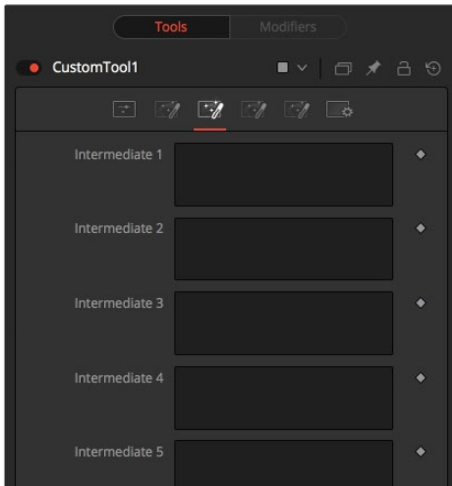
The Custom Tool Intermediate tab

Intermediate 1-4

An additional four expressions can be calculated in the Inter tab. The Inter expressions are evaluated once per pixel after the Setup expressions are evaluated but before the Channel expressions are evaluated. Per-pixel channel variables like r1, g1, b1, and a1 are allowable. Results are available as variables i1, i2, i3, and i4.

Custom Tool Config Tab
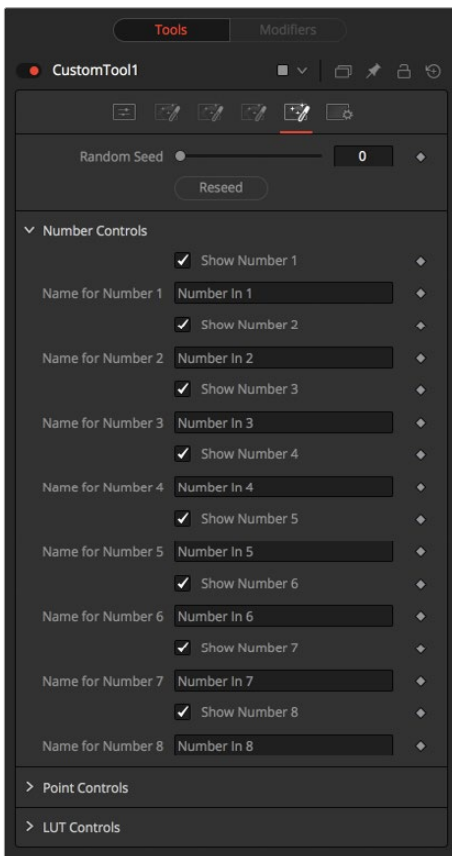
The Custom Tool Config tab

Random Seed

Use this to set the seed for the rand() and rands() functions. Click the Randomize button to set the seed to a random value. This control may be needed if multiple Custom Tool nodes are required with different random results for each.

Number Controls

There are eight sets of Number controls, corresponding to the eight Number In sliders in the Controls tab. Uncheck the Show Number checkbox to hide the corresponding Number In slider, or edit the Name for Number text field to change its name.

Point Controls

There are four sets of Point controls, corresponding to the four Point In controls in the Controls tab. Uncheck the Show Point checkbox to hide the corresponding Point In control and its crosshair in the viewer. Similarly, edit the Name for Point text field to change the control's name.

Custom Tool Channels Tab
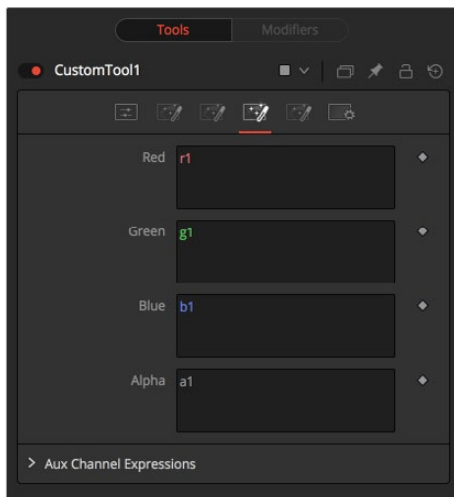
The Custom Tool Channel tab

RGBA, Z, UV Expressions, and XYZ Normal Expressions

The Channel tab is used to set up one expression per each available channel of the image. Each expression is evaluated once per pixel. The result creates the value for that pixel in the output of the image.

Color Channel expressions (RGBA) should generally return floating-point values between 0.0 and 1.0. Values beyond this are clipped if the destination image is an integer. Other expression fields should produce values appropriate to their channel (e.g., between -1.0 and 1.0 for Vector and Normal fields, 0.0 to 1.0 for Coverage, or any value for Depth). The Channel expressions may use the results from both the Setup expressions (as variables s1–s4) and Inter expressions (as variables i1–i4).

Custom Tool Node Syntax

Value Variables	
n1..n8	Numeric Inputs
p1x..p4x	Position Values (X-axis)
p1y..p4y	Position Values (Y-axis)
s1..s4	Setup Expression Results
i1..i4	Inter Expression Results
time	Current Frame
x	Horizontal co-ordinate of the current pixel, between 0.0 and 1.0
y	Vertical co-ordinate of the current pixel, between 0.0 and 1.0
w (or w1..w3)	Width of Image (for image1..image3)
h (or h1..h3)	Height of Image (for image1..image3)
ax (or ax1..ax3)	Image Aspect X (for image1..image3)
ay (or ay1..ay3)	Image Aspect Y (for image1..image3)

> **NOTE:** Use w and h and ax and ay without a following number to get the dimensions and aspect of the primary image.

Channel (Pixel) Variables	
c1..c3	Current Channel (for image1..image3)
r1..r3	Red (for image1..image3)
g1..g3	Green (for image1..image3)
b1..b3	Blue (for image1..image3)
a1..a3	Alpha (for image1..image3)
z1..z3	Z-Buffer (for image1..image3)
cv1..cv3	Z Coverage (for image1..image3)
u1..u3	U Coordinate (for image1..image3)
v1..v3 nx1..nx3	V Coordinate (for image1..image3) X Normal (for image1..image3)
ny1..ny3	Y Normal (for image1..image3)
nz1..nz3	Z Normal (for image1..image3)
bgr1..bgr3	Background Red (for image1..image3)
bgg1..bgg3	Background Green (for image1..image3)
bgb1..bgb3	Background Blue (for image1..image3)
bga1..bga3	Background Alpha (for image1..image3)

Channel (Pixel) Variables	
`vx1..vx3`	X Vector (for image1..image3)
`vy1..vy3`	Y Vector (for image1..image3)
`nz1..nz3`	Z Normal (for image1..image3)

> **NOTE:** Use c1, c2, c3 to refer to the value of a pixel in the current channel. This makes copying and pasting expressions easier. For example, if c1/2 is typed as the red expression, the result would be half the value of the red pixel from image 1, but if the expression is copied to the blue channel, now it would have the value of the pixel from the blue channel.

To refer to the red value of the current pixel in input 1, type r1. For the image in input 2, it would be r2.

— `get[ch][#]b(x, y)`   Read pixel at x,y, or 0 if out of bounds—e.g., getr1b(0,0)

— `get[ch][#]d(x, y)` Read pixel at x,y or edge pixel if out of bounds—e.g., getr1d(0,0)

— `get[ch][#]w(x, y)` Read pixel at x,y or wrap if out of bounds—e.g., getr1w(0,0)

> **NOTE:** There are a variety of methods used to refer to pixels from locations other than the current one in an image.

In the above description, [ch] is a letter representing the channel to access. The [#] is a number representing the input image. So to get the red component of the current pixel (equivalent to r), you would use getr1b(x,y). To get the alpha component of the pixel at the center of image 2, you would use geta2b(0.5, 0.5).

— `getr1b(x,y)` Output the red value of the pixel at position x, y, if there were a valid pixel present. It would output 0.0 if the position were beyond the boundaries of the image (all channels).

— `getr1d(x,y)`   Output the red value of the pixel at position x, y. If the position specified were outside of the boundaries of the image, the result would be from the outer edge of the image (RGBA only).

— `getr1w(x,y)` Output the red value of the pixel at position x, y. If the position specified were outside of the boundaries of the image, the x and y coordinates would wrap around to the other side of the image and continue from there (RGBA only).

To access other channel values with these functions, substitute the r in the above examples with the correct channel variable (r, g, b, a and, for the getr1b() functions only, z, and so on), as shown above. Substitute the 1 with either 2 or 3 in the above examples to access the images from the other image inputs.

Mathematical Expressions	
`pi`	The value of pi
`e`	The value of e
`log(x)`	The base-10 log of x
`ln(x)`	The natural (base-e) log of x
`sin(x)`	The sine of x (x is degrees)
`cos(x)`	The cosine of x (x is degrees)
`tan(x)`	The tangent of x (x is degrees)
`asin(x)`	The arcsine of x, in degrees
`acos(x)`	The arccosine of x, in degrees
`atan(x)`	The arctangent of x, in degrees
`atan2(x,y)`	The arctangent of x,y, in degrees
`abs(x)`	The absolute (positive) value of x
`int(x)`	The integer (whole) value of x
`frac(x)`	The fractional value of x
`sqrt(x)`	The Square Root of x
`rand(x,y)`	A random value between x and y
`rands(x,y,s)`	A random value between x and y, based on seed s
`min(x,y)`	The minimum (lowest) of x and y
`max(x,y)`	The maximum (highest) of x and y
`dist(x1,y1,x2,y2)`	The distance between point x1,y2 and x2,y2
`dist3d(x1,y1,z1,x2,y2,z2)`	The distance between 3D points x1,y2,z1 and x2,y2,z2
`noise(x)`	A smoothly varying Perlin noise value based on x
`noise2(x, y)`	A smoothly varying Perlin noise value based on x and y
`noise3(x, y, z)`	A smoothly varying Perlin noise value based on x, y and z
`if(c, x, y)`	returns x if c not 0, otherwise y

Mathematical Operators	
`!x`	1.0 if x = 0, otherwise 0.0
`–x`	(0.0 - x)
`+x`	(0.0 + x) i.e. effectively does nothing
`x ^ y`	x raised to the power of y
`x * y`	x multiplied by y
`x / y`	x divided by y
`x % y`	x modulo y, i.e. remainder of (x divided by y)
`x + y`	x plus y

Mathematical Operators

`x - y`	x minus y
`x < y`	1.0 if x is less than y, otherwise 0.0
`x > y`	1.0 if x is greater than y, otherwise 0.0
`x <= y`	1.0 if x is less than or equal to y, otherwise 0.0
`x >= y`	1.0 if x is greater than or equal to y, otherwise 0.0
`x = y`	1.0 if x is exactly equal to y, otherwise 0.0
`x == y`	1.0 if x is exactly equal to y, otherwise 0.0, identical to above
`x <> y`	1.0 if x is not equal to y, otherwise 0.0
`x != y`	1.0 if x is not equal to y, otherwise 0.0, i.e. identical to above
`x & y`	1.0 if both x and y are not 0.0, otherwise 0.0
`x && y`	1.0 if both x and y are not 0.0, otherwise 0.0, i.e. identical to above
`x|y`	1.0 if either x or y (or both) are not 0.0, otherwise 0.0
`x||y`	1.0 if either x or y (or both) are not 0.0, otherwise 0.0

EXAMPLE The following examples are intended to help you understand the various components of the Custom Tool node.

ROTATION

To rotate an image, we need the standard equations for 2D rotation:

```
x' = x * cos(theta) - y * sin(theta) y' = x * sin(theta) + y * cos(theta)
```

Using the n1 slider for the angle theta, and a sample function, we get (for the red channel):

```
getr1b(x * cos(n1) - y * sin(n1), x * sin(n1) + y * cos(n1))
```

This calculates the current pixel's (x,y) position rotated around the origin at (0,0) (the bottom-left corner), and then fetches the red component from the source pixel at this rotated position. For centered rotation, we need to subtract 0.5 from our x and y coordinates before we rotate them, and add 0.5 back to them afterward:

```
getr1b((x-.5) * cos(n1) - (y-.5) * sin(n1) + .5, (x-.5) * sin(n1) + (y-.5) * cos(n1) + .5)
```

Which brings us to the next lesson: Setup and Intermediate Expressions. These are useful for speeding things up by minimizing the work that gets done in the Channel expressions. The Setup expressions are executed only once, and their results don't change for any pixel, so you can use these for s1 and s2, respectively.

```
cos(n1) sin(n1)
```

Intermediate expressions are executed once for each pixel, so you can use these for i1 and i2:

```
(x-.5) * s1 - (y-.5) * s2 + .5

(x-.5) * s2 + (y-.5) * s1 + .5
```

These are the x and y parameters for the getr1b() function from above, but with the Setup results, s1 and s2, substituted so that the trig functions are executed only once per frame, not every pixel. Now you can use these intermediate results in your Channel expressions:

```
getr1b(i1, i2)

getg1b(i1, i2)

getb1b(i1, i2)

geta1b(i1, i2)
```

With the Intermediate expressions substituted in, we only have to do all the additions, subtractions, and multiplications once per pixel, instead of four times. As a rule of thumb, if it doesn't change, do it only once.

This is a simple rotation that doesn't take into account the image aspect at all. It is left as an exercise for you to include this (sorry). Another improvement could be to allow rotation around points other than the center.

FILTERING

Our second example duplicates the functionality of a 3 x 3 Custom Filter node set to average the current pixel together with the eight pixels surrounding it. To duplicate it with a Custom Tool node, add a Custom Tool node to the node tree, and enter the following expressions into the Setup tab.

(Leave the node disconnected to prevent it from updating until we are ready.)

```
S1

1.0/w1

S2

1.0/h1
```

These two expressions are evaluated at the beginning of each frame. S1 divides 1.0 by the current width of the frame, and S2 divides 1.0 by the height. This provides a floating-point value between 0.0 and 1.0 that represents the distance from the current pixel to the next pixel along each axis.

Now enter the following expression into the first text control of the Channel tab (r).

```
(getr1w(x-s1, y-s2) + getr1w(x, y-s2) + getr1w(x+s1, y-s2) + getr1w(x+s1,
y) + getr1w(x-s1, y) + r1 +getr1w(x-s1, y+s2) + getr1w(x, y+s2) +
getr1w(x+s1, y+s2)) / 9
```

This expression adds together the nine pixels above the current pixel by calling the getr1w() function nine times and providing it with values relative to the current position. Note that we referred to the pixels by using x+s1, y+s2, rather than using x+1, y+1.

Fusion refers to pixels as floating-point values between 0.0 and 1.0, which is why we created the expressions we used in the Setup tab. If we had used x+1, y+1 instead, the expression would have sampled the same pixel over and over again. (The function we used wraps the pixel position around the image if the offset values are out of range.)

That took care of the red channel; now use the following expressions for the green, blue, and alpha channels.

```
(getg1w(x-s1, y-s2) + getg1w(x, y-s2) + getg1w(x+s1, y-s2) + getg1w(x+s1,
y) + getg1w(x-s1, y) + g1 +getg1w(x-s1, y+s2) + getg1w(x, y+s2) +
getg1w(x+s1, y+s2)) / 9
```

```
(getb1w(x-s1, y-s2) + getb1w(x, y-s2) + getb1w(x+s1, y-s2) + getb1w(x+s1,
y) + getb1w(x-s1, y) + b1 +getb1w(x-s1, y+s2) + getb1w(x, y+s2) +
getb1w(x+s1, y+s2)) / 9
```

```
(geta1w(x-s1, y-s2) + geta1w(x, y-s2) + geta1w(x+s1, y-s2) + geta1w(x+s1,
y) + geta1w(x-s1, y) + a1 + geta1w(x-s1, y+s2) + geta1w(x, y+s2) +
geta1w(x+s1, y+s2)) / 9
```

It's time to view the results. Add a Background node set to a solid color and change the color to a pure red. Add a hard-edged Rectangular effects mask and connect it to the expression just created.

For comparison, add a Custom Filter node and duplicate the settings from the image above. Connect a pipe to this node from the background to the node and view the results. Alternate between viewing the Custom Tool node and the Custom Filter while zoomed in close to the top corners of the effects mask.

Of course, the Custom Filter node renders a lot faster than the Custom Tool node we created, but the flexibility of the Custom Tool node is its primary advantage. For example, you could use an image connected to input 2 to control the median applied to input one by changing all instances of getr1w, getg1w, and getb1w in the expression to getr2w, getg2w, and getb2w, but leaving the r1, g1, and b1s as they are.

This is just one example; the possibilities of the Custom Tool node are limitless.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other miscellaneous nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Fields [Flds]

The Fields node

Fields Node Introduction

The Fields node is a robust multipurpose utility offering several functions related to interlaced video frames. It interpolates video fields into frames or video frames into fields. Although the interlace preference and method type is defined in the MediaIn or Loader, and generators, this node can be used to assist in the standards conversion of PAL to NTSC and provides the ability to process fields and frames for specific portions of a node tree.

This node can also interlace two separate images together into a single interlace image.
The background input is the dominate field 1 and the foreground is field 2.

Inputs

The single input on the Fields node is used to connect a 2D image and an effect mask, which can be used to limit the blurred area.

— **Stream1 Input:** The orange background input is used for the primary 2D image that is interpolated or converted.
— **Stream2 Input:** The optional green foreground input is only used when merging two interlaced images together.

Basic Node Setup

The Fields node below is used to convert the background image from a PAL interlaced format to progressive frames.
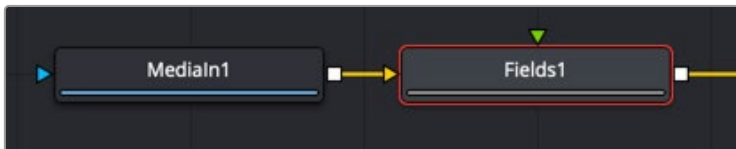
The background image can be modified to various interlaced formats.

Inspector
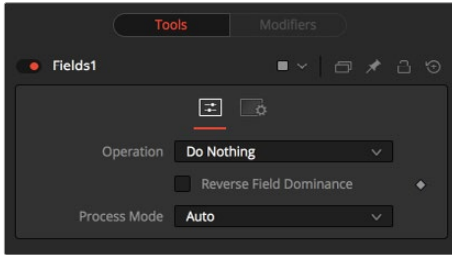
The Fields Controls tab

Controls Tab

The Controls tab includes two menus. The Operation menu is used to select the type of field conversion performed. The Process Mode menu is used to select the field's format for the output image.
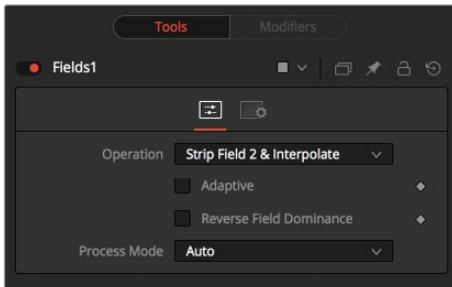
Operatiion Mode

Operation Menu

— **Do Nothing:** This causes the images to be affected by the Process Mode selection exclusively.

— **Strip Field 2:** This removes field 2 from the input image stream, which shortens the image to half of the original height.

— **Strip Field 1:** This removes field 1 from the input image stream, which shortens the image to half of the original height.

— **Strip Field 2 and Interpolate:** This removes field 2 from the input image stream and inserts a field interpolated from field 1 so that image height is maintained. Should be supplied with frames, not fields.

— **Strip Field 1 and Interpolate:** This removes field 1 from the input image stream and inserts a field interpolated from field 2 so that image height is maintained. Should be supplied with frames, not fields.

— **Interlace:** This combines fields from the input image stream(s). If supplied with one image stream, each pair of frames are combined to form half of the number of double-height frames. If supplied with two image streams, single frames from each stream are combined to form double-height images.

— **De-Interlace:** This separates fields from one input image stream. This will produce double the amount of half-height frames.

Reverse Field Dominance

When selected, the Field Order or Dominance of the image will be swapped.
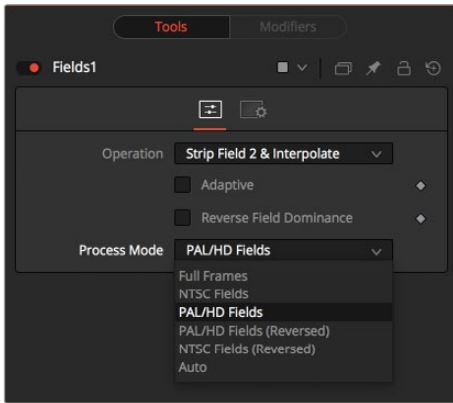
Process Mode

Process Mode Menu

— **Full Frames:** This forces Frame Processing. Useful for processing frames in a part of a node tree that is otherwise field processing.

— **NTSC Fields:** This forces NTSC Field Processing. Useful for processing fields in a part of a node tree that is otherwise frame processing.

— **PAL Fields:** This forces PAL Field Processing. Useful for processing fields in a part of a node tree that is otherwise frame processing.

— **PAL Fields (Reversed):** This forces PAL-swapped Field Processing.

— **NTSC Fields (Reversed):** This forces NTSC-swapped Field Processing.

— **Auto:** This attempts to match the mode of its input images. Fields are used if the input types are mixed.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other miscellaneous nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Frame Average [Avg]
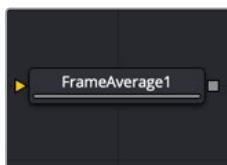
The Frame Average node

Frame Average Node Introduction

The Frame Average node averages together a series of frames to simulate clips shot with long shutter speeds. Aside from motion blur-style effects, it can be useful for time warps or noise removal.

Inputs

The single input on the Frame Average node is used to connect a 2D image that will have the averaging applied.

— **Input:** The orange input is used for the primary 2D image that will be averaged.

Basic Node Setup

The image connected to the orange input is frame averaged based on the settings in the Inspector.
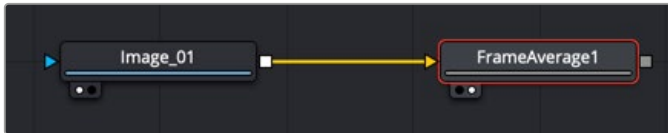
A Frame Average node blends the input image's frames.

Inspector
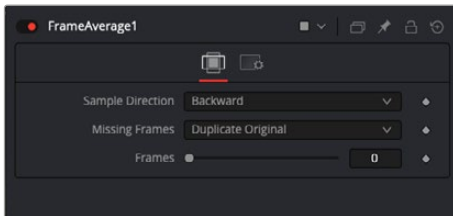
The Frame Average Controls tab

Controls Tab

The Controls tab contains the parameters for setting the duration and guidance of the averaged frames.

Sample Direction

The Sample Direction menu determines if the averaged frames are taken before the current frame, after, or a mix of the two.

— **Forward:** Averages the number of frames set by the Frames slider after the current frame.
— **Both:** Averages the number of frames set by the Frames slider, taking frames before and after the current frame.
— **Backward:** Averages the number of frames set by the Frames slider before the current frame.

Missing Frames

This control determines the behavior if a frame is missing from the clip.

— **Duplicate Original:** Uses the last original frame until a new frame is available.
— **Blank Frame:** Leaves missing frames blank.

Frames

This slider sets the number of frames that are averaged.

Keyframe Stretcher [KfS]

The Keyframe Stretcher node

Keyframe Stretcher Node Introduction

The Keyframe Stretcher node is inserted after animated nodes, so the keyframes stretch and the comp's duration is modified. It is used to scale the keyframes on the animation curve to the current duration of the clip. This is particularly useful when creating title templates in Fusion for use in DaVinci Resolve's Edit or Cut page.

> **TIP:** The Keyframe Stretcher can be used on a single parameter by applying the Keystretcher modifier.

Inputs

The single input on the Keyframe Stretcher node is used to connect a 2D image that contains keyframe animation.

— **Input:** The orange input is used for any node with keyframed animation. The input can be a Merge node that is not animated but contains foreground and background nodes that are animated.

Basic Node Setup

The Keyframe Stretcher is added just before the Media Out or Saver node. All nodes that include animation before the Keyframe Stretcher are modified if the comp changes duration.
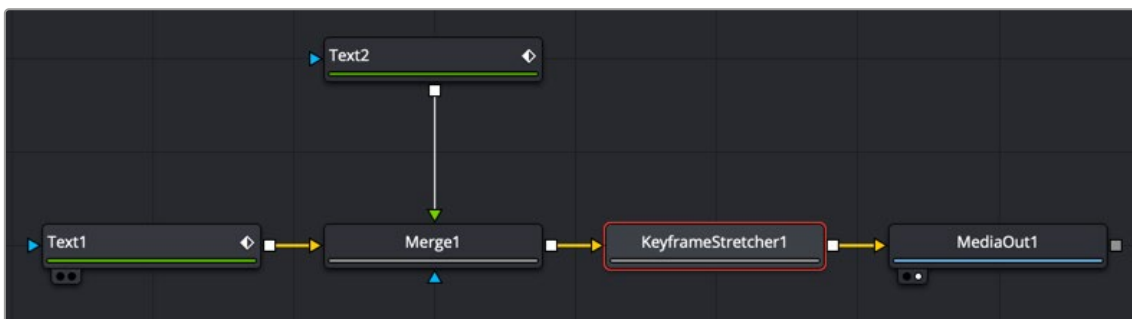
A Keyframe Stretcher changing the animation of two text nodes.

The diagram below shows the original 50-frame animation added to a parameter. The Keyframe Stretcher Start and End would be set to 0 and 50. The second keyframe is set at frame 10, and the third keyframe is set at frame 40. Setting the Stretch Start to frame 11 and the Stretch End to frame 39 will keep the existing keyframes at the same speed (number of frames.) The middle will be stretched.
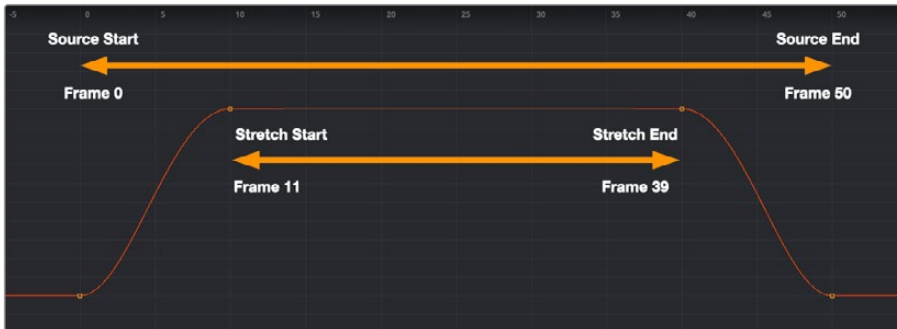
Original 50-frame animation

In the below example, the duration of the clip is extended to 75 frames. The first 10 frames and the last 10 frames of the animation run at the same speed as the original animation, while any animation in the middle is stretched to fill the difference.

> **NOTE:** The actual Spline Editor will show only the original keyframe positions. The splines are not changed by the Keyframe Stretcher; only the animation is changed.
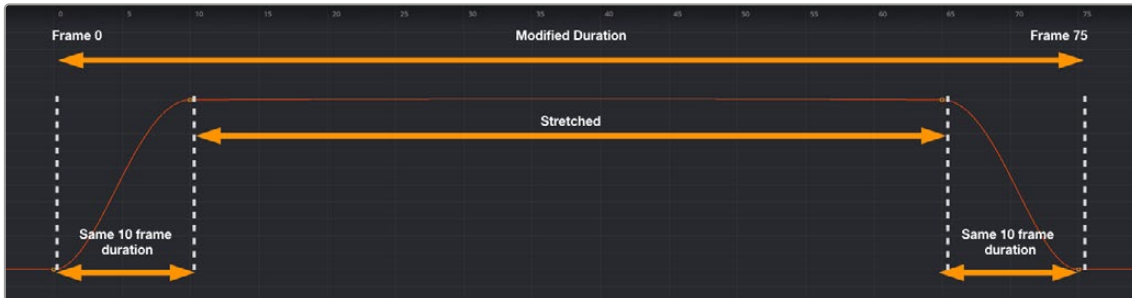
Animation modified to 75 frames but stretching only the middle of the animation

Inspector
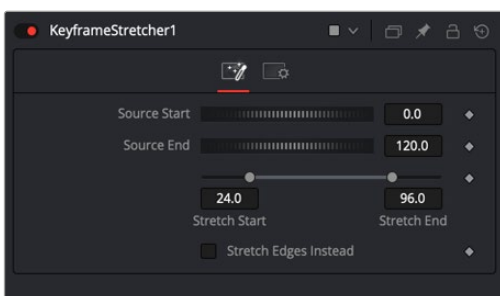
The Keyframe Stretcher Keyframes tab

Keyframes Tab

The Keyframes tab includes Source controls for setting the source duration and Stretch controls for setting the area of the animation that gets modified.

Source Start/Source End

A source range is specified using the Source Start and Source End controls. These are typically set to match the full range of the animation spline on the Keyframes control.

Stretch Start/Stretch End

The Stretch Start and Stretch End controls let you specify a middle zone where keyframes will be stretched or squished. Handles outside the range will not get scaled. Any keyframes outside the Stretch Start and End range always remain the same number of frames from the Start and End.

Any keyframe adjustments to the original control will be correspondingly scaled back to the source curve and will match the original timing as expected.

Stretch Edges Instead

Enabling the Stretch Edges Instead checkbox overrides the Stretch Start and Stretch End controls and stretches the edges of the animation.

Run Command [Run]
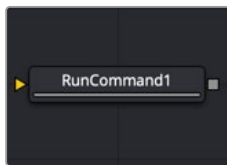
The Run Command node

Run Command Node Introduction

The Run Command node is used to execute an external command or batch file at specific points during a render. You can choose to run a command at the start or the end of a render. Or you can have the command execute once for each frame.

The Run Command can be used to net render other command line applications using the Fusion Render Manager, as well as a host of other useful functions.

Inputs

The single input on the Run Command node is used to pass through a 2D image.

— **Input:** The optional orange image input is not required for this node to operate. However, if it is connected to a node's output, the Run Command will only launch after the connected node has finished rendering. This is often useful when connected to a Saver, to ensure that the output frame has been fully saved to disk first. If the application launched returns a non-zero result, the node will also fail.

Basic Node Setup

The Run Command node can be connected after a Saver and run once the final frame is completed.
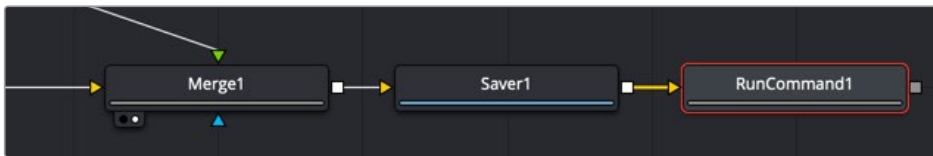
Run Command placed after a Saver node.

Inspector
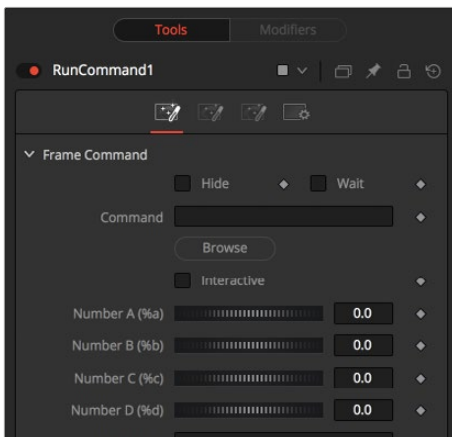
The Run Command Frame tab

Frame Tab

The Frame tab is where the command to execute is selected and modified.

Hide

Enable the Hide checkbox to prevent the application or script from displaying a window when it is executed.

Wait

Enable this checkbox to cause the node to wait for a remote application or tool to exit before continuing. If this checkbox is disabled, the Fusion continues rendering without waiting for the external application.

Frame Command

This field is used to specify the path for the command to be run after each frame is rendered. The Browse button can be used to identify the path.

Interactive

This checkbox determines whether the launched application should run interactively, allowing user input.

Number A (%a) and Number B (%b)

Various wildcards can be used with the frame commands; these wildcards will be substituted at render time with the correct values.

- **%a:** Outputs the number from the Number A thumbwheel control.
- **%b:** Outputs the number from the Number B thumbwheel control.
- **%t:** Outputs the current frame number (without zero padding).
- **%s:** Substitutes using the text from the large text entry field.

If you want to add zero paddings to the numbers generated by %t, refer to the wildcard with %0x, where x is the number of characters with which to pad the value. This also works for %a and %b.

For example, test%04t.tga would return the following values at render time:

```
test0000.tga
test0001.tga
test0009.tga
test0010.tga
```

You may also pad a value with spaces by calling the wildcard as %x, where x is the number of spaces with which you would like to pad the value.
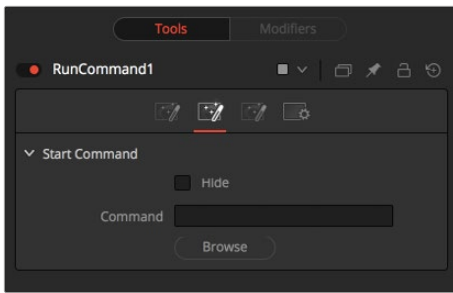
The Run Command Start tab   The Run Command End tab
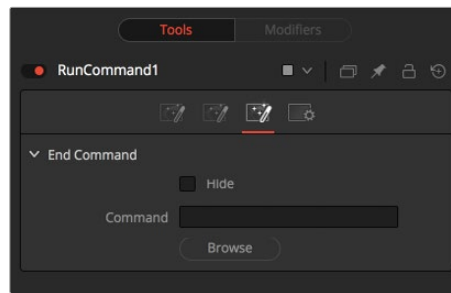
Start and End Tabs

The Start and End tabs contain a file browser for a command to be run when the composition starts to render and when the composition is done rendering.

> **EXAMPLE** To copy the saved files from a render to another directory as each frame is rendered, save the following text in a file called copyfile.bat to your C\ directory (the root folder).
>
> ```
> @echo off
> set parm=%1 %2
> copy %1 %2 set parm=
> ```
>
> Create or load any node tree that contains a Saver. The following example assumes a Saver is set to output D\ test0000.exr, test0001.exr, etc. You may have to modify the example to match.
>
> Add a Run Command node after the Saver to ensure the Saver has finished saving first. Now enter the following text into the Run Command node's Frame Command text box:
>
> ```
> C\copytest.bat D\test%04f.exr C\
> ```

Select the Hide Frame command checkbox to prevent the Command Prompt window from appearing briefly after every frame.

When this node tree is rendered, each file will be immediately copied to the C\ directory as it is rendered.

The Run Command node could be used to transfer the files via FTP to a remote drive on the network, to print out each frame as it is rendered, or to execute a custom image-processing tool.

The Run Command node is not restricted to executing simple batch files. FusionScript, VBScript, Jscript, CGI, and Perl files could also be used, as just a few examples.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other miscellaneous nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Set Domain [DoD]

The Set Domain node

Set Domain Node Introduction

Set Domain is used to adjust or set the active area of an image or the area of the image considered to have valid data.

It does not change the image's physical dimensions. Downstream nodes will not process anything outside the Domain of Definition (DoD), thus speeding up rendering of computation-intensive nodes.

This node provides an absolute mode, for setting the domain of definition manually, and a relative mode for adjusting the existing domain of definition.

Inputs

The two inputs on the Set Domain node are used to connect 2D images.

— **Input:** The orange background input must be connected. It accepts a 2D image with the DoD you want to replace or adjust.

— **Foreground:** The green image input is optional but also accepts a 2D image as its input. When the foreground input is connected, the Set Domain node will replace the Background input's domain of definition with the foreground's DoD.

Basic Node Setup

The example below assumes an image is connected to a Set Domain node to manually configure the DoD.
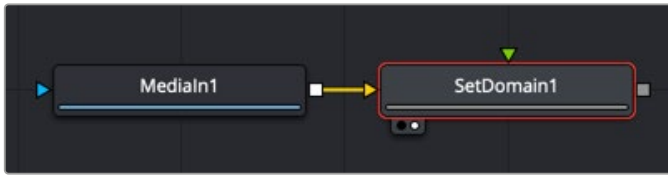
A Set Domain node manually sets the area to limit image processing.

Inspector
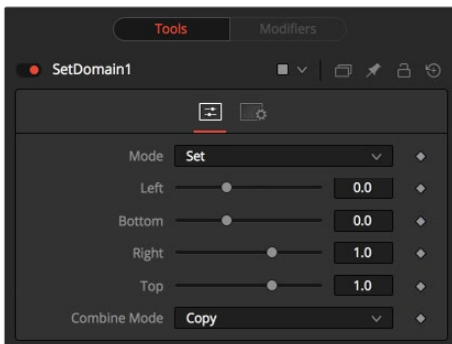
The Set Domain Controls tab in Set mode

Controls Tab

Mode

The Mode menu has two choices depending on whether you want to adjust or offset the existing domain or set precise values for it.

The same operations can be performed in Set or in Adjust mode. In Adjust mode, the sliders default to 0, marking their respective full extent of the image. Positive values shrink the DoD while negative values expand the DoD to include more data.

Set mode defaults to the full extent of the visible image. Sliders default to a scale of 0-1 from left to right and bottom to top.

Left

Defines the left border of the DoD. Higher values on this slider move the left border toward the right, excluding more data from the left margin.

1 represents the right border of the image; 0 represents the left border. The slider defaults to 0 (left border).

Bottom

Defines the bottom border of the DoD. Higher values on this slider move the bottom border toward the top, excluding more data from the bottom margin.

1 represents the top border of the image; 0 represents the bottom border. The slider defaults to 0 (bottom border).

Right

Defines the right border of the DoD. Higher values on this slider move the right border toward the left, excluding more data from the right margin.

1 represents the right border of the image; 0 represents the left border. In Set mode, the slider defaults to 1 (right border).

Top

Defines the top border of the DoD. Higher values on this slider move the top border toward the bottom, excluding more data from the top margin.

1 represents the top border of the image; 0 represents the bottom border. In Set mode, the slider defaults to 1 (top border).
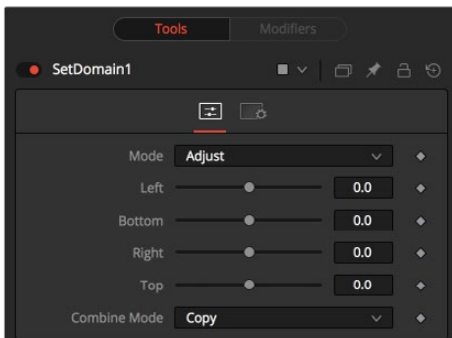
The Set Domain Controls tab in Adjust mode

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other miscellaneous nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Time Speed [TSpd]
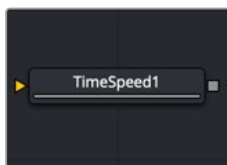
The Time Speed node

Time Speed Node Introduction

The Time Speed node allows image sequences to be sped up, slowed down, reversed, or delayed. Image Interpolation offers smooth, high-quality results. Time Speed should be used for static speed changes or to introduce delays in the footage. To apply animated changes in time, such as accelerating or decelerating time, use a Time Stretcher instead.

When operating in Flow mode, Optical Flow data is required. This node does not generate optical flow directly. You have to create it upstream using an Optical Flow node or by loading the forward/reverse vector channels from the image.

TimeSpeed does not interpolate the aux channels but instead destroys them. In particular, the Vector/ BackVector channels are consumed and destroyed after computation.

Add an Optical Flow after the Time Speed node if you want to generate flow vectors for the retimed footage.

Inputs

The single input on the Time Speed node is used to connect a 2D image that will be retimed.

— **Input:** The orange input is used for the primary 2D image that will be retimed.

Basic Node Setup

The Time Speed node setup is as simple as connecting a 2D image into the orange background input of the node.
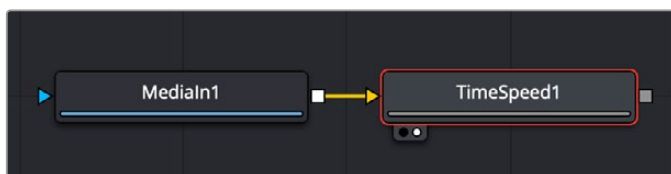
A MediaIn node having its speed changed in the Time Speed node.

Inspector
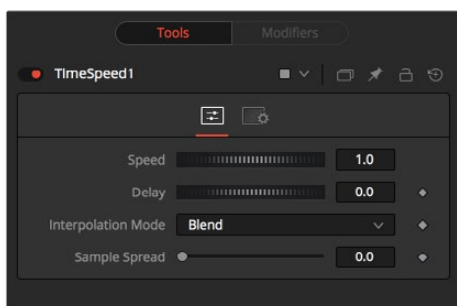
The Time Speed Controls tab

Speed

This control is used to adjust the Speed, in percentage values, of the outgoing image sequence. Negative values reverse the image sequence. 200% Speed is represented by a value of 2.0, 100% by 1.0, 50% by 0.5, and 10% by 0.1.

The Speed control cannot be animated.

Delay

Use this control to Delay the outgoing image sequence by the specified number of frames. Negative numbers offset time back, and positive numbers advance time.

Interpolate Mode

This menu determines the how the time speed is processed in order to improve its visual playback quality, especially in the case of clips that are slowed down. There are three choices in the menu.

— **Nearest:** The most processor efficient and least sophisticated method of processing; frames are either dropped for fast motion or duplicated for slow motion.

— **Blend:** Also processor efficient, but can produce smoother results; adjacent duplicated frames are dissolved together to smooth out slow or fast motion effects.

— **Flow:** The most processor intensive but highest quality method of speed effect processing. Using vector channels pre-generated from an Optical Flow node, new frames are generated to create slow or fast motion effects. The result can be exceptionally smooth when motion in a clip is linear. However, two moving elements crossing in different directions or unpredictable camera movement can cause unwanted artifacts.

Sample Spread

This slider is displayed only when Interpolation is set to Blend. The slider controls the strength of the interpolated frames on the current frame. A value of 0.5 blends 50% of the frame before and 50% of the frame ahead and 0% of the current frame.

Depth Ordering

This menu is displayed only when Interpolation is set to Flow. The Depth Ordering is used to determine which parts of the image should be rendered on top. This is best explained by example.

In a locked-off camera shot where a car is moving through the frame, the background does not move, so it produces small, or slow, vectors. The car produces larger, or faster, vectors.

The Depth Ordering, in this case, is Fastest on Top, since the car draws over the background.

In a shot where the camera pans to follow the car, the background has faster vectors, and the car has slower vectors, so the Depth ordering method would be Slowest on Top.

Clamp Edges

This checkbox is displayed only when Interpolation is set to Flow. Under certain circumstances, this option can remove the transparent gaps that may appear on the edges of interpolated frames. Clamp Edges can cause a stretching artifact near the edges of the frame that is especially visible with objects moving through it or when the camera is moving.

Because of these artifacts, it is a good idea to use clamp edges only to correct small gaps around the edges of an interpolated frame.

Edge Softness

This slider is only displayed when Interpolation is set to Flow and Clamp Edges is enabled. It helps to reduce the stretchy artifacts that might be introduced by Clamp Edges.

If you have more than one of the Source Frame and Warp Direction checkboxes turned on, this can lead to doubling up of the stretching effect near the edges. In this case, you'll want to keep the softness rather small at around 0.01. If you have only one checkbox enabled, you can use a larger softness at around 0.03.

Source Frame and Warp Direction

These checkboxes are displayed only when Interpolation is set to Flow. These controls determine which frames and which vectors are used to create the in-between frames. Each method ticked on will be blended into the result.

— **Prev Forward:** Takes the previous frame and uses the Forward vector to interpolate the new frame.

— **Next Forward:** Takes the next frame in the sequence and uses the Forward vector to interpolate the new frame.

— **Prev Backward:** Takes the previous frame and uses the Back Forward vector to interpolate the new frame.

— **Next Backward:** Takes the next frame in the sequence and uses the Back vector to interpolate the new frame.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other miscellaneous nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Time Stretcher [TSt]
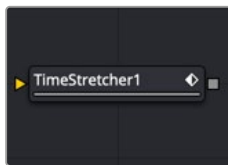
The Time Stretcher node

Time Stretcher Node Introduction

The Time Stretcher node is similar to the Time Speed node but permits the speed of the clip to be animated. Full spline control of the effect is provided, including smoothing. As a result, the Time Stretcher can be used to animate a single clip to 200, back to normal speed, pause for a second, and then play backward (like a VCR rewinding).

Image interpolation offers smooth, high-quality results, all using a spline curve to adjust time nonlinearly. To apply constant time changes such as frame rate changes, use a Time Speed instead.

When operating in Flow mode, Optical Flow data is required. This node does not generate optical flow directly; you must create it manually upstream using an Optical Flow node or by loading the forward/reverse vector channels from disk.

Flow Stretcher does not interpolate the aux channels but instead destroys them. In particular, the Vector/BackVector channels are consumed/destroyed. Add an Optical Flow after the Flow Stretcher if you want to generate flow vectors for the retimed footage.
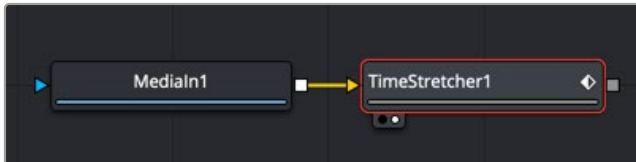
Inputs

The single input on the Time Stretcher node is used to connect a 2D image that will be time stretched.

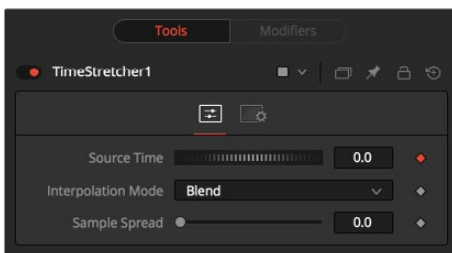— **Input:** The orange input is used for the primary 2D image that will be time stretched.

Basic Node Setup

Like the Time Speed node, the Time Stretcher setup is as simple as connecting a 2D image into the orange background input of the node.

A MediaIn node having its time ramped to various speeds in the Time Stretcher node.

Inspector

The Time Stretcher controls

Source Time

This control designates from which frame in the original sequence to begin sampling.

When a Time Stretcher node is added to the node tree, the Source Time control already contains a Bézier spline with a single keyframe set to 0.0. The keyframe position is determined by the current time when the node is added to the node tree.

> **NOTE:** The Source Time spline may not be immediately visible until Edit is selected from the Source Time's contextual menu, or Display all Splines is selected from the Spline Window's contextual menu.

Interpolate Mode

This menu determines the how the time speed is processed in order to improve its visual playback quality, especially in the case of clips that are slowed down.

There are three choices in the menu.

— **Nearest:** The most processor efficient and least sophisticated method of processing; frames are either dropped for fast motion or duplicated for slow motion.

— **Blend:** Also processor efficient but can produce smoother results; adjacent duplicated frames are dissolved together to smooth out slow or fast motion effects.

— **Flow:** The most processor intensive but highest quality method of speed effect processing. Using vector channels pre-generated from an Optical Flow node, new frames are generated to create slow or fast motion effects. The result can be exceptionally smooth when motion in a clip is linear. However, two moving elements crossing in different directions or unpredictable camera movement can cause unwanted artifacts.

Sample Spread

This slider is displayed only when Interpolation is set to Blend. The slider controls the strength of the interpolated frames on the current frame. A value of 0.5 blends 50% of the frame before and 50% of the frame ahead and 0% of the current frame.

Depth Ordering

This menu is displayed only when Interpolation is set to Flow. The Depth Ordering is used to determine which parts of the image should be rendered on top. This is best explained by example.

In a locked-off camera shot where a car is moving through the frame, the background does not move, so it produces small, or slow, vectors. The car produces larger, or faster, vectors.

The Depth Ordering in this case is Fastest on Top, since the car draws over the background.

In a shot where the camera pans to follow the car, the background has faster vectors, and the car has slower vectors, so the Depth ordering method would be Slowest on Top.

Clamp Edges

This checkbox is displayed only when Interpolation is set to Flow. Under certain circumstances, this option can remove the transparent gaps that may appear on the edges of interpolated frames. Clamp Edges can cause a stretching artifact near the edges of the frame that is especially visible with objects moving through it or when the camera is moving.

Because of these artifacts, it is a good idea to use clamp edges only to correct small gaps around the edges of an interpolated frame.

Edge Softness

This slider is displayed only when Interpolation is set to Flow and Clamp Edges is enabled. It helps to reduce the stretchy artifacts that might be introduced by Clamp Edges.

If you have more than one of the Source Frame and Warp Direction checkboxes turned on, this can lead to doubling up of the stretching effect near the edges. In this case, you'll want to keep the softness rather small at around 0.01. If you have only one checkbox enabled, you can use a larger softness at around 0.03.

Source Frame and Warp Direction

These checkboxes are displayed only when Interpolation is set to Flow. These controls determine which frames and which vectors are used to create the in-between frames. Each method ticked on will be blended into the result.

— **Prev Forward:** Takes the previous frame and uses the Forward vector to interpolate the new frame.

— **Next Forward:** Takes the next frame in the sequence and uses the Forward vector to interpolate the new frame.

— **Prev Backward:** Takes the previous frame and uses the Back Forward vector to interpolate the new frame.

— **Next Backward:** Takes the next frame in the sequence and uses the Back vector to interpolate the new frame.

> **EXAMPLE** Make sure that the current time is either the first or last frame of the clip to be affected in the project. Add the Time Stretcher node to the node tree. This will create a single point on the Source Time spline at the current frame. The value of the Source Time will be set to zero for the entire Global Range.
>
> Set the value of the Source Time to the frame number to be displayed from the original source, at the frame in time it will be displayed in during the project.
>
> To shrink a 100-frame sequence to 25 frames, follow these steps:
>
> 1. Change the Current Time to frame 0.
>
> 2. Change the Source Time control to 0.0.
>
> 3. Advance to frame 24.
>
> 4. Change the Source Time to 99.
>
> 5. Check that the spline result is linear.
>
> 6. Fusion will render 25 frames by interpolating down the 100 frames to a length of 25.
>
> 7. Hold the last frame for 30 frames, and then play the clip backward at regular speed. Continue the example from above and follow the steps below.
>
> 8. Advance to frame 129.
>
> 9. Right-click on the Source Time control and select Set Key from the menu.
>
> 10. Advance to frame 229 (129 + 100).
>
> 11. Set the Source time to 0.0.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other miscellaneous nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Wireless Link [Wire]

The Wireless Link node

Wireless Link Node Introduction

The Wireless Link node helps manage the tangle of connection lines in a node tree by wirelessly connecting one 2D node to another 2D node.

Although Wireless Links can be helpful, try to keep as much of a node tree as visible as possible; otherwise, you lose one of the main benefits of a node tree.
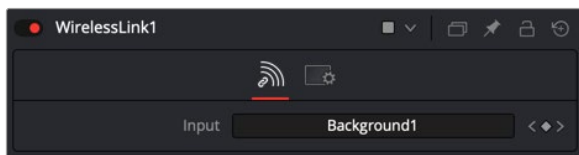
Inputs

There are no inputs on this node.

Basic Node Setup

There is no setup for this node. It is a free-standing node that connects "wirelessly" using the control in the Inspector.

Inspector

The Wireless Link Controls tab

Controls Tab

The Controls tab in the Wireless Link node contains a single Input field for the linked node.

Input

To use the Wireless Link node, in the Node Editor, drag the 2D node into the Input field of the Wireless Link node. Any change you make to the original node is wirelessly replicated in the Wireless Link node. You can use the output from the Wireless Link node to connect to a nearby node.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other miscellaneous nodes. These common controls are described in detail in the following "The Common Controls" section.

The Common Controls

Nodes that handle miscellaneous operations share a number of identical controls in the Inspector. This section describes controls that are common among miscellaneous nodes.

Inspector
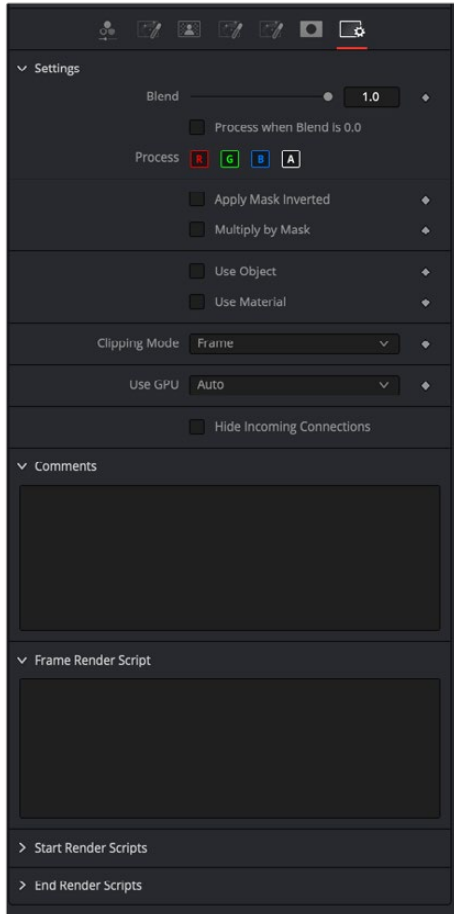
The Common Miscellaneous Settings tab

Settings Tab

The Settings tab in the Inspector can be found on every tool in the miscellaneous nodes. The controls are consistent and work the same way for each tool.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this will cause the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if processing of this node is scripted to trigger another task, but the value of the node is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the Red button on a Blur tool is deselected, the blur will first be applied to the image, and then the red channel from the original input will be copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this will generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option will cause the RGB values of the masked image to be multiplied by the mask channel's values. This will cause all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object ID and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels will be used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option is disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID will be used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the view. The image or sequence must have been rendered from a 3D software package with those channels included.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off GPU hardware-accelerated rendering. Enabled uses the GPU hardware for rendering the node. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available.

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 will cause Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one whole frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows for the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node will be displayed in the Inspector. Dragging a connected node from the node tree into the field will hide that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line will reappear.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Optical Flow

This chapter details the Optical Flow nodes available in Fusion.

The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Optical Flow [OF]
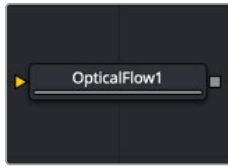
The OpticalFlow node

Optical Flow Node Introduction

This node analyzes a clip connected to its input using an Optical Flow algorithm. Think of optical flow as a per-pixel motion vector that matches up features over several frames.

The computed optical flow is stored within the Vector and Back Vector aux channels of the output. These channels can be used in other nodes like the Vector Motion Blur or Vector Distort. However, Optical Flow must render twice when connecting it to a Time Stretcher or Time Speed node. These nodes require the channels A. FwdVec and B. BackVec in that order, but Optical Flow generates A. BackVec and A. FwdVec when it processes.

If you find that optical flow is too slow, consider rendering it out into OpenEXR files using a Saver node.

> **TIP:** If the footage input flickers on a frame-by-frame basis, it is a good idea to deflicker the footage beforehand.

Inputs

The Optical Flow node includes a single orange image input.

— **Input:** The orange background input accepts a 2D image. This is the sequence of frames for which you want to compute optical flow. The output of the Optical Flow node includes the image and vector channels. The vector channels can be displayed by right-clicking in the viewer and choosing Channel > Vectors and then Options > Normalize Color Range.

Basic Node Setup

The Optical Flow node analyzes the frames connected to the background input. Trimming a Loader or MediaIn to only the range you need prevents analyzing unnecessary frames. The output of the node can then be connected to any node that takes advantage of vector channels, such as a Time Stretcher.

> **TIP:** When analyzing Optical Flow vectors, consider adding a Smooth Motion node afterward with smoothing for forward/ backward vectors enabled.

An Optical Flow node generating motion vectors on-the-fly.

Alternatively, if you find the Optical Flow node too slow to analyze the frames, consider rendering it out to an OpenEXR format using a Saver node. Then import the rendered EXR file as your new image with embedded vector channels.

An Optical Flow node rendered out through a Saver node.

Inspector
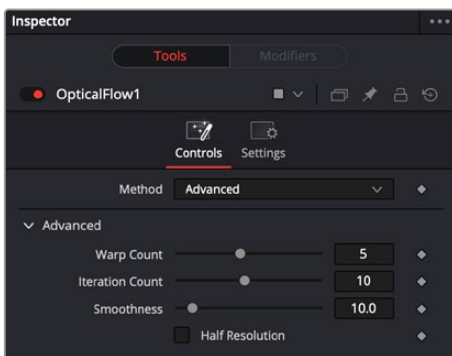
Optical Flow advanced controls

Controls Tab (Advanced)

When you add an Optical Flow, Repair Frame, or Tween node to a Comp, a Method drop-down menu in the Inspector allows you to choose between an Advanced GPU-based algorithm or a Classic CPU-based algorithm. This Advanced method is the same Optical Flow algorithm used in other DaVinci Resolve pages.

Warp Count

Decreasing this slider makes the optical flow computations faster. To understand what this option does, you must understand that the optical flow algorithm progressively warps one image until it matches with the other image. After some point, convergence is reached, and additional warps become a waste of computational time. You can tweak this value to speed up the computations, but it is good to watch what the optical flow is doing at the same time.

Iteration Count

Decreasing this slider makes the computations faster. In particular, just like adjusting the Warp Count, adjusting this option higher will eventually yield diminishing returns and not produce significantly better results. By default, this value is set to something that should converge for all possible shots and can be tweaked lower fairly often without reducing the disparity's quality.

Smoothness

This controls the smoothness of the optical flow. Higher smoothness helps deal with noise, while lower smoothness brings out more detail.

Half Resolution

The Half Resolution checkbox is used purely to speed up the calculation of the optical flow. The input images are resized down and tracked to produce the optical flow.

Controls Tab (Classic)

By choosing Classic from the Method drop-down menu in the Inspector, you can use the older CPU-based algorithm to maintain compatibility with Comps created in previous versions. This method may also be better suited for some Stereo3D processing.

When using the Classic method, a single slider at the top of the Inspector improves performance by generating proxies. The remaining Advanced section parameters tune the Optical Flow vector calculations. The default settings serve as a good standard. In most cases, tweaking of the advanced settings is not needed. Many deliver small or diminishing returns. However, depending on the settings, rendering time can easily vary by 10x. If you're interested in reducing process time, it is best to start by experimenting with the Proxy, Number of Iterations, and Number of Warps sliders and changing the filtering to Bilinear.

Proxy (for Tracking)

The Proxy slider is used purely to speed up the calculation of the optical flow. The input images are resized down by the proxy scale and tracked to produce the optical flow. The computational time is roughly proportional to the number of pixels in the image. This means a proxy scale of 2 will give a 4x speedup, and a proxy scale of 3 will give a 9x speedup.

Smoothness

This controls the smoothness of the optical flow. Higher smoothness helps deal with noise, while lower smoothness brings out more detail.

Edges

This slider is another control for smoothness but applies it based on the color channel. It tends to have the effect of determining how edges in the flow follow edges in the color images. When it is set to a low value, the optical flow becomes smoother and tends to overshoot edges. When it is set to a high value, details from the color images start to slip into the optical flow, which is not desirable. Edges in the flow end up more tightly aligning with the edges in the color images. This can result in streaked-out edges when the optical flow is used for interpolation. As a rough guideline, if you are using the disparity to produce a Z-channel for post effects like Depth of Field, then set it lower in value. If you are using the disparity to perform interpolation, you might want it to be higher in value.

Match Weight

This control sets a threshold for how neighboring groups of foreground/background pixels are matched over several frames. When set to a low value, large structural color features are matched. When set to higher values, small sharp variations in the color are matched. Typically, a good value for this slider is in the [0.7, 0.9] range. When dealing with stereo 3D, setting this option higher tends to improve the matching results in the presence of differences due to smoothly varying shadows or local lighting variations between the left and right images. The user should still perform a color match or deflickering on the initial images, if necessary, so they are as similar as possible. This option also helps with local variations like lighting differences due to light passing through a mirror rig.

Mismatch Penalty

This option controls how the penalty for mismatched regions grows as they become more dissimilar. The slider provides a choice between a balance of Quadratic and Linear penalties. Quadratic strongly penalizes large dissimilarities, while Linear is more robust to dissimilar matches. Moving this slider toward Quadratic tends to give a disparity with more small random variations in it, while Linear produces smoother, more visually pleasing results.

Warp Count

Decreasing this slider makes the optical flow computations faster. In particular, the computational time depends linearly upon this option. To understand what this option does, you must understand that the optical flow algorithm progressively warps one image until it matches with the other image. After some point, convergence is reached, and additional warps become a waste of computational time. The default value in Fusion is set high enough that convergence should always be reached. You can tweak this value to speed up the computations, but it is good to watch what the optical flow is doing at the same time.

Iteration Count

Decreasing this slider makes the computations faster. In particular, the computational time depends linearly upon this option. Just like adjusting the Warp Count, adjusting this option higher will eventually yield diminishing returns and not produce significantly better results. By default, this value is set to something that should converge for all possible shots and can be tweaked lower fairly often without reducing the disparity's quality.

Filtering

This option controls filtering operations used during flow generation. Catmull-Rom filtering will produce better results, but at the same time, turning on Catmull-Rom will increase the computation time steeply.
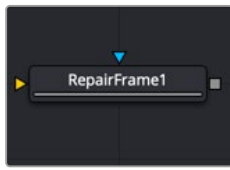
Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Optical Flow nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Repair Frame [Rep]

The Repair Frame node

Repair Frame Node Introduction

Repair Frame replaces damaged or missing frames or portions of frames with scratches or other temporally transient artifacts. It requires three frames: the repair frame and two neighboring frames. An Optical Flow node is not required for generating motion vectors since the Repair Frame node computes the optical flow. However, this can make it slow to process.

Repair Frame will not pass through, but rather destroys, any aux channels after the computation is done.

See the Optical Flow node for controls and settings information.

> **TIP:** If your footage varies in color from frame to frame, sometimes the repair can be noticeable because, to fill in the hole, Repair Frame must pull color values from adjacent frames. Consider using deflickering, color correction, or using a soft-edged mask to help reduce these kinds of artifacts.

Inputs

There are two inputs on the Repair Frame node. One is used to connect a 2D image that will be repaired and the other is for an effect mask.

— **Input:** The orange input is used for the primary 2D image that will be repaired.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the repairs to certain areas.

Basic Node Setup

The Repair Frame node analyzes the incoming MediaIn node and repairs single frame issues like dust or scratches.

A Repair Frame node set up to analyze a MediaIn node using internal optical flow analysis.

Inspector
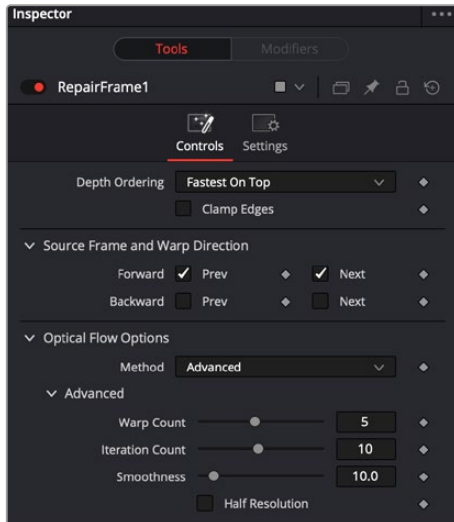
The Repair Frame Controls tab

Controls Tab

The Controls tab includes options for how to repair the frames. It also includes controls for adjusting the optical flow analysis, identical to those controls in the Optical Flow node.

Depth Ordering

The Depth Ordering determines which parts of the image should be rendered on top by selecting either Fastest On Top or Slowest On Top. The examples below best explain these options.

In a locked-off camera shot where a car is moving through the frame, the background does not move, so it produces small, or slow, vectors, while the car produces larger, or faster, vectors.

The depth ordering in this case is Fastest On Top since the car draws over the background.

In a shot where the camera pans to follow the car, the background has faster vectors, and the car has slower vectors, so the Depth Ordering method is Slowest On Top.

Clamp Edges

Under certain circumstances, this option can remove the transparent gaps that may appear on the edges of interpolated frames. Clamp Edges causes a stretching artifact near the edges of the frame that is especially visible with objects moving through it or when the camera is moving.

Because of these artifacts, it is a good idea to use clamp edges only to correct small gaps around the edges of an interpolated frame.

Edge Softness

This slider is displayed only when Clamp Edges is enabled. The slider helps to reduce the stretchy artifacts that might be introduced by Clamp Edges.

If you have more than one of the Source Frame and Warp Direction checkboxes turned on, this can lead to doubling up of the stretching effect near the edges. In this case, you'll want to keep the softness rather small at around 0.01. If you have only one checkbox enabled, you can use a larger softness at around 0.03.

Source Frame and Warp Direction

These checkboxes allow you to choose which frames and vectors create the in-between frames. Each method ticked on will be blended into the result.

— **Prev Forward:** Takes the previous frame and uses the Forward vector to interpolate the new frame.

— **Next Forward:** Takes the next frame in the sequence and uses the Forward vector to interpolate the new frame.

— **Prev Backward:** Takes the previous frame and uses the Back Forward vector to interpolate the new frame.

— **Next Backward:** Takes the next frame in the sequence and uses the Back vector to interpolate the new frame.

Optical Flow Options

These settings tweak the optical flow analysis. See the Classic and Advanced Controls section for the Optical Flow node earlier in this chapter.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Optical Flow nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Smooth Motion [SM]

The Smooth Motion node

Smooth Motion Node Introduction

The Smooth Motion node smooths various AOV (Arbitrary Output Variables) channels in a clip using optical flow to look at neighboring frames. It can be used for smoothing the Disparity channel in a stereo 3D clip, where it helps reduce temporal edge/fringing artifacts, but it can also smooth a wide range of channels like vectors, normals, and Z.

It is required that the image connected to the input on the node have precomputed Vector and Back Vector channels; otherwise, this tool prints error messages in the Console window.

Check on the channels you want to temporally smooth. Be aware that if a channel selected for smoothing is not present, Smooth Motion will not fail, nor will it print any error messages.

It can also be used to smooth the Vector and Back Vector channels; however, sometimes, this can make the interpolated results worse if there are conflicting motions or objects in the shot that move around erratically, jitter, or bounce rapidly.

> **TIP:** You can use two or more Smooth Motion nodes in sequence to get additional smoothing. With one Smooth Motion node, the previous, current, and next frames are examined for a total of 3; with two Smooth Motion nodes, 5 frames are examined; and with three Smooth Motion nodes, 7 frames are examined.

Another technique using two Smooth Motion nodes is to use the first Smooth Motion node to smooth the Vector and Back Vector channels. Use the second Smooth Motion to smooth the channels you want to smooth (e.g., Disparity). This way, you use the smoothed vector channels to smooth Disparity.

You can also try using the smoothed motion channels to smooth the motion channels.

Inputs

The Smooth Motion node includes a single orange image input.

— **Input:** The orange image input accepts a 2D image. This is the sequence of images for which you want to compute smooth motion. This image must have precomputed Vector and Back Vector channels either generated from an Optical Flow node or saved in EXR format with vector channels.

Basic Node Setup

The Smooth Motion node takes the output of the Optical Flow node for the required Vector and Back Vector channels. The Smooth Motion node can then be used to smooth those channels or AO channels.

A Smooth Motion node using Vector and Back Vector channels from the Optical Flow node.

Inspector
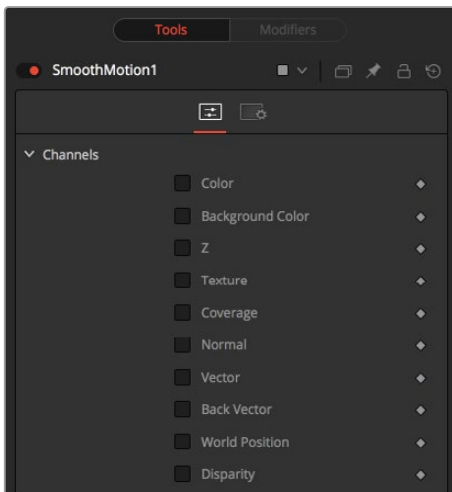
The Smooth Motion Controls tab

Controls Tab

The Controls tab includes checkboxes for the channels you want to smooth. If a channel selected for smoothing is not available in the input image, Smooth Motion will not fail, nor will it print any error messages to the Console.

Channel

Smooth Motion can be applied to more than just the RGBA channels. It can also be applied to the other AOV channels.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Optical Flow nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Tween [Tw]

The Tween node

Tween Node Introduction

Tween reconstructs a missing frame by interpolating between two neighboring frames using the optical flow. Tween is nearly identical in functionality to Time Speed and Time Stretcher. The major difference is that it works on two images that are not serial members of a sequence. As a consequence, it cannot use the Vector or Back Vector aux channels stored in the images. The Tween node manually generates the optical flow, so there is no need to add an Optical Flow node before the Tween node. The generated optical flow is thrown away and is not stored back into the output frames.

Since optical flow is based on color matching, it is a good idea to color correct your images to match ahead of time. Also, if you are having trouble with noisy images, it may also help to remove some of the noise ahead of time.

Tween destroys any input aux channels. See the Optical Flow node for controls and settings information.

Inputs

There are two image inputs on the Tween node and an effects mask input.

— **Input 0:** The orange input, labeled input 0, is the previous frame to the one you are generating.

— **Input 1:** The green input, labeled input 1, is the next frame after the one you are generating.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Tween to certain areas.

Basic Node Setup

The Tween node receives two inputs for the two neighboring frames to the one you are generating. Below, the previous frame, frame 01, is connected to the orange input 0. The next frame, frame 03, is connected to the green input 1. The Tween node will generate frame 02 and output the sequence.

The Tween node receives two neighboring frames and generates the middle one.

Inspector
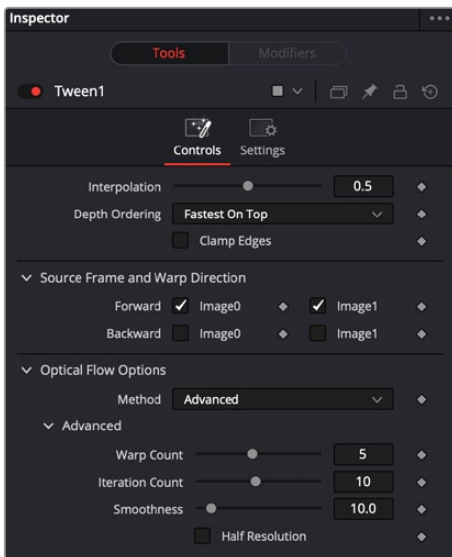
The Tween Controls tab

Controls Tab

The Controls tab includes options for how to tween frames. It also includes controls for adjusting the optical flow analysis, identical to those controls in the Optical Flow node.

Interpolation Parameter

This option determines where the frame you are interpolating is, relative to the two source frames A and B. An Interpolation Parameter of 0.0 will result in frame A, a parameter of 1.0 will result in frame B, and a parameter of 0.5 will yield a result halfway between A and B.

Depth Ordering

The Depth Ordering determines which parts of the image should be rendered on top by selecting either Fastest On Top or Slowest On Top. The examples below best explain these options.

In a locked-off camera shot where a car is moving through the frame, the background does not move, so it produces small, or slow, vectors, while the car produces larger, or faster, vectors.

The Depth Ordering in this case is Fastest On Top since the car draws over the background.

In a shot where the camera pans to follow the car, the background has faster vectors, and the car has slower vectors, so the Depth Ordering method is Slowest On Top.

Clamp Edges

Under certain circumstances, this option can remove the transparent gaps that may appear on the edges of interpolated frames. Clamp Edges causes a stretching artifact near the edges of the frame that is especially visible with objects moving through it or when the camera is moving.

Because of these artifacts, it is a good idea to use Clamp Edges only to correct small gaps around the edges of an interpolated frame.

Edge Softness

This slider is displayed only when Clamp Edges is enabled. The slider helps to reduce the stretchy artifacts that might be introduced by Clamp Edges.

If you have more than one of the Source Frame and Warp Direction checkboxes turned on, this can lead to doubling up of the stretching effect near the edges. In this case, you'll want to keep the softness rather small at around 0.01. If you have only one checkbox enabled, you can use a larger softness at around 0.03.

Source Frame and Warp Direction

These checkboxes allow you to choose which frames and vectors create the in-between frames. Each method ticked on will be blended into the result.

— **Prev Forward:** Takes the previous frame and uses the Forward vector to interpolate the new frame.

— **Next Forward:** Takes the next frame in the sequence and uses the Forward vector to interpolate the new frame.

— **Prev Backward:** Takes the previous frame and uses the Back Forward vector to interpolate the new frame.

— **Next Backward:** Takes the next frame in the sequence and uses the Back vector to interpolate the new frame.

Optical Flow Options

These settings tweak the optical flow analysis. See the Class and Advanced Controls section for the Optical Flow node earlier in this chapter.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Optical Flow nodes. These common controls are described in detail in the following "The Common Controls" section.

The Common Controls

Nodes that handle optical flow operations share a number of identical controls in the Inspector. This section describes controls that are common among Optical Flow nodes.

Inspector
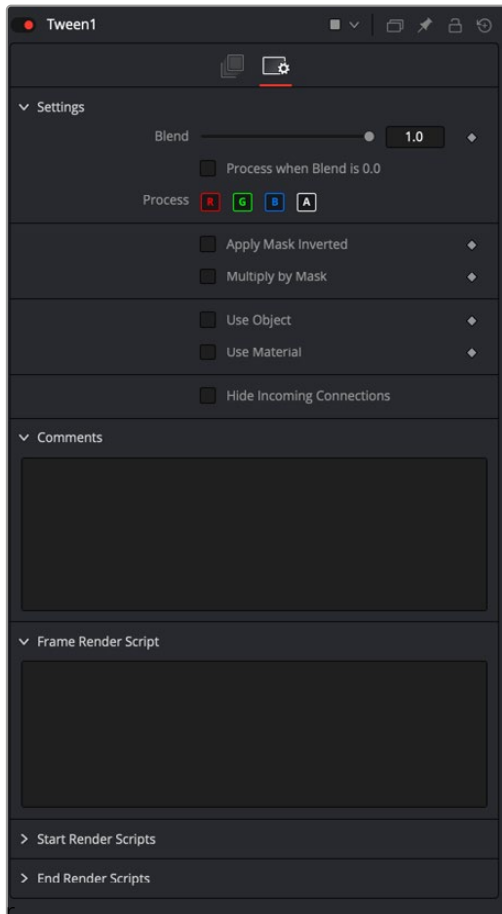
The Common Optical Flow Settings tab

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Optical Flow category. The controls are consistent and work the same way for each tool.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this causes the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful when this node process is scripted to trigger another task, but the value of the node is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the Red button on a Blur tool is deselected, the blur is first applied to the image, and then the red channel from the original input will be copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option causes the RGB values of the masked image to be multiplied by the mask channel's values. This causes all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object ID and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels are used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID is used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the view. The image or sequence must have been rendered from a 3D software package with those channels included.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node are displayed in the Inspector. Dragging a connected node from the node tree into the empty field hides that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line reappears.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, see the Fusion scripting documentation.

Paint Node

This chapter details the Paint node available in Fusion.

The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Paint

The Paint node

Paint Node Introduction

Paint is an extremely flexible, stroke-based tool for wire and rig removal, image cloning, or to create custom masks and mattes rapidly. Fusion's paint can even be used to create new images and artistic elements from scratch.

Each Paint node is made up of a series of brush strokes. These strokes are vector shapes with editable brush, size, and effect. A wide range of apply modes and brush types are available.

Most Brushstrokes styles are editable polylines for fine control. They can be animated to change shape, length, and size over time. The opacity and size of a stroke can be affected by velocity and pressure when used with a supported tablet.

Unlimited undo and redo of paint provides the ability to experiment before committing changes to an image sequence. Paint strokes can be reordered, deleted, and modified with virtually infinite flexibility.

Inputs

The two inputs on the Paint node are used to connect a 2D image and an effect mask which can be used to limit the painted area.

— **Input:** It is required to connect the orange input with a 2D image that creates the size of the "canvas" on which you paint.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Paint to only those pixels within the mask.

Basic Node Setup

The Paint node always needs an input connection. The simplest setup is to paint directly on the incoming MediaIn node.
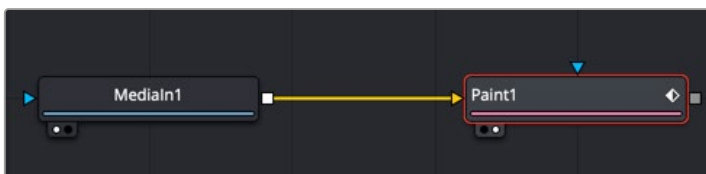
A MediaIn node connected directly to the Paint input

A more flexible setup is to use a Background node to set the size that matches the image you are painting on. In the Inspector, the background would be set to be fully transparent. Then, the Paint tool can be merged as the foreground over the actual image you want to paint on.

A Paint node merged over the top of a MediaIn for more flexibility

Types of Paint Strokes
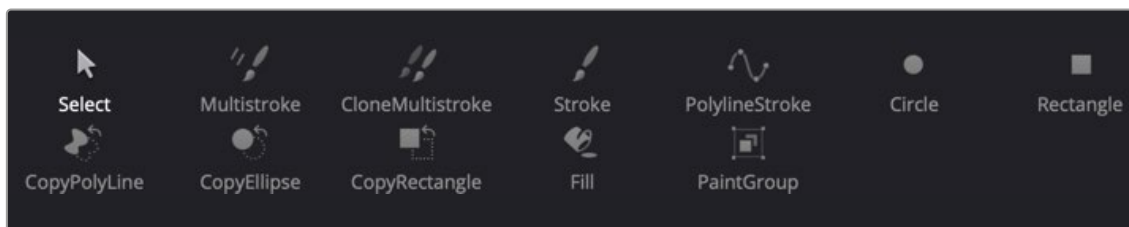
Paint stroke types located in the viewer toolbar.

To begin working with the Paint tool, first select the paint stroke type from the Paint toolbar above the viewer. There are ten stroke types to choose from as well as two additional tools for selecting and grouping paint strokes. The stroke types and tools are described below in the order they appear in the toolbar.

— **Multistroke:** Although this is the default selection and the first actual brush type in the toolbar, Multistroke is not typically the stroke type most often used. However, it's perfect for those 100-strokes-per-frame retouching paint jobs like removing tracking markers. Multistroke is much faster than the Stroke type but is not editable after it is created. By default, Multistroke lasts for one frame and cannot be modified after it has been painted. Use the Duration setting in the Stroke controls to set the number of frames before painting. A shaded area of the Multistroke duration is visible but not editable in the Keyframes Editor. While Multistrokes aren't directly editable, they can be grouped with the PaintGroup modifier, then tracked, moved, and rotated by animating the PaintGroup instead.

— **Clone Multistroke:** Similar to Multistroke but specifically meant to clone elements from one area or image to the other. Perfect for those 100-strokes-per-frame retouching paint jobs like removing tracking markers. Clone Multistroke is faster than the Stroke type but is not editable after it is created. By default, Clone Multistroke lasts for one frame and cannot be modified after it has been painted. Use the Duration setting in the Stroke controls to set the number of frames before painting. A shaded area of the Clone Multistroke duration is visible but not editable in the Keyframes Editor.

— **Stroke:** In most cases, the Stroke tool is what people think of when they think of paint and is the tool of choice for most operations. It is a fully animatable and editable vector-based paint stroke. It can become slow if hundreds of strokes are used in an image; when creating a lot of paint strokes, it is better to use Multistroke. The Stroke type has a duration of the entire global range. However, you can edit its duration at any time in the Keyframes Editor. When the painting is complete, choose the Select button in the Paint toolbar to avoid accidentally adding new strokes.

— **Polyline Stroke:** This provides the ability to create and manipulate a stroke in the same way that a Bézier path or polygon mask might be created. To add a Polyline Stroke, select the Polyline button and click in the viewer to add the first point. Continue clicking to add additional points to the polyline. This click append style is the default, but polyline strokes can also be created in draw append mode. Polylines can be tracked or connected to existing polylines like masks or animation paths. The Polyline Stroke has a default duration of the entire global range. However, you can edit its duration at any time in the Keyframes Editor.

— **Circle:** Creates a circular shape with animatable control over radius and center. The Circle type has a duration of the entire global range. However, you can edit its duration at any time in the Keyframes Editor.

— **Rectangle:** Creates a rectangular area. The Rectangle type has a duration of the entire global range. However, you can edit its duration at anytime in the Keyframes Editor.

— **Copy Polyline:** Allows you to create a closed Polyline area with animatable offset to clone elements from one area or image to the other. The Copy Polyline type has a duration of the entire global range. However, you can edit its duration at any time in the Keyframes Editor.

— **Copy Circle/Rectangle:** A circular or rectangular shape area with animatable offset to clone elements from one image to the other. The Copy Circle/Rectangle type has a duration of the entire global range. However, you can edit its duration at any time in the Keyframes Editor.

— **Fill:** Similar to the Wand mask tool. This tool fills similarly colored adjacent pixels with a fill color based on the selected color channel. The tool has a duration of the entire global range. However, you can edit its duration at any time in the Keyframes Editor

— **Paint Group:** Allows you to group multiple paint strokes with full control over center and size. Since Multistroke and Clone Multistroke styles cannot be tracked, grouping these paint stroke types and tracking the paint group is one common use of groups.

Editing Options Toolbar
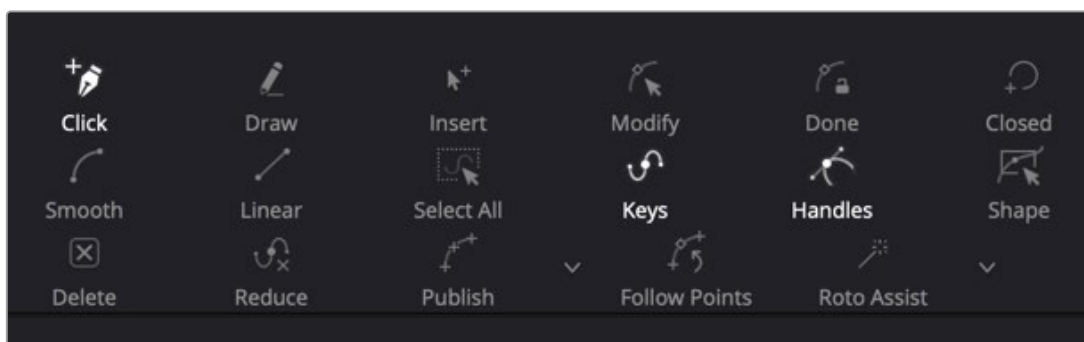
Paint edit options are displayed in the viewer after a Polyline stroke is created.

Polyline-based paint strokes include a second toolbar in the viewer to select different editing options. The paint strokes that include this second toolbar are Polyline Stroke and Copy Polyline. The Stroke style also displays this toolbar after the stroke is selected and the Make Editable button is clicked in the Inspector.

— **Click Append:** This is the default option when creating a polyline stroke. It works more like a Bézier pen drawing tool than a paintbrush tool. Clicking sets a control point and appends the next control point when you click again in a different location.

— **Draw Append:** This is a freehand drawing tool. It paints a stroke similar to drawing with a pencil on paper. You can create a new Polyline Stroke or Copy Polyline Stroke using the Draw tool, or you can extend a Stroke style after clicking the Make Editable button in the Inspector.

— **Insert:** Insert adds a new control point along the paint stroke spline.

— **Modify:** Modify allows you to safely move or smooth any exiting point along a spline without worrying about adding a new point accidentally.

— **Done:** Prevents any point along the spline from being moved or modified. Also, new points cannot be added. You can, however, move and rotate the entire spline.

— **Closed:** Closes an open polyline.

— **Smooth:** Changes the selected stroke or control point from a linear to a smooth curve.

— **Linear:** Changes the selected stroke or control point from a smooth curve to linear.

— **Select All:** Selects all the control points on the polyline.

— **Keys:** Shows or hides the control points along the polyline.

— **Handles:** Shows or hides the Bézier handles along the polyline.

— **Shape:** Places a reshape rectangle around the selected polyline control points. Using the reshape rectangle, you can deform groups of polyline control points or entire shapes much easier than modifying each point.

— **Delete:** Deletes the selected control point(s).

— **Reduce:** Opens a Freehand precision window that can be used to reduce the number of control points on a polyline. This can make the paint stroke easier to modify, especially if it has been created using the Draw tool.

— **Publish:** You can use the Publish menu to either publish control points or the path. Publishing is a form of parameter linking. It makes the selected item available for use by other controls, or to attach a control point to a tracker.

— **Follow Points:** Allows a selected point to follow the path of a published point. The point follows the published point using an offset position.

— **Roto Assist:** Enable the Roto Assist button when you begin painting with the Polyline Stroke tool. The polyline points snap to the closest edge as you click to add points to the shape. A cyan outline indicates the points that have snapped to an edge. There are three main Roto Assist options selectable through the drop-down menu:

  — **Multiple Points:** When enabled, a single click on a high-contrast edge will add multiple points to define the entire edge, instead of having to add each point individually. This is a one time only click. The second click reverts to single point edge detection.

  — **Distance 8:** Opens a dialog where you can set the pixel range within which searching for an edge will take place.

  — **Reset:** Used for resetting the snap attribute of all snapped points. After resetting, the points will become unavailable for tracking.

Inspector
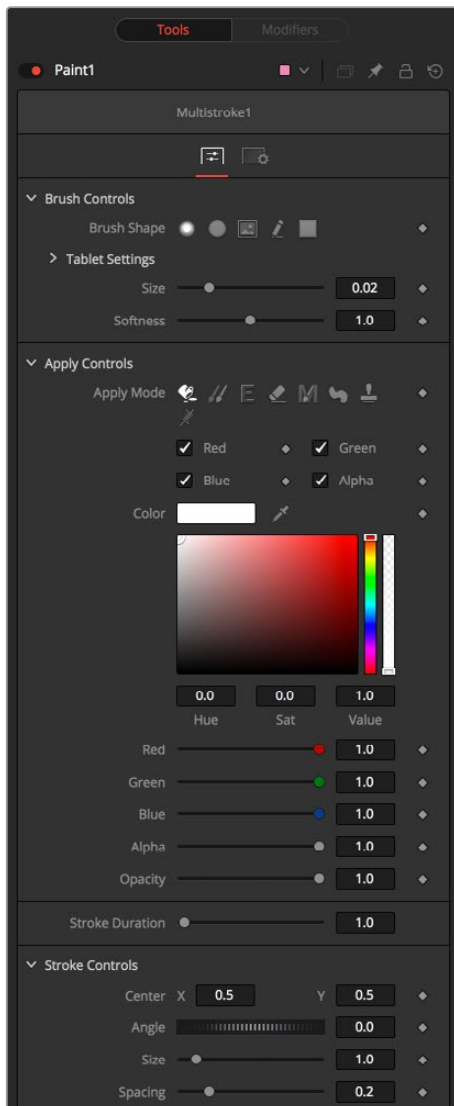
The Paint Controls tab

Controls Tab

Not all of the controls described here appear in all modes. Some controls are useful only in a specific Paint mode and do not appear when they are not applicable. The Controls tab is used to configure your paint settings before painting. Once a paint stroke is created, except for the Multistroke and Clone Multistroke, you can select the stroke in the viewer and update the controls.

Brush Controls

Brush Shape

The brush shape buttons select the brush tip shape. Except for the single pixel shape, you can modify the size of the brush shape in the viewer by holding down the Command or Ctrl key while dragging the mouse.

— **Soft Brush:** The Soft Brush type is a circular brush tip with soft edges.

— **Circular Brush:** A Circular Brush is a brush tip shape with hard edges.

— **Image Brush:** The Image Brush allows images from any node in the node tree, or from a file system, to be used as a brush tip.

— **Single Pixel Brush:** The Single Pixel Brush is perfect for fine detail work, creating a brush tip precisely one pixel in size. No anti-aliasing is applied to the single pixel brush.

— **Square Brush:** A Square Brush is a brush tip with hard edges.

Vary Size

Vary size settings change the stroke size based on speed or a pressure-sensitive pen and tablet.

— **Constant:** The brush tip remains a constant size over the stroke.

— **With Pressure:** The stroke size varies with the actual applied pressure.

— **With Velocity:** The stroke size varies with the speed of painting. The faster the stroke, the thinner it is.

Vary Opacity

Vary opacity settings change the stroke opacity based on speed or a pressure-sensitive pen and tablet.

— **Constant:** The brush tip remains at a constant transparency setting over the entire stroke.

— **With Pressure:** The stroke transparency varies with the applied pressure.

— **With Velocity:** The stroke transparency varies with the speed of painting. The faster the stroke, the more transparent it is.

Softness

Use this control to increase or decrease the Softness of a soft brush.

Image Source

When using the Image Source brush type, select between three possible source brush images.

— **Node:** The image source is derived from the output of a node in the node tree. Drag the node into the Inspector's Source node input field to set the source.

— **Clip:** The image source is derived from an image or sequence on disk. Any file supported by Fusion's Loader or MediaIn node can be used.

— **Brush:** Select an image to use as a brush from the menu. Images located in the Fusion > Brushes directory are used to populate the menu.

Color Space

When the Fill tool is selected, a Color Space menu selects the color space when sampling colors around the Fill tool center for inclusion in the fill range.

Channel

When the Fill tool is selected, a Channel menu selects which color channel is used in the fill paint. For example, with alpha selected, the fill occurs on contiguous pixels of the alpha channel.

Apply Controls

Apply Mode

The Apply Modes are buttons that change a brush's painting functionality.

— **Color:** The Color Apply Mode paints simple colored strokes. When used in conjunction with an image brush, it can also be used to tint the image.

— **Clone:** The Clone Apply Mode copies an area from the same image using adjustable positions and time offsets. This mode can also copy portions of one image into another image. Any image from the node tree can be used as the source image.

— **Emboss:** The Emboss Apply Mode embosses the portions of the image covered by the brush stroke.

— **Erase:** Erase reveals the underlying image through all other strokes, effectively erasing portions of the strokes beneath it without actually destroying the strokes.

— **Merge:** This Apply Mode effectively merges the brush onto the image. This mode behaves in much the same way as the Color Apply Mode but has no color controls. It is best suited for use with the image brush type.

— **Smear:** Smear the image using the direction and strength of the brushstroke as a guide.

— **Stamp:** Stamps the brush onto the image, completely ignoring any alpha channel or transparency information. This mode is best suited for applying decals to the target image.

— **Wire:** This Wire Removal Mode is used to remove wires, rigging, and other small elements in the frame by sampling adjacent pixels and draw them in toward the stroke.

Stroke Controls

The stroke controls contain parameters that adjust the entire stroke of paint as well as control it over time.

— **Size:** This control adjusts the size of the brush when the brush type is set to either Soft Brush or Circle. The diameter of the brush is drawn in the viewer as a small circle surrounding the mouse pointer. The size can also be adjusted interactively in the viewer by holding the Command or Ctrl key while dragging the mouse pointer.

— **Spacing:** The Spacing slider determines the distance between dabs (samples used to draw a continuous stroke along the underlying vector shape). Increasing this value increases the density of the stroke, whereas decreasing this value causes the stroke to assume the appearance of a dotted line.

— **Stroke Animation:** The Stroke Animation menu provides several pre-built animation effects that can be applied to a paint stroke. This menu appears only for vector strokes like Stroke and Polyline Stroke.

— **All Frames:** This default displays the stroke for all frames of the image connected to the orange background input of the Paint node.

— **Limited Duration:** This exists on the number of frames specified by the Duration slider.

— **Write On:** When Write On is selected, an animation spline is added to the paint stroke that precisely duplicates the timing of the paint stroke's creation. The stroke is written on the image exactly as it was drawn. To adjust the timing of the Write On effect, switch to the Spline Editor and use the Time Stretcher node to adjust the overall length of the animation spline. To smooth or manually adjust the motion, try reducing the points in the animation spline.

- **Write Off:** Write Off performs the reverse of Write On, drawing the stroke starting from the end and working backward to the start of the stroke.
- **Write On Then Off:** This menu option applies a Write On and then a Write Off animation mode to the stroke.
- **Trail:** Selecting the Trail option from the menu causes both the start and end points of the stroke to be animated simultaneously, offset from each other by the amount specified in the Duration control. This has the effect of creating a painted segment that follows the stroke path. As with the Write On and Write Off effects, this starts on the current frame when the animation mode is selected. The timing of the animation can be adjusted manually using the Spline or Keyframes Editors.

- **Duration:** Duration sets the duration of each stroke in frames. This control is present only for Multistroke and Clone Multistroke, or when the stroke animation mode is set to Limited Duration. It is most commonly employed for frame-by-frame rotoscoping through a scene.

  Each Vector stroke applied to a scene has a duration in the Keyframes Editor that can be trimmed independently from one stroke to the next. The duration can be set to 0.5, which allows each stroke to last for a single field only when the node tree is processing in Fields mode.

- **Write On and Write Off:** This range slider appears when the Stroke Animation is set to one of the Write On and Write Off methods. The range represents the beginning and end points of the stroke. Increase the Start value from 0.0 to 1.0 to erase the stroke, or increase the End value from 0.0 to 1.0 to draw the stroke on the screen. This control can be animated to good effect. It works most effectively when automatically animated through the use of the Write On and Write Off modes of the Stroke Animation menu.

- **Make Editable:** This button appears only for Vector strokes. Clicking on Make Editable turns the current stroke into a polyline spline so that the shape can be adjusted or animated.

Paint Node Modifiers

Every paint stroke created in the viewer creates an associated modifier stroke. These modifier strokes are represented as a list of paint stroke operations in the Modifiers tab of the Inspector. Each stroke you create can be modified or deleted, or applied in a different order using the modifier stack.

> **NOTE:** The MultiStroke tools are built for speed and can contain many strokes internally without creating a huge list stack in the modifiers

Each Paint modifier stroke contains Brush controls, Apply controls, and Stroke controls identical to those found in the main Controls tab of the Inspector.

Keyboard Shortcuts

Keyboard shortcuts allow you to adjust painting styles and color without having to navigate menus.

While painting:

Hold Command or Ctrl while left-dragging to change brush size.

Hold Option or Alt while clicking to pick a color in the viewer.

While cloning:

Option-click or Alt-click to set the clone source position. Strokes start cloning from the selected location.

Hold O to temporarily enable a 50% transparent overlay of the clone source (% can be changed with pref Tweaks.CloneOverlayBlend).

Press P to toggle an opaque overlay of the clone source.

While overlay is showing:

Arrow keys change the clone source position; you can also drag crosshairs and adjust angle control or size sliders.

Option + Left/Right or Alt + Left/Right Arrow keys change the clone source angle.

Option + Up/Down or Alt + Up/Down Arrow keys change the clone source size.

Shift + Command or Shift + Ctrl can be used with the above for greater or lesser adjustments. Left and right square brackets, [and], change the clone source Time Offset (this requires a specific Clone Source node to be set in the Source Node field).

Copy Rect/Ellipse:

Shift + drag out the source to constrain the shape.

With a single stroke selected (not available on multi or polyline strokes):

Press X or Y to flip the stroke.

Paint Groups:

Command + drag or Ctrl + drag to change the position of a group's crosshair, without changing the position of the group.

Particle Nodes

This chapter details the Particle nodes available in Fusion.

The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Particle Nodes

The Particle nodes are used to generate a large number of duplicated objects that automatically animate. They are used to create elements like falling rain, fireworks, smoke, pixie dust, and much more. There are endless possibilities. Particles in Fusion consist of a set of nodes that are strung together in a chain for generating, modifying, and rendering particles in a 2D or 3D scene.

To begin, every particle system you create must contain two fundamental nodes:

— **pEmitter:** Used to generate the particles and control their basic look, motion. and behavior.

— **pRender:** Used to render the output of the pEmitter into a 2D or 3D scene. When creating particles, you only ever view the pRender node.
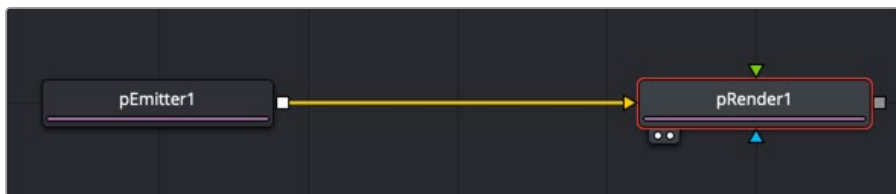
Particle Nodes

The remaining particle nodes modify the pEmitter results to simulate natural phenomena like gravity, flocking, and bounce. The names of particle nodes all begin with a lowercase p to differentiate them from non-particle nodes. They can be found in the particles category in the Effects Library.

pAvoid [pAv]
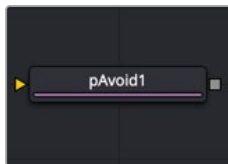
The pAvoid node

pAvoid Node Introduction

The pAvoid node is used to create a region or area within the image that affected particles attempt to avoid entering and crossing.

It has two primary controls. The first determines the distance from the region a particle should be before it begins to move away from the region. The second determines how strongly the particle moves away from the region.

A pAvoid node creates a "desire" in a particle to move away from a specific region. If the velocity of the particle is stronger than the combined distance and strength of the pAvoid region, the particle's desire to avoid the region does not overcome its momentum and the particle crosses that region regardless.

Inputs

The pAvoid node has a single orange input by default. Like most particle nodes, this orange input accepts only other particle nodes. A green bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.

— **Region:** The green or magenta region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area particles avoid.

Basic Node Setup

The pAvoid node is placed in between the pEmitter and pRender. A Shape 3D node is used to create the region the particles will avoid.
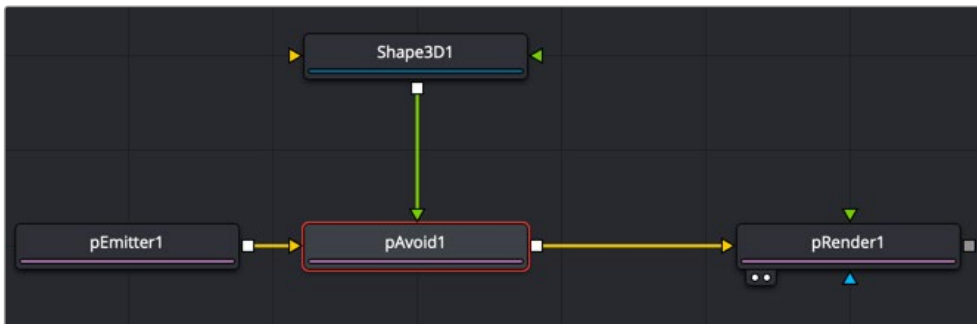
A pAvoid node using a Shape 3D node as the region to avoid

Inspector
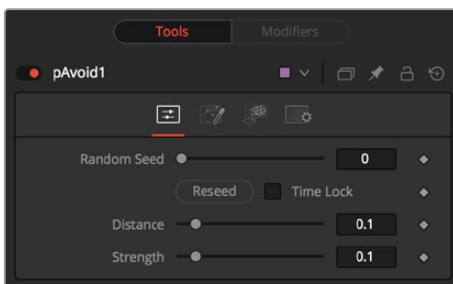
The pAvoid controls

Randomize

The Random Seed slider and Randomize button are presented whenever a Fusion node relies on a random result. Two nodes with the same seed values will produce the same random results. Click the Randomize button to randomly select a new seed value, or adjust the slider to manually select a new seed value.

Distance

Determines the distance from the region a particle should be before it begins to move away from the region.

Strength

Determines how strongly the particle moves away from the region. Negative values make the particles move toward the region instead.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pBounce [pBn]
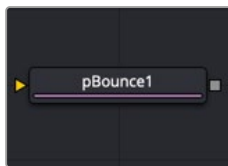
The pBounce node

pBounce Node Introduction

The pBounce tool is used to create a region from which affected particles will bounce off when they come into contact.

Inputs

The pBounce node has a single orange input by default. Like most particle nodes, this orange input accepts only other particle nodes. A green or magenta bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Region:** The green or magenta region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area particles bounce off.

Basic Node Setup

The pBounce node is placed in between the pEmitter and pRender. A Shape 3D node is used to create the region the particles bounce off.
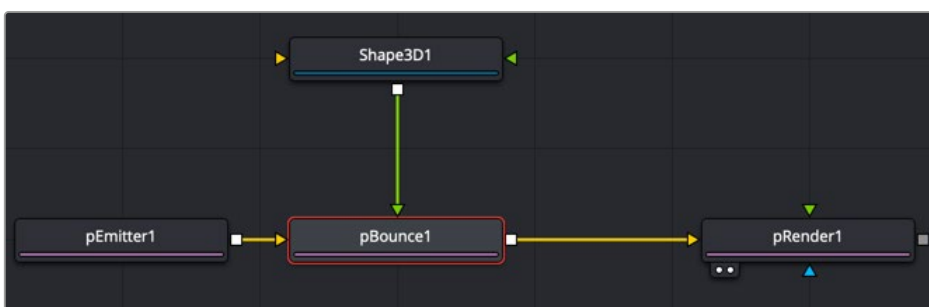
A pBounce node using a Shape 3D node as the region on which particles bounce off

Inspector
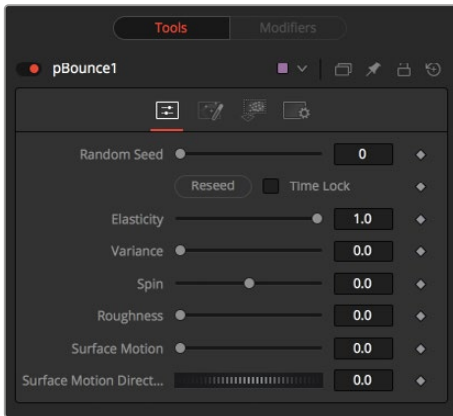
The pBounce controls

Randomize

The Random Seed slider and Randomize button are presented whenever a Fusion node relies on a random result.

Two nodes with the same seed values will produce the same random results. Click the Randomize button to randomly select a new seed value, or adjust the slider to manually select a new seed value.

Elasticity

Elasticity affects the strength of a bounce, or how much velocity the particle will have remaining after impacting upon the Bounce region. A value of 1.0 will cause the particle to possess the same velocity after the bounce as it had entering the bounce. A value of 0.1 will cause the particle to lose 90% of its velocity upon bouncing off of the region.

The range of this control is 0.0 to 1.0 by default, but greater values can be entered manually. This will cause the particles to gain momentum after an impact, rather than lose it. Negative values will be accepted but do not produce a useful result.

Variance

By default, particles that strike the Bounce region will reflect evenly off the edge of the Bounce region, according to the vector or angle of the region. Increasing the Variance above 0.0 will introduce a degree of variation to that angle of reflection. This can be used to simulate the effect of a rougher surface.

Spin

By default, particles that strike the region will not have their angle or orientation affected in any way. Increasing or decreasing the Spin value will cause the Bounce region to impart a spin to the particle based on the angle of collision, or to modify any existing spin on the particle. Positive values will impart a forward spin, and negative values impart a backward spin. The larger the value, the faster the spin applied to the particle will be.

Roughness

This slider varies the bounce off the surface to slightly randomize particle direction.

Surface Motion

This slider makes the bounce surface behave as if it had motion, thus affecting the particles.

Surface Motion Direction

This thumbwheel control sets the angle relative to the bounce surface.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pChangeStyle [pCS]
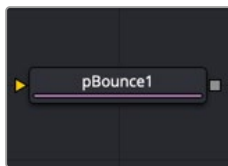
The pChange Style node

pChange Style Node Introduction

The pChange Style node provides a mechanism for changing the appearance or style of particles that interact with a defined region. The primary controls mirror those found in the Style tab of the pEmitter node. Particles that intersect or enter the defined region change based on the parameters of this node.

Except for the pCustom node, this is the only node that modifies the particles' appearance rather than its motion. It is often used to trigger a change in the appearance in response to some event, such as striking a barrier.

Inputs

The pChange Style node has a single orange input by default. Like most particle nodes, this orange input accepts only other particle nodes. A green or magenta bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Region:** The green or magenta region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where the custom particle node takes effect.

Basic Node Setup

Opposite of what you may think, to create a change in style that appears to be caused by some physical event, the pChange Style node should be placed before the node that creates the event. For example, below, the particles generated by the Emitter node change style after bouncing off a pBounce. Both the pChange Style and pBounce use the same Shape 3D node as the region.

The pChange Style must be placed before the pBounce. If the pChange Style node is placed after the pBounce, the particles bounce off the region before the pChange Style calculates its effect. The particle will never get to intersect with the pChange Style node's region, and so the style never changes.
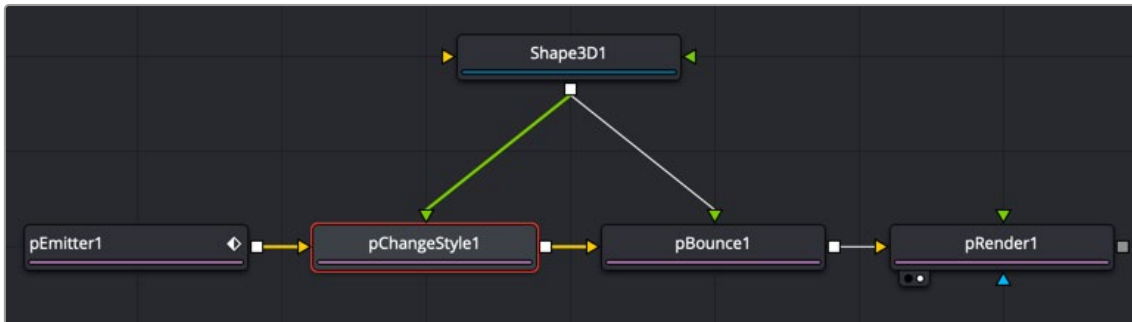
A pChange Style node placed before the pBounce node

Inspector
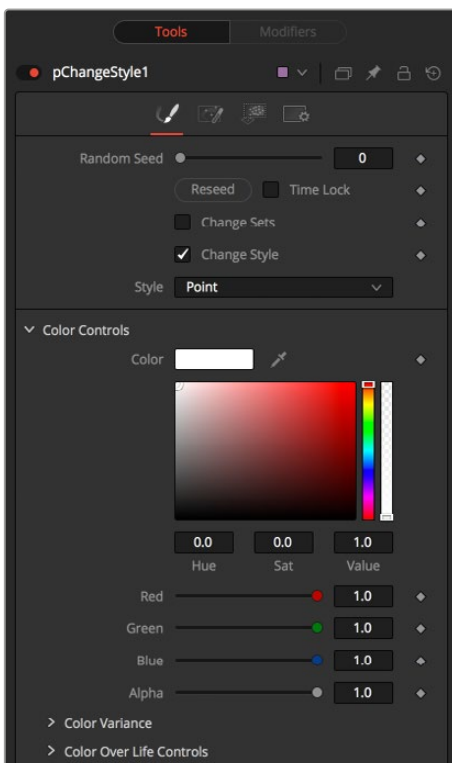
The pChange Style controls

Randomize

The Random Seed slider and Randomize button are presented whenever a Fusion node relies on a random result. Two nodes with the same seed values will produce the same random results. Click the Randomize button to randomly select a new seed value, or adjust the slider to manually select a new seed value.

Change Sets

This option allows the user to change the particle's Set to become influenced by forces other than the original particle. See "The Common Controls" in this chapter to learn more about Sets.

Style

This option allows the user to change the particle's Style and thus the look. See "The Common Controls" in this chapter to learn more about Styles.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pCustom [pCu]
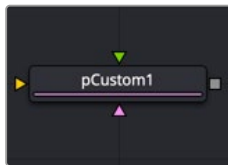
The pCustom node

pCustom Node Introduction

The pCustom node is used to create custom expressions that affect the properties of particles. This node is similar to the Custom node, except the calculations affect particles rather than pixels.

Inputs

The pCustom node has three inputs. Like most particle nodes, this orange input accepts only other particle nodes. The green and magenta inputs are 2D image inputs for custom image calculations. Optionally, there are teal or white bitmap or mesh inputs, which appear on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Image 1 and 2:** The green and magenta image inputs accept 2D images that are used for per pixel calculations and compositing functions.
— **Region:** The teal or white region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where the custom particle node takes effect.

Basic Node Setup

The pCustom node is placed in between the pEmitter and pRender. A Shape 3D node is used to create the region where the Custom particle event occurs.

A pCustom node using a Shape 3D node as the region where the custom event occurs

Inspector

All the same operators, functions, and conditional statements described for the Custom node apply to the pCustom node as well, including Pixel-read functions for the two image inputs (e.g., getr1w(x,y), getz2b(x,y), and so on).
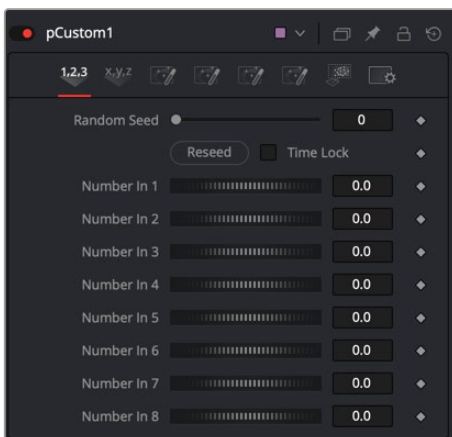

The pCustom controls

Number 1-8

Numbers are variables with a dial control that can be animated or connected to modifiers exactly as any other control might. The numbers can be used in equations on particles at current time: n1, n2, n3, n4, … or at any time: n1_at(float t), n2_at(float t), n3_at(float t), n4_at(float t), where t is the time you want. The values of these controls are available to expressions in the Setup and Intermediate tabs.

Position 1-8

These eight point controls include 3D X,Y,Z position controls. They are normal positional controls and can be animated or connected to modifiers as any other node might. They are available to expressions entered in the Setup, Intermediate, and Channels tabs.
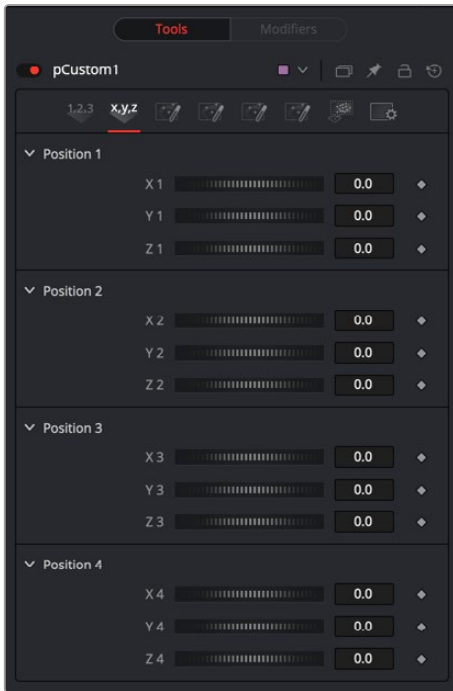
pCustom Position tab
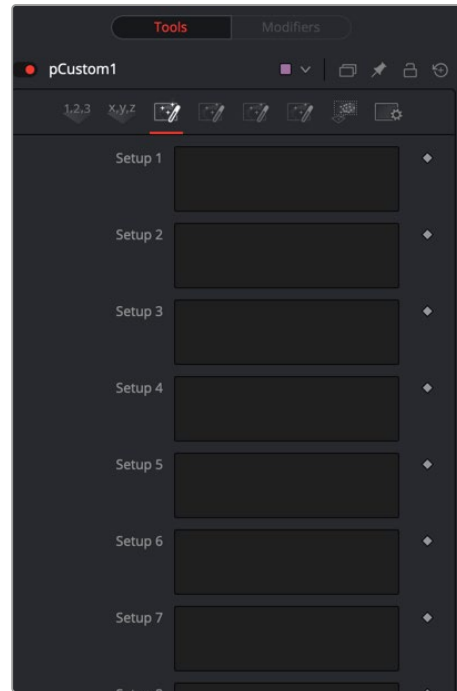
pCustom Setup tab

Setup 1-8

Up to eight separate expressions can be calculated in the Setup tab of the pCustom node. The Setup expressions are evaluated once per frame, before any other calculations are performed. The results are then made available to the other expressions in the node as variables s1, s2, s3, and s4.

Think of them as global setup scripts that can be referenced by the intermediate and channel scripts.
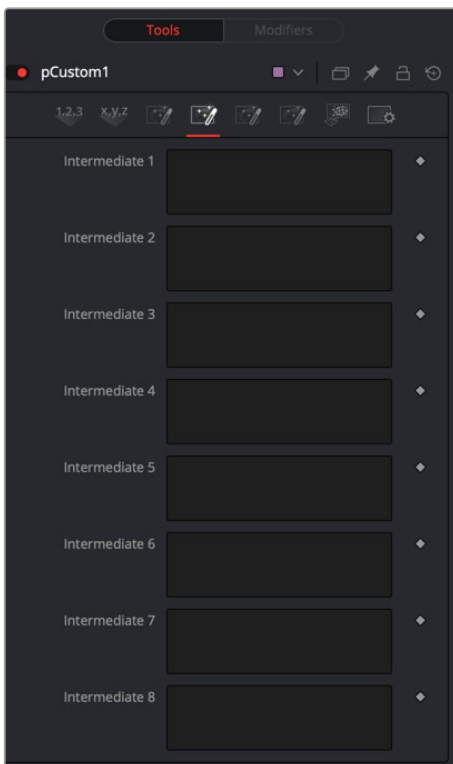
pCustom Intermediate tab

Inter 1-8

An additional eight expressions can be calculated in the Intermediate tab. The Intermediate expressions are evaluated once per frame, after the Setup expressions are evaluated. Results are available as variables i1, i2, i3, i4, i5, i6, i7, i8, which can be referenced by channel scripts.

Particle

Particle position, velocity, rotation, and other controls are available in the Particle tab.

The following particle properties are exposed to the pCustom control:

The following particle properties are exposed to the pCustom control:	
px, py, pz	particle position on the x, y, and z axis
vx, vy, vz	particle velocity on the x, y and z axis
rx, ry, rz	particle rotation on the x, y, and z axis
sx, sy, sz	particle spin on the x, y, and z axis
pxi1, pyi1	the 2d position of a particle, corrected for image 1's aspect
pxi2, pyi2	the 2d position of a particle, corrected for image 2's aspect
mass	not currently used by anything
size	the current size of a particle
id	the particle's identifier
r, g, b, a	the particles red, green, blue and alpha color values
rgnhit	this value is 1 if the particle hit the pCustom node's defined region
rgndist	this variable contains the particles distance from the region
condscale	the strength of the region at the particle's position
rgnix, rgniy, rgniz	values representing where on the region the particle hit
rgnnx, rgnny, rgnnz	region surface normal of the particle when it hit the region
w1, h1	image 1 width and height
w2 h2	image 2 width and height
i1, i2, i3, i4	the result of the intermediate calculations 1 through 4
s1, s2, s3, s4	the result of the setup calculations 1 through 4
n1..n8	the values of numeric inputs 1 through 8
p1x, p1y, p1z .. p4x, p4y, p4z	the values of position inputs 1 through 4
time	the current time or frame of the compositions
age	the current age of the particle
lifespan	the lifespan of the current particle

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pCustomForce [pCF]
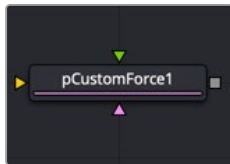
The pCustom Force node

pCustom Force Node Introduction

The pCustom Force node allows you to change the forces applied to a particle system or subset. This node is one of the most complex and the most powerful node in Fusion. If you are experienced with scripting or C++ programming, you should find the structure and terminology used by the Custom Force node to be familiar.

The forces on a particle within a system can have their positions and rotations affected by forces. The position in XYZ and the Torque, which is the spin of the particle, are controlled by independent custom equations. The Custom Force node is used to create custom expressions and filters to modify the behavior. In addition to providing three image inputs, this node will allow for the connection of up to eight numeric inputs and as many as four XY position values from other controls and parameters in the node tree.

Inputs

The pCustom Force node has three inputs. Like most particle nodes, this orange input accepts only other particle nodes. A green and magenta are 2D image inputs for custom image calculations. Optionally there are teal or white bitmap or mesh inputs, which appear on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.

— **Image 1 and 2:** The green and magenta image inputs accept 2D images that are used for per-pixel calculations and compositing functions.

— **Region:** The teal or white region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where the pCustom Force takes effect.

Basic Node Setup

The pCustom Force node is inserted between a pEmitter and pRender node to serve as a catalyst for particles using advanced C++ and scripting.
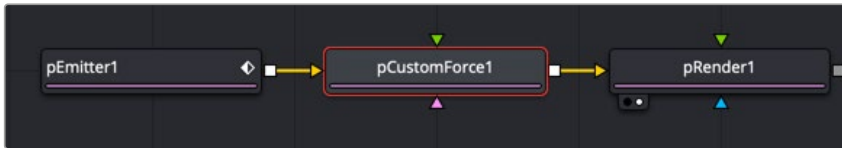
A pCustom Force is applied to the particles generated by the pEmitter.

Inspector
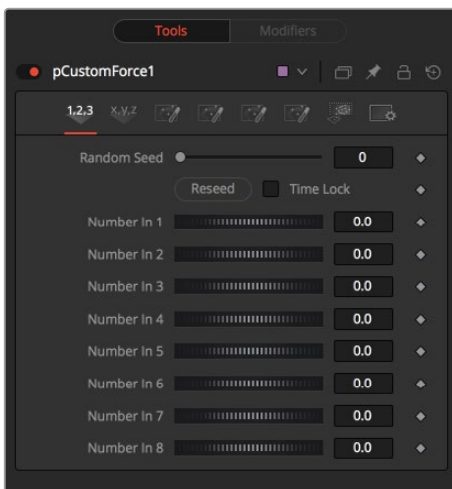
The pCustom Force controls

The tabs and controls located in the Inspector are similar to the controls found in the pCustom node. Refer to the pCustom node in this chapter for more information.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pDirectionalForce [pDF]

The pDirectional Force node

pDirectional Force Node Introduction

This node applies a unidirectional force that pulls the affected particles in a specified direction. Its primary controls affect the strength of the force, and the angle of the force's pull along the X, Y, and Z axis.

Since the most common use of this node is to simulate gravity, the default direction of the pull is down along the Y axis (-90 degrees), and the default behavior is to ignore regions and affect all particles.

Inputs

The pDirectional Force node has a single orange input by default. Like most particle nodes, this orange input accepts only other particle nodes. A green or magenta bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Region:** The green or magenta region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where the directional force takes effect.

Basic Node Setup

The pDirectional Force node is placed in between the pEmitter and pRender and is often used to create gravity.
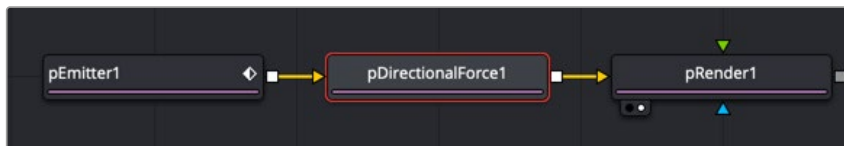
A pDirectional Force node placed between the pEmitter and pRender nodes

Inspector
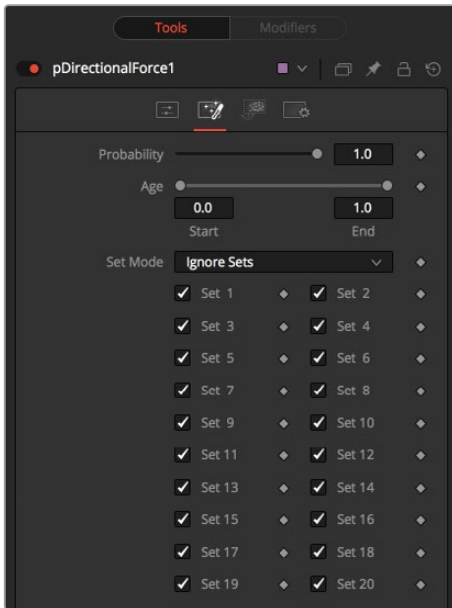
The pDirectional Force controls

Randomize

The Random Seed slider and Randomize button are presented whenever a Fusion node relies on a random result. Two nodes with the same seed values will produce the same random results. Click the Randomize button to select a new seed value randomly, or adjust the slider to select a new seed value manually.

Strength

Determines the power of the force. Positive values will move the particles in the direction set by the controls; negative values will move the particles in the opposite direction.

Direction

Determines the direction in X/Y space.

Direction Z

Determines the direction in Z space.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pEmitter [pEm]

The pEmitter node

pEmitter Node Introduction

The pEmitter node is the main source of particles (pImage Emitter is another) and will usually be the first node used in any new particle system. This node contains controls for setting the initial position, orientation, and motion of the particles, as well as controls for the visual style of each particle.

Like all other Particle nodes (with the exception of the pRender node), the pEmitter produces a particle set, not a visible image, and therefore cannot be displayed directly in a viewer. To view the output of a particle system, add a pRender node after the pEmitter.

Inputs

By default, the pEmitter node has no inputs at all. You can enable an image input by selecting Bitmap from the Style menu in the Style tab. Also, two region inputs, one for bitmap and one for mesh, appear on the node when you set the Region menu in the Region tab to either Bitmap or Mesh. The colors of these inputs change depending on the order in which they are enabled.

— **Style Bitmap Input:** This image input accepts a 2D image to use as the particles' image. Since this image duplicates into potentially thousands of particles, it is best to keep these images small and square—for instance, 256 x 256 pixels.

— **Region:** The region inputs takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where the particles are emitted.

Basic Node Setup

The pEmitter node starts the branch of a particle system that always ends with a pRender node. The pEmitter can feed directly into a pRender node to feed other particle nodes.
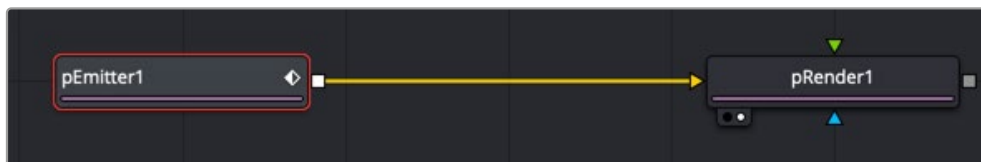
A pEmitter node connected to a pRender node is a typical setup for more particle systems.

Inspector

The pEmitter inspector is divided into four main tabs and a common settings tab. The controls tab is the first tab displayed and it contains settings that affect the general setup of the particle cells emitted by the node. These settings do not directly affect the appearance of the cells or shape of the emitter region. They modify fundamental behaviors like quantity, duration, speed, and rotation of the particle cells.

The pEmitter controls

Randomize and Random Seed

The Random Seed slider is used to seed all the variance and random number generators used by the node when creating the particle system. Two pEmitter nodes with exactly the same settings for all controls and the same random seed will generate exactly the same particle system. Changing the random seed will cause variation between the nodes. Click the Randomize button to automatically set a randomly chosen value for the Random Seed.

Number

This control is used to set the amount of new particles generated on each frame. A value of 1 would cause one new particle to be generated each frame. By frame 10, there would be a total of 10 particles in existence (unless Particle Lifespan was set to fewer than 10 frames).

Animate this parameter to specify the number of particles generated in total. For example, if only 25 particles in total are desired, animate the control to produce five particles on frame 0–4, then set a key on frame five to generate zero particles for the remainder of the project.

Number Variance

This modifies the amount of particles generated for each frame, as specified by the Number control. For example, if Number is set to 10.0 and Number Variance is set to 2.0, the emitter will produce anywhere from 9-11 particles per frame. If the value of Number Variance is more than twice as large as the value of Number, it is possible that no particles will be generated for a given frame.

Lifespan

This control determines how long a particle will exist before it disappears or 'dies.' The default value of this control is 100 frames, although this can be set to any value. The timing of many other particle controls is relative to the Lifespan of the particle. For example, the size of a particle can be set to increase over the last 80% of its life, using the Size Over Life graph in the Style tab of the pEmitter.

Lifespan Variance

Like Number Variance, the Lifespan Variance control allows the Lifespan of particles produced to be modified. If Particle Lifespan was set to 100 frames and the Lifespan Variance to 20 frames, particles generated by the emitter would have a lifespan of 90–110 frames.

Color

This provides the ability to specify from where the color of each particle is derived. The default setting is Use Style Color, which will provide the color from each particle according to the settings in the Style tab of the pEmitter node.

The alternate setting is Use Color From Region, which overrides the color settings from the Style tab and uses the color of the underlying bitmap region.

The Use Color From Region option only makes sense when the pEmitter region is set to use a bitmap produced by another node in the composition. Particles generated in a region other than a bitmap region will be rendered as white when the Use Color From Region option is selected.

Position Variance

This control determines whether or not particles can be 'born' outside the boundaries of the pEmitter region. By default, the value is set to zero, which will restrict the creation area for new particles to the exact boundaries of the defined region. Increasing this control's value above 0.0 will allow the particle to be born slightly outside the boundaries of that region. The higher the value, the 'softer' the region's edge will become.

Temporal Distribution

In general, an effect is processed per frame, based on the comp frame rate. However, processing some particles only at the exact frame boundaries can cause pulsing. To make the behavior subtly more realistic, the particles can be birthed in subframe increments.

The default, At The Same Time setting renders on frame boundaries, where as the other two settings take advantage of sub frame rendering. Randomly Distributed randomizes birth times +/- around the frame number, eg birth 10 particles at random sub times 24.1 24.85, 24.21, 24.37 etc. one particle at a time. Evenly Distributed births particles at regular sub times, eg 10 particles, birth 1 at at time at 24.0, 24.1, 24.2, 24.3, 24.4, 24.5 ... 24.8, 24.9.

These settings are influenced by the Sub Frame Accuracy setting in the pRender node. The Sub Frame Accuracy slider controls how many in-between frames are calculated between each frame. The higher the value the more accurate the particle calculation but the longer the render times.

Velocity

The controls in the Velocity section determine the speed and direction of the particle cells as the are generated from the emitter region.

Velocity and Velocity Variance

These determine the initial speed or velocity of new particles. By default, the particle has no velocity and will not move from its point of origin unless acted upon by outside forces. A velocity setting of 10.0 would cause the particle to cross the entire width of the image in one step so a velocity of 1.0 would cause the particle to cross the width of the image over 10 frames.

Velocity Variance modifies the velocity of each particle at birth, in the same manner described in Lifespan Variance and Number Variance above.

Inherit

Inherit Velocity passes the emitter region's velocity on to the particles. This slider has a wide range that includes negative and positive values. A negative value causes the particles to move in the opposite direction, a value of 1 will cause the particles to move with a velocity that matches the emitter region's velocity, and a value of 2 causes the particles to move ahead of the emitter region.

Angle and Angle Variance

This determines the angle at which particles with velocity applied will be heading at their birth.

Angle Z and Angle Z Variance

This is as above, except this control determines the angle of the particles along the Z space axis (toward or away from the camera).

Rotation

Rotation controls are used to set the orientation of particle cells and animating that orientation over time .

Rotation Mode

This menu control provides two options to help determine the orientation of the particles emitted. When the particles are spherical, the effect of this control will be unnoticeable.

— **Absolute Rotation:** The particles will be oriented as specified by the Rotation controls, regardless of velocity and heading.

— **Rotation Relative To Motion:** The particles will be oriented in the same direction as the particle is moving. The Rotation controls can now be used to rotate the particle's orientation away from its heading.

Rotation XYZ and Rotation XYZ Variance

These controls allow for Rotation of the individual particles. This can be particularly useful when dealing with a bitmap particle type, as the incoming bitmap may not be oriented in the desired direction.

Rotation XYZ Variance can be used to randomly vary the rotation by a specified amount around the center of the Rotation XYZ value to avoid having every particle oriented in the exact same direction.

Spin

Spin controls are auto animated controls that change the orientation of particle cells over time.

Spin XYZ and Spin Variance

These provide a spin to be applied to each particle at birth. The particles will rotate ‚x‘ degrees each frame, as determined by the value of Spin XYZ.

The Spin XYZ variances will vary the amount of rotation applied to each frame in the manner described by Number Variance and Lifespan Variance documented above.

Sets Tab

This tab contains settings that affect the physics of the particles emitted by the node. These settings do not directly affect the appearance of the particles. Instead, they modify behavior like velocity, spin, quantity, and lifespan.
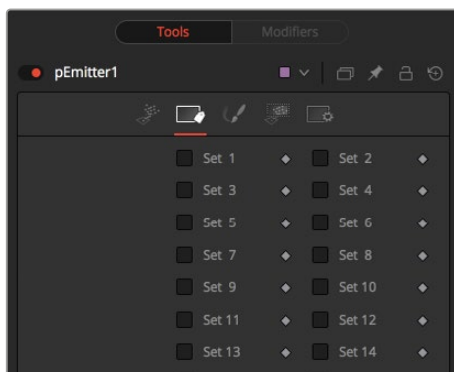
The pEmitter Sets tab

Set 1-32

To assign the particles created by a pEmitter to a given set, simply select the checkbox of the set number you want to assign. A single pEmitter node can be assigned to one or multiple sets. Once they are assigned in the pEmitter, you can enable sets in other particle nodes so they only affect particles from specific pEmitters.

Style Tab

The Style tab provides controls that affect the appearance of the particles. For detailed information about the style Tab, see the "The Common Controls" section at the end of this chapter.
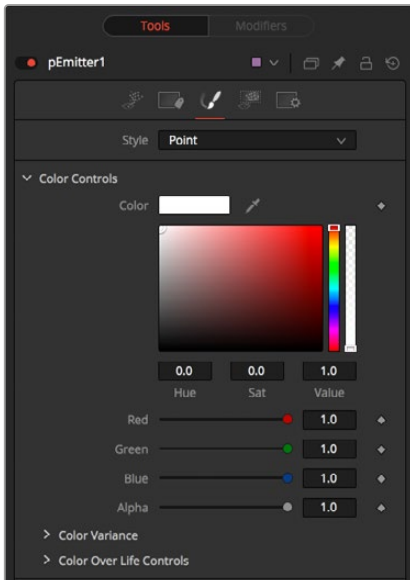
The pEmitter Style tab

Region Tab

The Region tab controls the shape, size, and location of the area that emits the particle cells. This is often called the Emitter. Only one emitter region can be set for a single pEmitter node. If the pRender is set to 2D, then the emitter region will produce particles along a flat plane in Z Space. 3D emitter regions possess depth and can produce particles inside a user-defined, three-dimensional region. For more detail on the Region tab, see "The Common Controls" section at the end of this chapter.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pFlock [pFl]
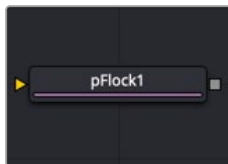
The pFlock node

pFlock Node Introduction

The pFlock node can be used to simulate the behavior of organic systems, such as a flock of birds or a colony of ants. Its use can make an otherwise mindless particle system appear to be motivated, or acting under the direction of intelligence.

The pFlock node works through two basic principles. Each particle attempts to stay close to other particles and each particle attempts to maintain a minimum distance from other particles.

The strength of these "desires" produces the seemingly motivated behavior perceived by the viewer.

Inputs

The pFlock node has a single orange input by default. Like most particle nodes, this orange input accepts only other particle nodes. A green or magenta bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange background input takes the output of other particle nodes.
— **Region:** The green or magenta region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where the flocking takes effect.

Basic Node Setup

When combined with pFollow, the pFlock node can produce natural swarming behaviors that change direction.

A pFlock node applying more herd-type mentality to particles

Inspector
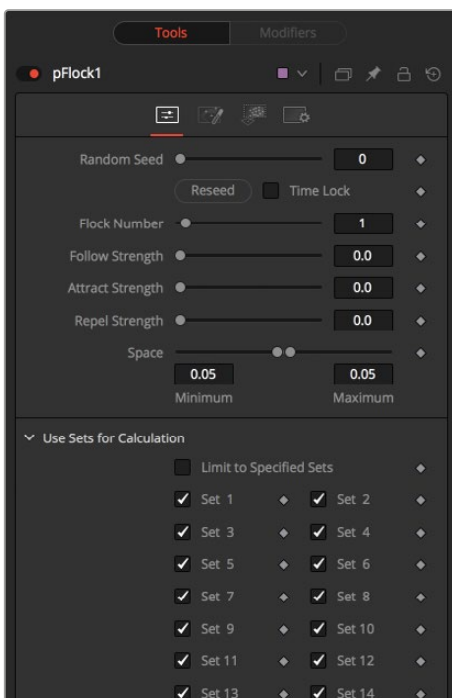
The pFlock controls

Randomize

The Random Seed slider and Randomize button are presented whenever a Fusion node relies on a random result. Two nodes with the same seed values will produce the same random results. Click the Randomize button to randomly select a new seed value, or adjust the slider to manually select a new seed value.

Flock Number

The value of this control represents the number of other particles that the affected particle will attempt to follow. The higher the value, the more visible "clumping" will appear in the particle system and the larger the groups of particles will appear.

Follow Strength

This value represents the strength of each particle's desire to follow other particles. Higher values will cause the particle to appear to expend more energy and effort to follow other particles. Lower values increase the likelihood that a given particle will break away from the pack.

Attract Strength

This value represents the strength of attraction between particles. When a particle moves farther from other particles than the Maximum Space defined in the pFlock node, it will attempt to move closer to other particles. Higher values cause the particle to maintain its spacing energetically, resolving conflicts in spacing more rapidly.

Repel Strength

This value represents the force applied to particles that get closer together than the distance defined by the Minimum Space control of the pFlock node. Higher values will cause particles to move away from neighboring particles more rapidly, shooting away from the pack.

Minimum/Maximum Space

This range control represents the distance each particle attempts to maintain between it and other particles. Particles will attempt to get no closer or farther than the space defined by the Minimum/ Maximum values of this range control. Smaller ranges will give the appearance of more organized motion. Larger ranges will be perceived as disorganized and chaotic.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pFollow [pFo]

The pFollow node

pFollow Node Introduction

Inserting the pFollow node into a particle branch causes the particles to spring back and forth toward a follow object. The follow object can be positioned in 3D or animated to create a new motion path for the particles.

Inputs

The pFollow node has a single orange input by default. Like most particle nodes, this orange background input accepts only other particle nodes. A green bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Region:** The green or magenta region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where particles will follow the position point.

Basic Node Setup

When combined with pFlock, the pFollow node can produce natural swarming behaviors that change direction.
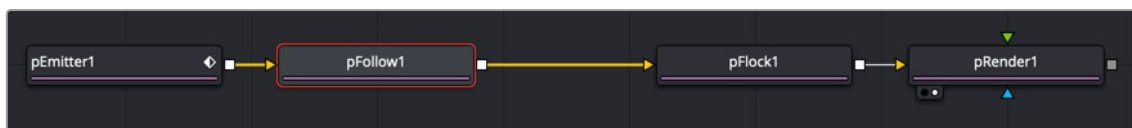
A pFollow node introduces a follow object that influences the particles' motion.

Inspector
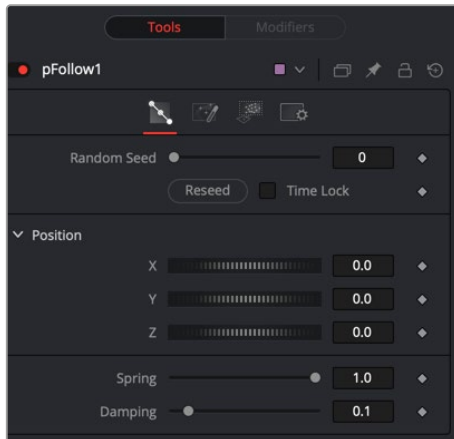
The pFollow Controls tab

Random Seed

The Random Seed slider and Randomize button are presented whenever a Fusion node relies on a random result. Two nodes with the same seed values will produce the same random results. Click the Randomize button to randomly select a new seed value, or adjust the slider to manually select a new seed value.

Position XYZ

The position controls are used to create the new path by positioning the follow object. Moving the XYZ parameters displays the onscreen position of the follow object. Animating these parameters creates the new path the particles will be influenced by.

Spring

The Spring setting causes the particles to move back and forth along the path. The spread of the spring motion increases over the life of the particles depending on the distance between the particles and the follow object. Higher spring settings increase the elasticity, while lower settings decrease elasticity.

Dampen

This value attenuates the spring action. A lower setting offers less resistance to the back and forth spring action. A higher setting applies more resistance.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pFriction [pFr]

The pFriction node

pFriction Node Introduction

The pFriction node applies resistance to the motion of a particle, slowing the particle's motion through a defined region. This node produces two types of Friction. One type reduces the velocity of any particle intersecting/crossing the defined region, and one reduces or eliminates spin and rotation.

Inputs

The pFriction node has a single orange input by default. Like most particle nodes, this orange input accepts only other particle nodes. A green or magenta bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Region:** The green or magenta region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where the friction occurs.

Basic Node Setup

The pFriction node is placed in between the pEmitter and pRender. A Shape 3D node is used to create the region the particles bounce off.
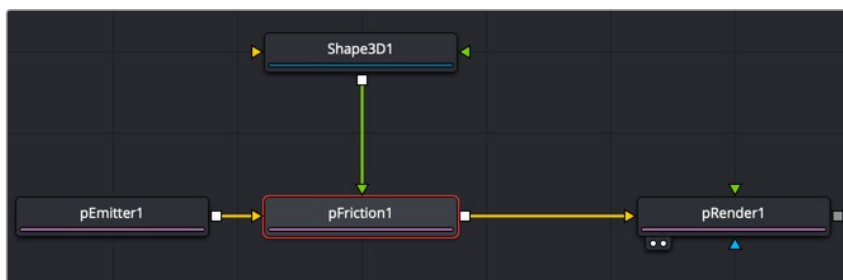
A pFriction node using a Shape 3D node as the
region where friction is introduced to the particles.

Inspector
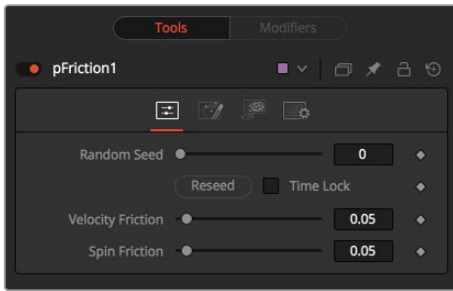
The pFriction controls

Random Seed

The Random Seed slider and Randomize button are presented whenever a Fusion node relies on a random result. Two nodes with the same seed values will produce the same random results. Click the Randomize button to randomly select a new seed value, or adjust the slider to manually select a new seed value.

Velocity Friction

This value represents the Friction force applied to the particle's Velocity. The larger the value, the greater the friction, thus slowing down the particle.

Spin Friction

This value represents the Friction force applied to the particle's Rotation or Spin. The larger the value, the greater the friction, thus slowing down the rotation of the particle.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pGradientForce [pGF]
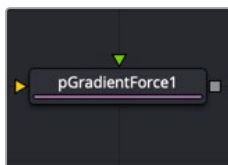
The pGradient Force node

pGradient Force Node Introduction

The particles are affected by a force generated by the gradients in the alpha values of the input image. Particles will accelerate along the gradient, moving from white to black (high values to low values).

This node can be used to give particles the appearance of moving downhill or following the contour of a provided shape.

Inputs

The pGradient Force node accepts two inputs: the default orange input from a particle node and one from a bitmap image with an alpha channel gradient. A magenta or teal bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Input:** The green input takes the 2D image that contains the alpha channel gradient.
— **Region:** The magenta or teal region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where the gradient force occurs.

Basic Node Setup

The pGradient Force node is placed in between the pEmitter and pRender nodes. A Fast Noise node is used to create the alpha gradient used to modify the velocity of the particles.
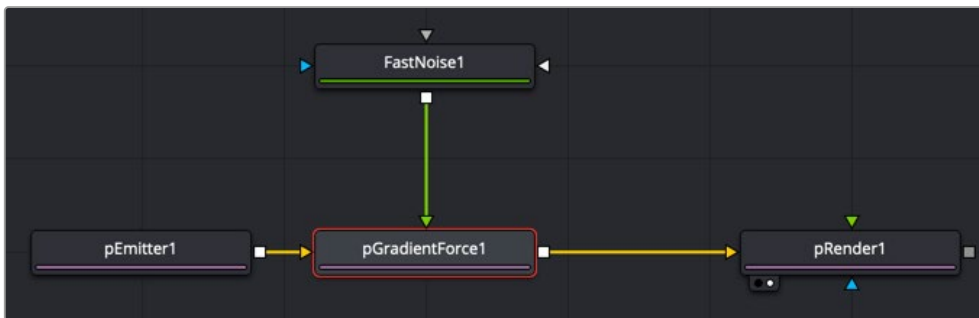
A pGradient Force node using a Fast Noise node as the gradient to modify the particles' motion

Inspector
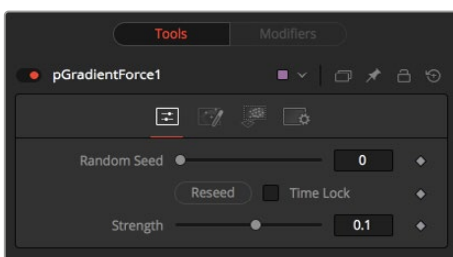
The pGradient Force controls

Randomize

The Random Seed slider and Randomize button are presented whenever a Fusion node relies on a random result.

Two nodes with the same seed values will produce the same random results. Click the Randomize button to randomly select a new seed value, or adjust the slider to manually select a new seed value.

Strength

Gradient Force has only one specific control, which affects the strength of the force and acceleration applied to the particles. Negative values on this control will cause the Gradient Force to be applied from black to white (low values to high values).

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pImage Emitter [pIE]

The pImage Emitter node

pImage Emitter Node Introduction

The pImage Emitter node takes an input image and treats each pixel of the image as if it were a particle. The main differences between the pImage Emitter and the normal pEmitter is that instead of emitting particles randomly within a given region, this node emits pixels in a regular 2D grid with colors based on the input image.

Inputs

The pImage Emitter node has three inputs. Like most particle nodes, the orange input accepts only other particle nodes. Green and magenta inputs are 2D image inputs for custom image calculations. Optionally, there are teal or white bitmap or mesh inputs, which appear on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** Unlike most other particle nodes, the orange input on the pImage Emitter accepts a 2D image used as the emitter of the particles. If a region is defined for the emitter, this input is used to define the color of the particles.

— **Style Bitmap Input:** This image input accepts a 2D image to use as the particles' image. Since this image duplicates into potentially thousands of particles, it is best to keep these images small and square—for instance, 256 x 256 pixels.

— **Region:** The teal or white region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where the particles are emitted.
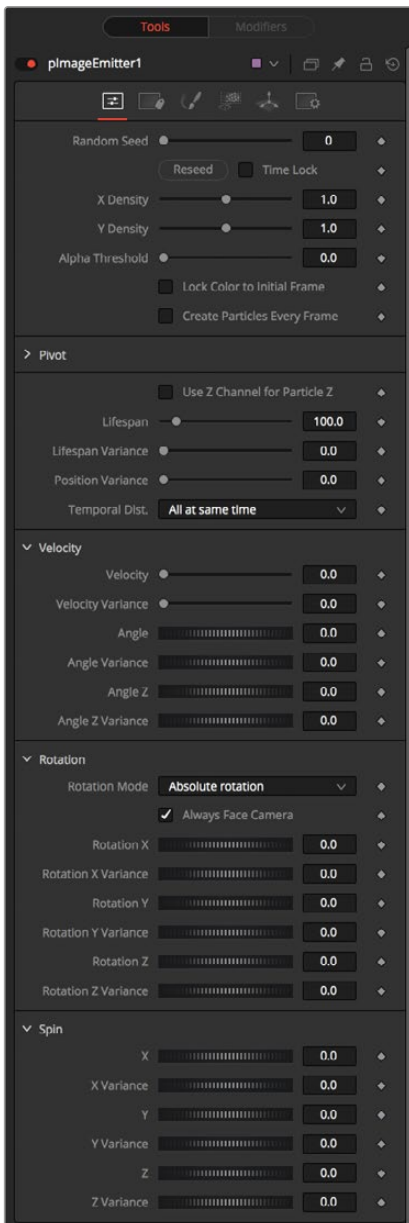
Basic Node Setup

The pImage Emitter node is placed at the start of a particle branch, replacing the location of a pEmitter node. Below, a MediaIn node is used to emit particles using the colors from the clip.
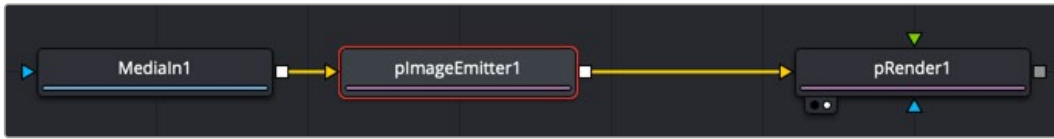
A pImage Emitter node emits particles based on an image connected to the orange input.

Inspector

The pImage Emitter controls

The great majority of controls in this node are identical to those found in the pEmitter, and those controls are documented in that previous section. Below are the descriptions of the controls unique to the pImage Emitter node.

X and Y Density

The X and Y Density sliders are used to set the mapping of particles to pixels for each axis. They control the density of the sampling grid. A value of 1.0 for either slider indicates 1 sample per pixel. Smaller values will produce a looser, more pointillistic distribution of particles, while values above 1.0 will create multiple particles per pixel in the image.

Alpha Threshold

The Alpha Threshold is used for limiting particle generation so that pixels with semitransparent alpha values will not produce particles. This can be used to harden the edges of an otherwise soft alpha channel. The higher the threshold value, the more opaque a pixel must be before it will generate a particle. Note that the default threshold of 0.0 will create particles for every pixel, regardless of alpha, although many may be transparent and invisible.

Lock Particle Color to Initial Frame

Select this checkbox to force the particles to keep the color with which they were born throughout the life of the particle. If this is off, and the input image changes on successive frames, the particles will also change color to match the image. This allows video playback on a grid of particles.

Create Particles Every Frame

Enabling this creates a whole new set of particles every frame, instead of just one set on the frame. This can lead to very large particle systems but allows some interesting effects—for example, if the particles are given some initial velocity or if emitting from an animated source. Try a small velocity, an Angle Z of -90, and a seething Fast Noise as a source to get smoothly varying clouds of particles that you could fly through. Note that if this checkbox is left off, only one set of particles is ever created, and thus animating any of the emitter's other controls will have no effect.

X/Y/Z Pivot

These controls allow you to position the grid of emitted particles.

Use Z Channel for Particle Z

If the input image used to generate the particles has a Z depth channel, that channel can be used to determine the initial position of the particle in Z space. This can have an interesting hollow shell effect when used in conjunction with camera rotation in the pRender node.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

Sets Tab
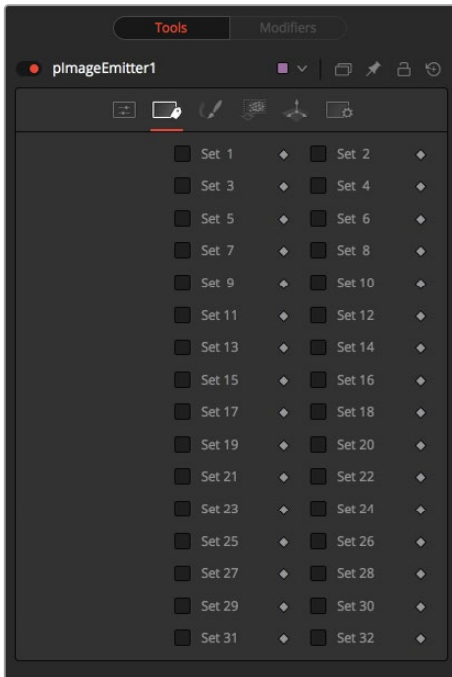
The pImage Emitter Sets tab

NOTE: Pixels with a black (transparent) alpha channel will still generate invisible particles, unless you raise the Alpha Threshold above 0.0. This can slow down rendering significantly.

An Alpha Threshold value of 1/255 = 0.004 is good for eliminating all fully transparent pixels.

The pixels are emitted in a fixed-size 2D grid on the XY plane, centered on the Pivot position. Changing the Region from the default of All allows you to restrict particle creation to more limited areas. If you need to change the size of this grid, use a Transform 3D node after the pRender.

Remember that the various emitter controls apply only to particles when they are emitted. That is, they set the initial state of the particle and do not affect it for the rest of its lifespan. Since pImageEmitter (by default) emits particles only on the first frame, animating these controls will have no effect. However, if the Create Particles Every Frame checkbox is turned on, new particles will be emitted each frame and will use the specified initial settings for that frame.

pKill [pKl]

The pKill node

pKill Node Introduction

The Kill node is used to destroy (kill) any particle that crosses or intersects its region. It has no specific controls, as it has only one possible affect on a particle. The controls found in the Region tab are normally used to limit this node by restricting the effect to particles that fall within a certain region, age, set, or by reducing the probability of the node applying to a given particle.

Inputs

The pKill node has a single orange input by default. Like most particle nodes, this orange input accepts only other particle nodes. A green bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Region:** The green or magenta region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area particles are killed.

Basic Node Setup

The pKill node is placed in between the pEmitter and pRender nodes. A Shape 3D node is used to create the region where the particles die.
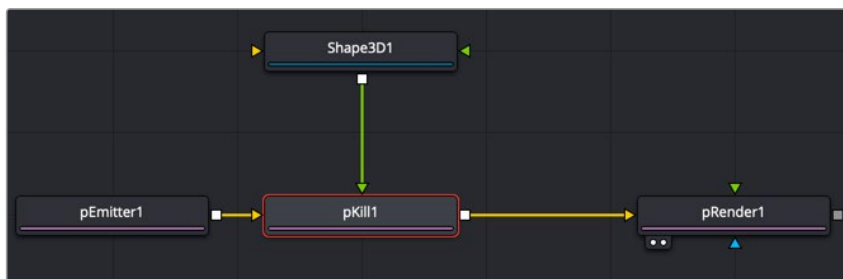
A pKill node using a Shape 3D node as the region where particles die

Inspector

This node only contains common controls in the Conditions and Regions tabs. The Conditions and Regions controls are used to define the location, age, and set of particles that are killed.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pMerge [pMg]
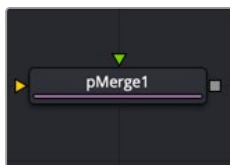
The pMerge node

pMerge Node Introduction

This node has no controls whatsoever. It serves to combine particles from two streams. Any nodes downstream of the pMerge node will treat the two streams as one.

The combined particles will preserve any sets assigned to them when they were created, making it possible for nodes downstream of the pMerge to isolate specific particles when necessary.

Inputs

The pMerge node has two identical inputs, one orange and one green. These two inputs accept only other particle nodes.

— **Particle 1 and 2 Input:** The two inputs accept two streams of particles and merge them.

Basic Node Setup

The pMerge node connects two pEmitter nodes. The output of the pMerge can go on to feed other particle nodes or to a pRender.
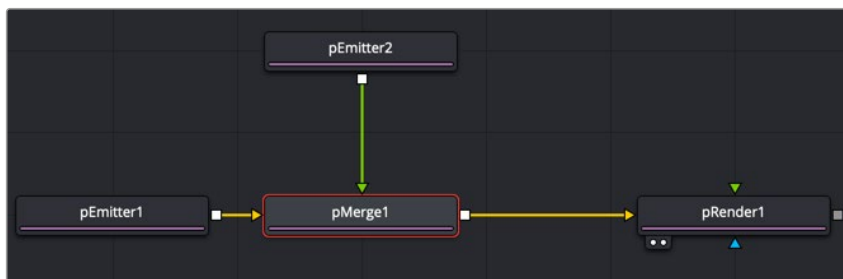
A pMerge node combining two pEmitter nodes.

pPoint Force [pPF]

The pPoint Force node

pPoint Force Node Introduction

This node applies a force to the particles that emanate from a single point in 3D space. The pPoint Force can either attract or repel particles within its sphere of influence. There are four controls specific to the pPoint Force node.

Inputs

The pPoint Force node has a single orange input by default. Like most particle nodes, this orange input accepts only other particle nodes. A green bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Region:** The green or magenta region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where the point force affects the particles.

Basic Node Setup

The pPoint Force node is inserted between a pEmitter and a pRender node.

The pPoint Force node positions a tangent force that particles are attracted to or repelled from.

Inspector
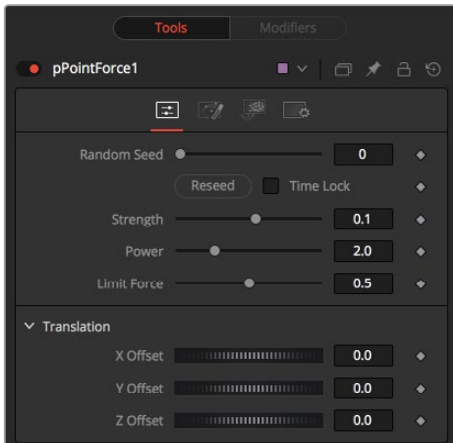
The pPoint Force controls

Randomize

The Random Seed slider and Randomize button are presented whenever a Fusion node relies on a random result. Two nodes with the same seed values will produce the same random results. Click the Randomize button to randomly select a new seed value, or adjust the slider to manually select a new seed value.

Strength

This parameter sets the Strength of the force emitted by the node. Positive values represent attractive forces; negative values represent repellent forces.

Power

This determines the degree to which the Strength of the force falls off over distance. A value of zero causes no falloff of strength. Higher values will impose an ever-sharper falloff in strength of the force with distance.

Limit Force

The Limit Force control is used to counterbalance potential problems with temporal sub-sampling. Because the position of a particle is sampled only once a frame (unless sub-sampling is increased in the pRender node), it is possible that a particle can overshoot the Point Force's position and end up getting thrown off in the opposite direction. Increasing the value of this control reduces the likelihood that this will happen.

X, Y, Z Center Position

These controls are used to represent the X, Y, and Z coordinates of the point force in 3D space.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pRender [pRn]

The pRender node

pRender Node Introduction

The pRender node converts the particle system to either an image or geometry. The default is a 3D particle system, which must be connected to a Renderer 3D to produce an image. This allows the particles to be integrated with other elements in a 3D scene before they are rendered.

Inputs

The pRender node has one orange input, a green camera input, and a blue effects mask input. Like most particle nodes, this orange input accepts only other particle nodes. A green bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Camera Input:** The optional green camera input accepts a camera node directly or a 3D scene with a camera connected that is used to frame the particles during rendering.
— **Effect Mask:** The optional blue input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input for 2D particles crops the output of the particles so they are seen only within the mask.

Basic Node Setup

The pRender node is always placed at the end of a particle branch. If the pRender is set to 2D, then the output connects to other 2D nodes like a Merge node. If the pRender is set to 3D, the output connects to a 3D node like a Merge 3D.
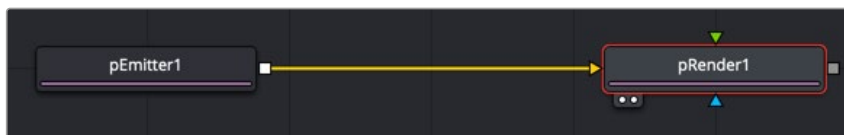
All particle branches end with a pRender node.

Inspector
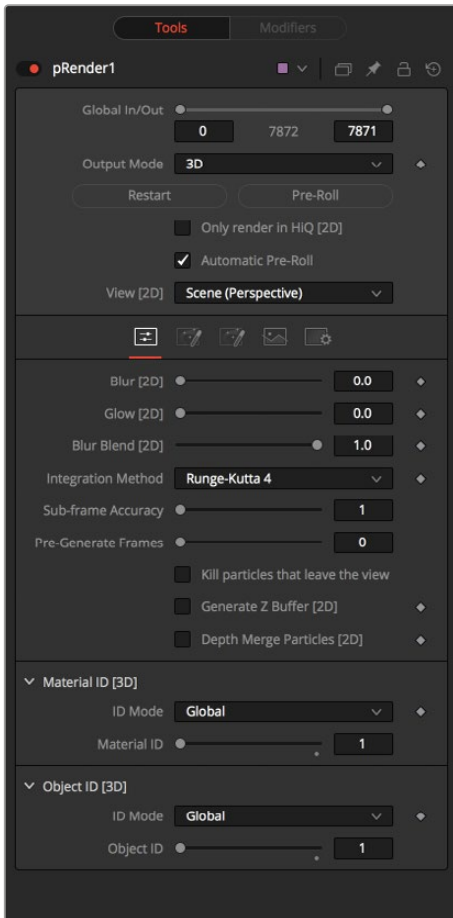
The pRender controls

Output Mode (2D/3D)

While the pRender defaults to 3D output, it can be made to render a 2D image instead. This is done with the 3D and 2D buttons on the Output Mode control. If the pRender is not connected to a 3D-only or 2D-only node, you can also switch it by selecting View > 2D Viewer from the viewer's pop-up menu.

In 3D mode, the only controls in the pRender node that have any effect at all are Restart, Pre-Roll and Automatic Pre-Roll, Sub-Frame Calculation Accuracy, and Pre-Generate frames. The remaining controls affect 2D particle renders only. The pRender node also has a Camera input on the node tree that allows the connection of a Camera 3D node. This can be used in both 2D and 3D modes to allow control of the viewpoint used to render an output image.

Render and the Viewers

When the pRender node is selected in a node tree, all the onscreen controls from Particle nodes connected to it are presented in the viewers. This provides a fast, easy-to-modify overview of the forces applied to the particle system as a whole.

Pre-Roll Options

Particle nodes generally need to know the position of each particle on the last frame before they can calculate the effect of the forces applied to them on the current frame. This makes changing current time manually by anything but single frame intervals likely to produce an inaccurate image.

The controls here are used to help accommodate this by providing methods of calculating the intervening frames.

Restart

This control also works in 3D. Clicking on the Restart button will restart the particle system at the current frame, removing any particles created up to that point and starting the particle system from scratch at the current frame.

Pre-Roll

This control also works in 3D. Clicking on this button causes the particle system to recalculate, starting from the beginning of the render range up to the current frame. It does not render the image produced. It only calculates the position of each particle. This provides a relatively quick mechanism to ensure that the particles displayed in the views are correctly positioned.

If the pRender node is displayed when the Pre-Roll button is selected, the progress of the pre-roll is shown in the viewer, with each particle shown as point style only.

Automatic Pre-Roll

Selecting the Automatic Pre-Roll checkbox causes the particle system to automatically pre-roll the particles to the current frame whenever the current frame changes. This prevents the need to manually select the Pre-Roll button whenever advancing through time in jumps larger than a single frame. The progress of the particle system during an Automatic Pre-Roll is not displayed to the viewers to prevent distracting visual disruptions.

About Pre-Roll

Pre-Roll is necessary because the state of a particle system is entirely dependent on the last known position of the particles. If the current time were changed to a frame where the last frame particle state is unknown, the display of the particle is calculated on the last known position, producing inaccurate results.

To demonstrate:

1   Add a pEmitter and a pRender node to the composition.
2   View the pRender in one of the viewers.
3   Set the Velocity of the particles to 0.1.
4   Place the pEmitter on the left edge of the screen.
5   Set the Current Frame to 0.
6   Set a Render Range from 0–100 and press the Play button.
7   Observe how the particle system behaves.
8   Stop the playback and return the current time to frame 0.
9   In the pRender node, disable the Automatic Pre-Roll option.
10 Use the current time number field to jump to frame 10, and then to frames 60 and 90.

Notice how the particle system only adds to the particles it has already created and does not try to create the particles that would have been emitted in the intervening frames. Try selecting the Pre-Roll button in the pRender node. Now the particle system state is represented correctly.

For simple, fast-rendering particle systems, it is recommended to leave the Automatic Pre-Roll option on. For slower particle systems with long time ranges, it may be desirable to only Pre-Roll manually, as required.

— **Only Render in Hi-Q**

Selecting this checkbox causes the style of the particles to be overridden when the Hi-Q checkbox is deselected, producing only fast rendering Point-style particles. This is useful when working with a large quantity of slow Image-based or Blob-style particles. To see the particles as they would appear in a final render, simply enable the Hi-Q checkbox.

— **View**

This drop-down list provides options to determine the position of the camera view in a 3D particle system. The default option of Scene (Perspective) will render the particle system from the perspective of a virtual camera, the position of which can be modified using the controls in the Scene tab. The other options provide orthographic views of the front, top, and side of the particle system.

It is important to realize that the position of the onscreen controls for Particle nodes is unaffected by this control. In 2D mode, the onscreen controls are always drawn as if the viewer were showing the front orthographic view. (3D mode gets the position of controls right at all times.)

The View setting is ignored if a Camera 3D node is connected to the pRender node's Camera input on the node tree, or if the pRender is in 3D mode.

Conditions

Blur, Glow, and Blur Blend

When generating 2D particles, these sliders apply a Gaussian blur, glows, and blur blending to the image as it is rendered, which can be used to soften the particles and blend them. The result is no different than adding a Blur after the pRender node in the node tree.

Sub-Frame Calculation Accuracy

This determines the number of sub-samples taken between frames when calculating the particle system. Higher values increase the accuracy of the calculation but also increase the amount of time to render the particle system.

Pre-Generate Frames

This control is used to cause the particle system to pre-generate a set number of frames before its first valid frame. This is used to give a particle system an initial state from which to start.

A good example of when this might be useful is in a shot where particles are used to create the smoke rising from a chimney. Set Pre-Generate Frames to a number high enough to ensure that the smoke is already present in the scene before the render begins, rather than having it just starting to emerge from the emitter for the first few frames.

Kill Particles That Leave the View

Selecting this checkbox control automatically destroys any particles that leave the visible boundaries of the image. This can help to speed render times. Particles destroyed in this fashion never return, regardless of any external forces acting upon them.

Generate Z Buffer

Selecting this checkbox causes the pRender node to produce a Z Buffer channel in the image. The depth of each particle is represented in the Z Buffer. This channel can then be used for additional depth operations like Depth Blur, Depth Fog, and Downstream Z Merging.

Enabling this option is likely to increase the render times for the particle system dramatically.

Depth Merge Particles

Enabling this option causes the particles to be merged using Depth Merge techniques, rather than layer-based techniques.

Scene Tab
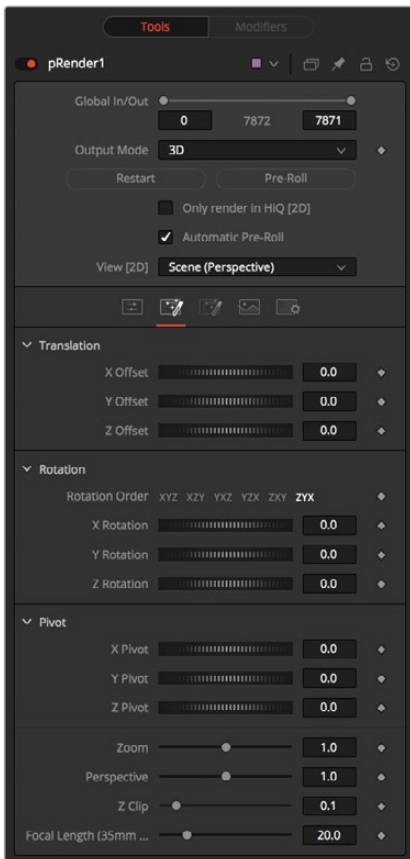
The pRender Scene tab

Z Clip

The Z Clip control is used to set a clipping plane in front of the camera. Particles that cross this plane are clipped, preventing them from impacting on the virtual lens of the camera and dominating the scene.

Grid Tab

These controls do not apply to 3D particles.

The grid is a helpful, non-rendering display guide used to orient the 2D particles in 3D space. The grid is never seen in renders, just like a center crosshair is never seen in a render. The width, depth, number of lines, and grid color can be set using the controls found in this tab.

These controls cannot be animated.

The pRender Grid tab

Image Tab

The controls in this tab are used to set the resolution, color depth, and pixel aspect of the rendered image produced by the node.
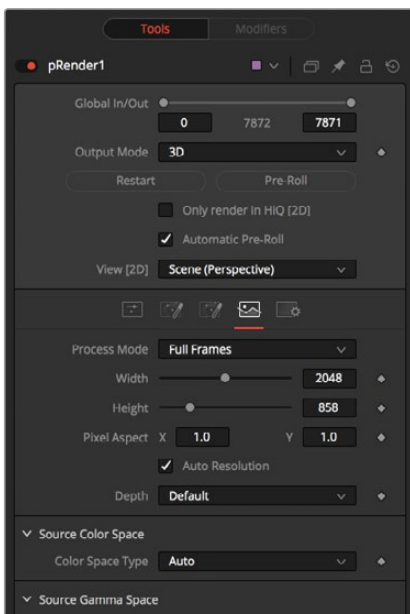
The pRender Image tab

Process Mode

Use this menu control to select the Fields Processing mode used by Fusion to render changes to the image. The default option is determined by the Has Fields checkbox control in the Frame Format preferences.

Use Frame Format Settings

When this checkbox is selected, the width, height, and pixel aspect of the rendered images by the node will be locked to values defined in the composition's Frame Format preferences. If the Frame Format preferences change, the resolution of the image produced by the node will change to match. Disabling this option can be useful to build a composition at a different resolution than the eventual target resolution for the final render.

Width/Height

This pair of controls is used to set the Width and Height dimensions of the image to be rendered by the node.

Pixel Aspect

This control is used to specify the Pixel Aspect ratio of the rendered particles. An aspect ratio of 1:1 would generate a square pixel with the same dimensions on either side (like a computer display monitor), and an aspect of 0.9:1 would create a slightly rectangular pixel (like an NTSC monitor).

> **NOTE:** Right-click on the Width, Height, or Pixel Aspect controls to display a menu listing the file formats defined in the preferences Frame Format tab. Selecting any of the listed options will set the width, height, and pixel aspect to the values for that format, accordingly.

Depth

The Depth menu is used to set the pixel color depth of the particles. 32-bit pixels require 4X the memory of 8-bit pixels but have far greater color accuracy. Float pixels allow high dynamic range values outside the normal 0...1 range, for representing colors that are brighter than white or darker than black.

Source Color Space

You can use the Source Color Space menu to set the Color Space of the footage to help achieve a linear workflow. Unlike the Gamut tool, this doesn't perform any actual color space conversion, but rather adds the source space data into the metadata, if that metadata doesn't exist. The metadata can then be used downstream by a Gamut tool with the From Image option, or in a Saver, if explicit output spaces are defined there. There are two options to choose from:

— **Auto:** Automatically reads and passes on the metadata that may be in the image.
— **Space:** Displays a Color Space Type menu where you can choose the correct color space of the image.

Source Gamma Space

Using the Curve type menu, you can set the Gamma Space of the footage and choose to remove it by way of the Remove Curve checkbox when working in a linear workflow. There are three choices in the Curve type menu:

— **Auto:** Automatically reads and passes on the metadata that may be in the image.

— **Space:** Displays a Gamma Space Type menu where you can choose the correct gamma curve of the image.

— **Log:** Brings up the Log/Lin settings, similar to the Cineon tool. For more information, *see Chapter 98, "Film Nodes."* in the DaVinci Resolve Reference Manual, or Chapter 37 in the Fusion Reference Manual.

Remove Curve

Depending on the selected Gamma Space or on the Gamma Space found in Auto mode, the Gamma Curve is removed from, or a log-lin conversion is performed on, the material, effectively converting it to a linear output space.

Motion Blur

As with other 2D nodes in Fusion, Motion Blur is enabled from within the Settings tab. You may set Quality, Shutter Angle, Sample Center, and Bias, and Blur will be applied to all moving particles.

> **NOTE:** Motion Blur on 3D mode particles (rendered with a Renderer 3D) also requires that identical motion blur settings are applied to the Renderer 3D node.

pSpawn [pSp]

The pSpawn node

pSpawn Node Introduction

The pSpawn node makes each affected particle act as an emitter that can produce one or more particles of its own. The original particle continues until the end of its lifespan, and each of the particles it emits becomes wholly independent with a lifespan and properties of its own.

As long as a particle falls under the effect of the pSpawn node, it will continue to generate particles. It is important to restrict the effect of the node with limiters like Start and End Age, Probability, Sets and Regions, and by animating the parameters of the emitter so that the node is operative only when required.

Inputs

By default, the pSpawn node has a single orange input. Like most particle nodes, this orange input accepts only other particle nodes. You can enable an image input by selecting Bitmap from the Style menu in the Style tab. Also, two region inputs, one for bitmap and one for mesh, appear on the node

when you set the Region menu in the Region tab to either Bitmap or Mesh. The colors of these inputs change depending on the order they are enabled.

— **Input:** The orange input accepts the output of other particle nodes.
— **Style Bitmap Input:** This image input accepts a 2D image to use as the particles' image. Since this image duplicates into potentially thousands of particles, it is best to keep these images small and square—for instance, 256 x 256 pixels.
— **Region:** The region inputs take a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where the particles are emitted.

Basic Node Setup

The pSpawn node is placed between the pEmitter and pRender nodes. Using the Age parameter in the pSpawn's Conditions tab, you can spawn new particles as the old ones die off. This is one way to have a trail of particles shoot up in the air like a rocket and burst into sparkling fireworks.

A pSpawn node used to generate new particles at specific points in the old particles' life

Inspector

The pSpawn node has a large number of controls, most of which exactly duplicate those found within the pEmitter node. There are a few controls that are unique to the pSpawn node, and their effects are described below.
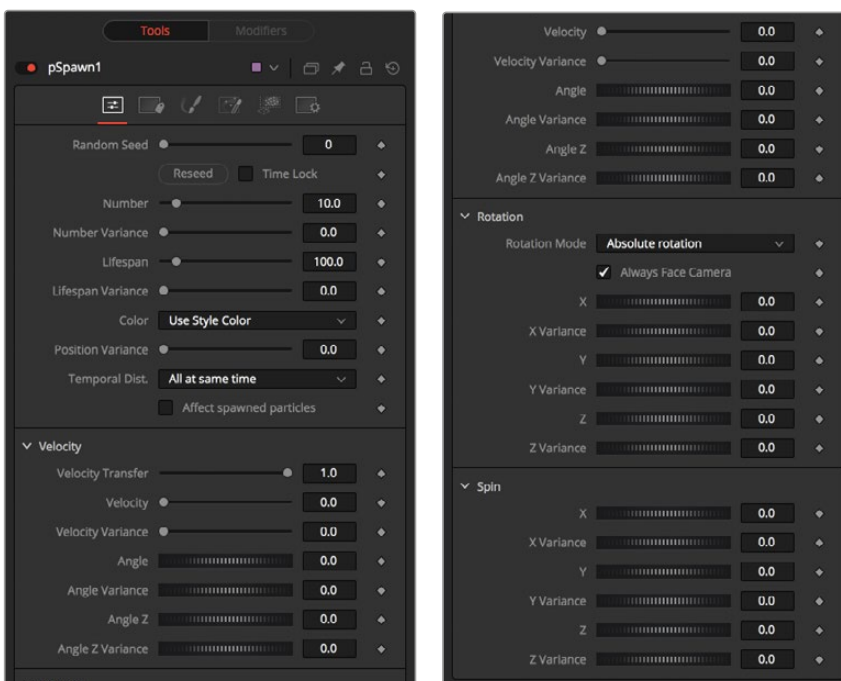
The pSpawn controls

Affect Spawned Particles

Selecting this checkbox causes particles created by spawning to also become affected by the pSpawn node on subsequent frames. This can exponentially increase the number of particles in the system, driving render times up to an unreasonable degree. Use this checkbox cautiously.

Velocity Transfer

This control determines how much velocity of the source particle is transferred to the particles it spawns. The default value of 1.0 causes each new particle to adopt 100 percent of the velocity and direction from its source particle. Lower values will transfer less of the original motion to the new particle.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pTangent Force [pTF]
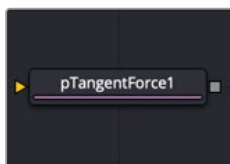
The pTangent Force node

pTangent Force Node Introduction

This node is used to apply a tangential force to the particles—a force that is applied perpendicularly to the vector between the pTangent Force's region and the particle it is affecting.

Inputs

The pTangent Force node has a single orange input by default. Like most particle nodes, this orange input accepts only other particle nodes. A green bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Region:** The green or magenta region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area where the tangent force effects the particles.
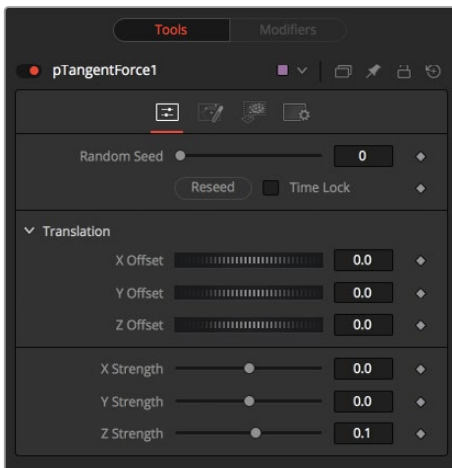
Basic Node Setup

The pTangent Force node is inserted between a pEmitter and a pRender node.
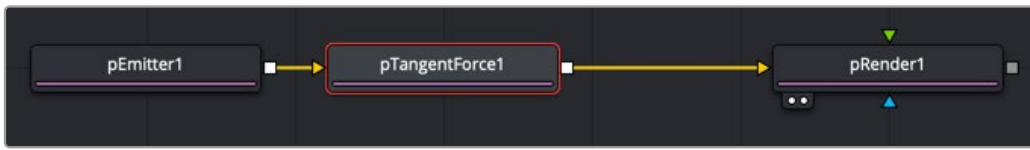
The pTangent Force node positions a tangent force that particles maneuver around.

Inspector

The pTangent Force controls

The controls for this node are used to position the offset in 3D space and to determine the strength of the tangential force along each axis independently.

Randomize

The Random Seed slider and Randomize button are presented whenever a Fusion node relies on a random result.

Two nodes with the same seed values will produce the same random results. Click the Randomize button to randomly select a new seed value, or adjust the slider to manually select a new seed value.

X, Y, Z Center Position

These controls are used to represent the X, Y, and Z coordinates of the Tangent force in 3D space.

X, Y, Z Center Strength

These controls are used to determine the Strength of the Tangent force in 3D space.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pTurbulence [pTr]

The pTurbulence node

pTurbulence Node Introduction

The pTurbulence node imposes a frequency-based chaos on the position of each particle, causing the motion to become unpredictable and uneven. The controls for this node affect the strength and density of the Turbulence along each axis.

Inputs

The pTurbulence node has a single orange input by default. Like most particle nodes, this orange input accepts only other particle nodes. A green bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Region:** The green or magenta region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area of turbulence.

Basic Node Setup

The pTurbulence node is inserted between a pEmitter and a pRender node.
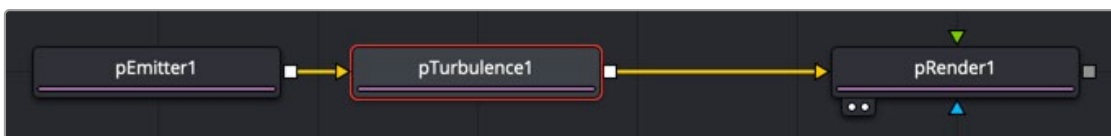
The pTurbulence node disturbs the rigid flow of particles for a more natural motion.

Inspector
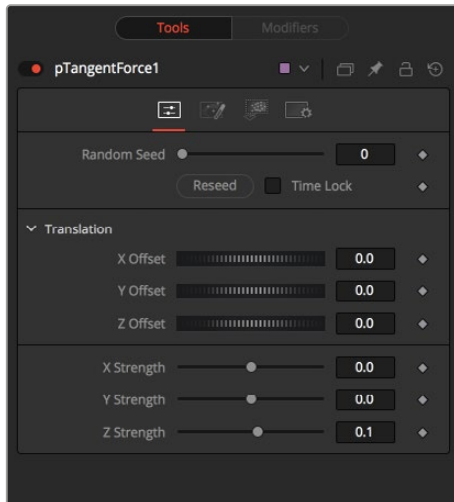
The pTurbulence controls

Randomize

The Random Seed slider and Randomize button are presented whenever a Fusion node relies on a random result. Two nodes with the same seed values will produce the same random results. Click the Randomize button to randomly select a new seed value, or adjust the slider to manually select a new seed value.

X, Y, and Z Strength

The Strength control affects the amount of chaotic motion imparted to particles.

Strength Over Life

This mini Spline Editor control can be used to control the amount of turbulence applied to a particle according to its age. For example, a fire particle may originally have very little turbulence applied at the start of its life, and as it ages, the turbulence increases.

Density

Use this control to adjust the density in the turbulence field. Lower values causes more particle cells to be affected similarly, almost as if "waves" of the turbulence field run through the particles, affecting groups of cells at the same time. Higher values add finer variations to more individual particle cells causing more of a spread in the turbulence field.
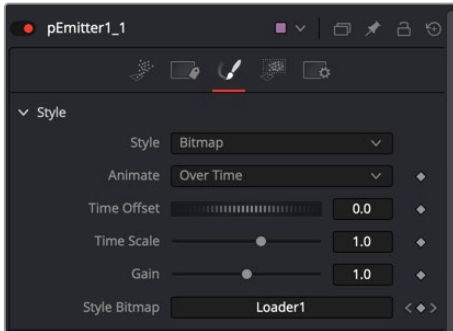
Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Style, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in "The Common Controls" section at the end of this chapter.

pVortex [pVt]

The pVortex node

pVortex Node Introduction

The pVortex node applies a rotational force to each particle, causing them to be drawn toward the source of the Vortex. In addition to the Common Particle Controls, the pVortex node also has the following controls.

Inputs

The pVortex node has a single orange input by default. Like most particle nodes, this orange input accepts only other particle nodes. A green bitmap or mesh input appears on the node when you set the Region menu in the Region tab to either Bitmap or Mesh.

— **Input:** The orange input takes the output of other particle nodes.
— **Region:** The green or magenta region input takes a 2D image or a 3D mesh depending on whether you set the Region menu to Bitmap or Mesh. The color of the input is determined by whichever is selected first in the menu. The 3D mesh or a selectable channel from the bitmap defines the area of the vortex.

Basic Node Setup

The pVortex node is placed in between the pEmitter and pRender nodes.

A pVortex node creates a spiraling motion for particles that fall within its pull.

Inspector
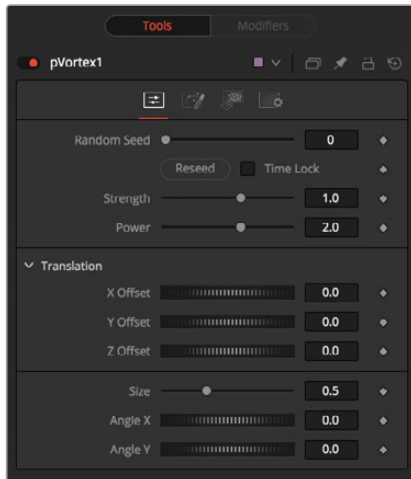
The pVortex controls

Randomize

The Random Seed slider and Randomize button are presented whenever a Fusion node relies on a random result. Two nodes with the same seed values will produce the same random results. Click the Randomize button to randomly select a new seed value, or adjust the slider to manually select a new seed value.

Strength

This control determines the Strength of the Vortex Force applied to each particle.

Power

This control determines the degree to which the Strength of the Vortex Force falls off with distance.

X, Y, and Z Offset

Use these sliders to set the amount by which the vortex Offsets the affected particles.

Size

This is used to set the Size of the Vortex Force.

Angle X and Y

These sliders control the amount of rotational force applied by the Vortex along the X and Y axes.

Common Controls

Conditions, Style, Region, and Settings Tabs

The Conditions, Region, and Settings tabs are common to all Particle nodes, so their descriptions can be found in the following "The Common Controls" section.

The Common Controls

Particle nodes share a number of identical controls in the inspector. This section describes the Style, Conditions, Region, and Settings tabs that are common among particle nodes.

Inspector

The pEmitter Style tab

Style Tab

The Style Tab is common to the pEmitter, pSpawn, pChangeStyle, and pImage Emitter. It controls the appearance of the particles using general controls like type, size, and color.

Style

The Style menu provides access to the various types of particles supported by the Particle Suite. Each style has its specific controls, as well as controls it will share with other styles.

— **Point:** This option produces particles precisely one pixel in size. Controls that are specific to Point Style are Apply Mode and Sub Pixel Rendered.

    — **Apply Mode:** This menu applies only to 2D particles. 3D particle systems are not affected. It includes Apply modes for Add and Merge. Add combines overlapping particles by adding together the color values of each particle. Merge uses a simple over operation to combine overlapping particles.

    — **Sub Pixel Rendered:** This checkbox determines whether the point particles are rendered with Sub Pixel precision, which provides smoother-looking motion but blurrier particles that take slightly longer to render.

— **Bitmap:** This style produces particle cells based on an image file or another node in the Node editor. When this option is selected an orange image input appears on the node in the node editor. There are several controls for affecting the appearance and animation. In addition to the controls in the Style section, a Merge section is displayed at the bottom of the inspector when Bitmap is selected as the Style. The Merge section includes controls for additive or subtractive merges when the particle cells overlap.

    — **Animate Over Time:** This menu includes three options for determining how movie files play when they are used as particle cell bitmaps. The Over Time setting plays the movie file sequentially. For instance, when the comps is on frame 2, frame 2 of the movie file is displayed, when the comp is on frame 3, frame 3 of the movie files is displayed and so on. If a particle cell is not generated until frame 50, it begins with frame 50 of the movie file. This

causes all particle cells to use the same image on any give frame of the comp. The Particle Age setting causes each particle cell to begin with the first frame of the movie file, regardless of when the particle cell is generated. The Particle Birth Time setting causes each particle to begin with the frame that coincides with the frame of the particle cell birth time. For instance, if the particle is generated on frame 25, then it uses frame 25 of the movie file for the entire comp. Unlike the other two options, the Particle Birth Time setting holds the same frame for the duration of the comp

— **Time Offset:** This dial is used to slip or offset the starting frame used from the movie file. For instance, setting it to 10 will cause the movie file to start at frame 10 instead of frame 1.

— **Time Scale:** This slider is a multiplier on the frame. Instead of using an offset, it changes the starting frame by multiplying the frame by the value selected with the slider. For instance, if a value of 2 is selected then when the playhead reaches frame 2, the movie files displays frame 4 (2x2=4) and when the playhead reaches frame 8, the movie file displays frame 16 (8x2=16).

— **Gain:** The gain slider is a multiplier of the pixel value. It is used to apply a correction to the overall Gain of the Bitmap. Let's say you have a bitmap particle cell that contains a pixel value of R0.5 G0.5 B0.4 and you add a Gain of 1.2, you end up with a pixel value of R0.6 G0.6, B0.48 (i.e., 0.4 * 1.2 = 0.48) while leaving black pixels unaffected. Higher values produce a brighter image, whereas lower values reduce both the brightness and the transparency of the image.

— **Style Bitmap:** This control appears when the Bitmap style is selected, along with an orange Style Bitmap input on the node's icon in the Node view. Connect a 2D node to this input to provide images to be used for the particles. You can do this on the Node view, or you may drag and drop the image source node onto the Style Bitmap control from the Node Editor or Timeline, or right-click on the control and select the desired source from the Connect To menu.

— **Blob:** This option produces large, soft spherical particles, with controls for Color, Size, Fade timing, Merge method, and Noise.

— **Noise:** This slider only applies to 2D Blob particles. The noise slider is used to introduce a computer generated Perlin noise pattern into the blob particles in order to give the blobs more texture. A setting of 0 introduces no noise to the Blob particles and a setting of 1 introduces the maximum amount of noise.

— **Brush:** This styles produces particle cells based on any image file located in the brushes directory. There are numerous controls for affecting the appearance and animation.

— **Gain:** The gain slider is a multiplier of the pixel value. It is used to apply a correction to the overall Gain of the image that is used as the Brush. Let's say you have a brush particle cell that contains a pixel value of R0.5 G0.5 B0.4 and you add a Gain of 1.2, you end up with a pixel value of R0.6 G0.6, B0.48 (i.e., 0.4 * 1.2 = 0.48) while leaving black pixels unaffected. Higher values produce a brighter image, whereas lower values reduce both the brightness and the transparency of the image.

— **Brush:** This menu shows the names of any image files stored in the Brushes directory. The location of the Brushes directory is defined in the Preferences dialog, under Path Maps. The default is the Brushes subdirectory within Fusion's install folder.

— **Use Aspect From:** The Use Aspect From menu includes three settings for the aspect ratio of the brush image. You can choose image format to use the brush image's native aspect ration. Choose Frame Format to use the aspect ratio set in the Frame Format Setting in the Fusion Preferences, or choose Custom to enter your own Pixel X and Y dimensions.

— **Line:** This style produces straight line-type particles with optional "falloff." The Size to Velocity control described below (under Size Controls) is often useful with this Line type. The Fade control adjusts the amount of falloff over the length of the line.

— **Point Cluster:** This style produces small clusters of single-pixel particles. Point Clusters are similar to the Point style; however, they are more efficient when a large quantity of particles is required. This style shares parameters with the Point style. Additional controls specific to Point Cluster style are Number of Points and Number Variance.

— **Sub Pixel Rendered:** This checkbox determines whether the point particles are rendered with Sub Pixel precision, which provides smoother-looking motion but blurrier particles that take slightly longer to render.

— **Number of Points and Variance:** The value of this control determines how many points are in each Point Cluster.

Color Controls

The Color Controls select the color and Alpha values of the particles generated by the emitter.

Color Variance

These range controls provide a means of expanding the colors produced by the pEmitter. Setting the Red variance range at -0.2 to +0.2 will produce colors that vary 20% on either side of the red channel, for a total variance of 40%. If the pEmitter is set to produce R0.5, G0.5, B0.5 (pure gray), the variance shown above will produce points with a color range between R0.3, G0.5, B0.5, and R0.7, G0.5, B0.5.

To visualize color space as values between 0-256 or as 0-65535, change the values used by Fusion using the Show Color As option provided in the General tab within the Preferences dialog.

Lock Color Variance

This checkbox locks the color variance of the particles. Unlocking this allows the color variance to be applied differently to each color channel, giving rise to a broader range of colors.

Particles Color Over Life controls

Color Over Life

This standard gradient control allows for the selection of a range of color values to which the particle will adhere over its lifetime.

The left point of the gradient represents the particle color at birth. The right point shows the color of the particle at the end of its lifespan.

Additional points can be added to the gradient control to cause the particle color to shift throughout its life.

This type of control can be useful for fire-type effects (for example, the flame may start blue, turn orange, and end a darker red). The gradient itself can be animated over time by right-clicking on the control and selecting Animate from the contextual menu. All points on the gradient will be controlled by a single Color Over Life spline, which controls the speed at which the gradient itself changes. You may also use the From Image modifier, which produces a gradient from the range of colors in an image along a line between two points.

Particles Size and Fade controls

Size Controls

The majority of the Size Controls are self-explanatory. The Size and Size Variance controls are used to determine the size and degree of size variation for each particle. It is worth noting that the Point style does not have size controls (each point is a single pixel in size, and there is no additional control).

When a Bitmap Particle style is used, a value of 1.0 indicates that each particle should be the same size as the input bitmap. A value of 2.0 will scale the particle up in size by 200%. For the best quality particles, always try to make the input bitmap as big, or bigger, than the largest particle produced by the system.

For the Point Cluster style, the size control adjusts the density of the cluster, or how close together each particle will get.

There are additional size controls that can be used to adjust further the size of particles based on velocity and depth.

Size to Velocity

This increases the size of each particle relative to the velocity or speed of the particle. The velocity of the particle is added to the size, scaled by the value of this control.

1.0 on this control, such as for a particle traveling at 0.1, will add another 0.1 to the size (velocity * size to velocity + size = new size). This is most useful for Line styles, but the control can be used to adjust the size of any style.

Size Z Scale

This control measures the degree to which the size of each particle changes according to its Z position. The effect is to exaggerate or reduce the impact of perspective. The default value is 1.0, which provides a relatively realistic perspective effect.

Objects on the focal plane (Z = 0.0) will be actual-sized. Objects farther along Z will become smaller. Objects closer along Z will get larger.

A value of 2.0 will exaggerate the effect dramatically, whereas a value of 0.0 will cancel the effects of perspective entirely.

Size Over Life

This spline control determines the size of a particle throughout its lifespan. The vertical scale represents a percentage of the value defined by the Size control, from 0 to 200%. The horizontal scale represents a percentage of the particle's lifespan (0 to 100%).

This graph supports all the features available to a standard spline editor. These features can be accessed by right-clicking on the graph. It is also possible to view and edit the graph spline in the larger Spline Editor.

Fade Controls

This simple range slider provides a mechanism for fading a particle at the start and end of its lifetime. Increasing the Fade In value will cause the particle to fade in at the start of its life. Decreasing the Fade Out value will cause the particle to fade out at the end of its life.

This control's values represent a percentage of the particle's overall life, therefore, setting the Fade In to 0.1 would cause the particle to fade in over the first 10% of its total lifespan. For example, a particle with a life of 100 frames would fade in from frame 0…10.

Merge Controls

This set of particle controls affects the way individual particles are merged together. The Subtractive/Additive slider works as documented in the standard Merge node. The Burn-In control will cause the particles to overexpose, or "blow out," when they are combined.

None of the Merge controls will have any effect on a 3D particle system.

Particles Blur controls

Blur Controls

This set of particle controls can be used to apply a Blur to the individual particles. Blurring can be applied globally, by age, or by Z depth position.

None of the Blur controls will have any effect on a 3D particle system.

Blur (2D) and Blur Variance (2D)

These controls apply blur to each particle. Unlike the Blur in the pRender node, this is applied to each particle independently before the particles are merged together. The Blur Variance slider modifies the amount of blur applied to each particle.

Blur Over Life

This spline graph controls the amount of blur that is applied to the particle over its life. The vertical scale represents a percentage of the value defined by the Blur control. The horizontal scale represents a percentage of the particle's lifespan.

This graph supports all of the features available to a standard Spline Editor. These features can be accessed by right-clicking on the graph. It is also possible to view and edit the spline in the larger Spline editor.

Z Blur (DoF) (2D) and DoF Focus

This slider control applies blur to each particle based on its position along the Z axis.

The DoF Focus range control is used to determine what area of the image remains in focus. Lower values along Z are closer to the camera. Higher values are farther away. Particles within the range will remain in focus. Particles outside that range will have the blur defined by the Z Blur control applied to them.

Conditions
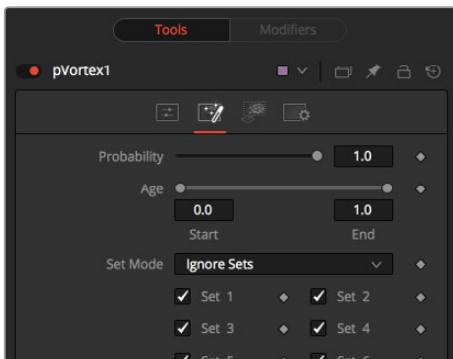
pVortex Conditions tab

Conditions Tab

The Conditions tab limits the particles that are affected by the node's behavior. You can limit the particle using probability or more specifically using sets.

Probability

The Probability slider determines the percentage of chance that the node affects any given particle.

The default value of 1.0 affects all particles. A setting of 0.6 would mean that each particle has a 60 percent chance of being affected by the control.

Probability is calculated for each particle on each frame. For example, a particle that is not affected by a force on one frame has the same chance of being affected on the next frame.

Start/End Age

This range control can be used to restrict the effect of the node to a specified percentage of the particle lifespan.

For example, to restrict the effect of a node to the last 20 percent of a particle's life, set the Start value to 0.8, and the End value remains at 1.0. The node on frames 80 through 100 only affects a particle with a lifespan of 100 frames.

Set Mode Menu

The Set Mode menu drives how the particle node influences the active particle sets. There are three options from this menu:

— **Ignore Sets:** The particle node disregards the state of the Set checkboxes and applies to all nodes.
— **Affect Specified Sets:** The particle node applies its behavior to the active Set checkboxes only.
— **Ignore Specified Sets:** The particle node applies its behavior to the inactive Set checkboxes only.

Set #

The state of a Set # checkbox determines if the Particle node's effect will be applied to the particles in the set. It allows you to limit the effects of some nodes to a subset of particles.

Sets are assigned by the nodes that create particles. These include the pEmitter, pImage Emitter, pChangeStyle, and the pSpawn nodes.

Region Tab

The Region tab is used to restrict the node's effect to a geometric region or plane, and to determine the area where particles are created if it's a pEmitter node or where the behavior of a node has influence.
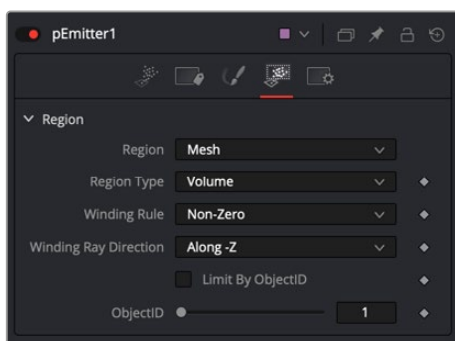
pEmitter Regions tab set to mesh

The Region tab is common to almost all particle nodes. In the pEmitter node Emitter Regions are used to determine the area where particles are created. In most other tools it is used to restrict the tool's effect to a geometric region or plane. There are seven types of regions, each with its own controls. Only one emitter region can be set for a single pEmitter node. If the pRender is set to 2D, then the emitter region will produce particles along a flat plane in Z Space. 3D emitter regions possess depth and can produce particles inside a user-defined, three-dimensional region.

Region Mode Menu

The Region Mode menu includes seven types of regions to define the area, each with its controls.

— **All:** In 2D, the particles will be created anywhere within the boundaries of the image. In 3D, this region describes a cube 1.0 x 1.0 x 1.0 units in size.

— **Bézier:** Bézier mode uses a user-created polyline to determine the region where particles are created. The Bézier mode works in both 2D and 3D modes; however, the Bézier polyline region can only be created in 2D.

  To animate the shape of the polyline over time or to connect it to another polyline, right-click the Shape animation label at the bottom of the inspector and select the appropriate option from the drop-down menu.

— **Bitmap:** A Bitmap source from one of the other nodes in the composition will be used as the region where particles are born.

— **Cube:** A full 3D Cube is used to determine the region within which particles are created. The height, width, depth, and XYZ positions can all be determined by the user and be animated over time.

— **Line:** A simple line control determines where particles are created. The line is composed of two end-points, which can be connected to Paths or Trackers, as necessary. This type of emitter region includes X, Y, and Z position controls for the start and end of the line

— **Mesh:** Any 3D Mesh can be used as a region. In Mesh mode, the region can also be restricted by the Object ID using the ObjectID slider. See below for a more in-depth explanation of how mesh regions work.

— **Rectangle:** The Rectangle region type is like the Cube type, except that this region has no depth in Z space. Unlike other 2D emitter regions, this region can be positioned and rotated in Z space.

— **Sphere:** This is a spherical 3D emitter region with Size and Center Z controls. Sphere (3D) is the default region type for a new pEmitter node.

Mesh Regions

Region Type

The Region Type drop-down menu allows you to choose whether the region will include the inner volume or just the surface. For example, with a pEmitter mesh region, this determines if the particles emit from the surface or the full volume.

Winding Rule and Winding Ray Direction

The Winding Rule and Winding Ray Direction parameters determine how the mesh region handles particle creation with meshes that are not closed, as is common in many meshes imported from external applications. This scenario is common with imported mesh geometry, and even geometry that appears closed will frequently appear to "leak" thanks to improperly welded vertices.

To determine if a particle is in the interior of an object, a ray is cast from infinity through that particle and then out to -infinity. The Winding Ray Direction determines which direction this ray is cast in. Each time a surface is pierced by the ray, it is recorded and added onto a total to generate a winding number. Going against a surfaces normal counts as +1, and going with the normal counts as -1.

The Winding Rule is then used to determine what is inside/outside. For example, setting the Winding Rule to Odd means that only particles with odd values for the winding number are kept when creating the particles. The exact same approach is used to ensure that polylines that intersect themselves are closed properly.

For example, the following node tree and image show two image planes being used as a mesh region for particle creation.
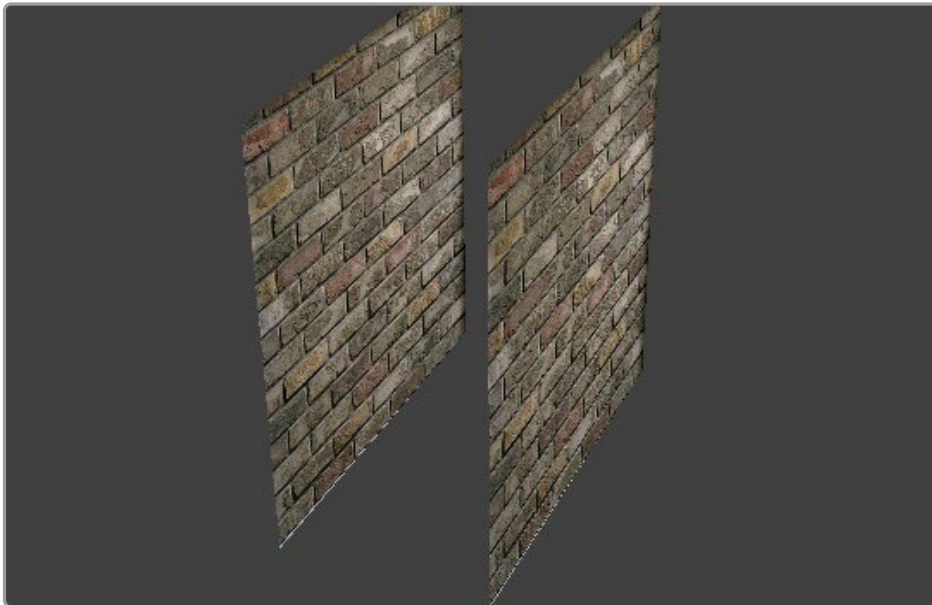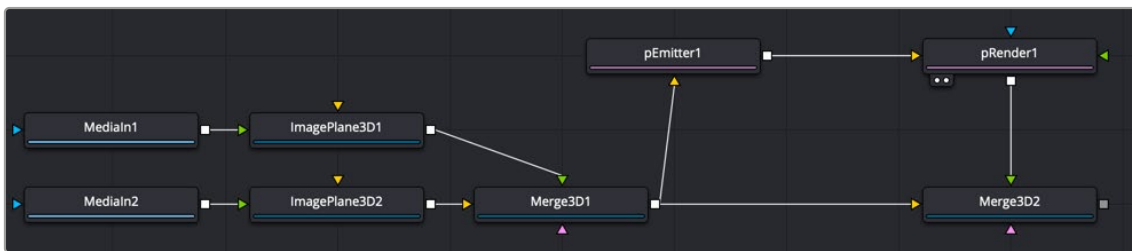
By setting the region's Winding Ray Direction to the Z (blue) axis, this mesh can then be treated as a closed volume for purposes of particle creation, as pictured below.
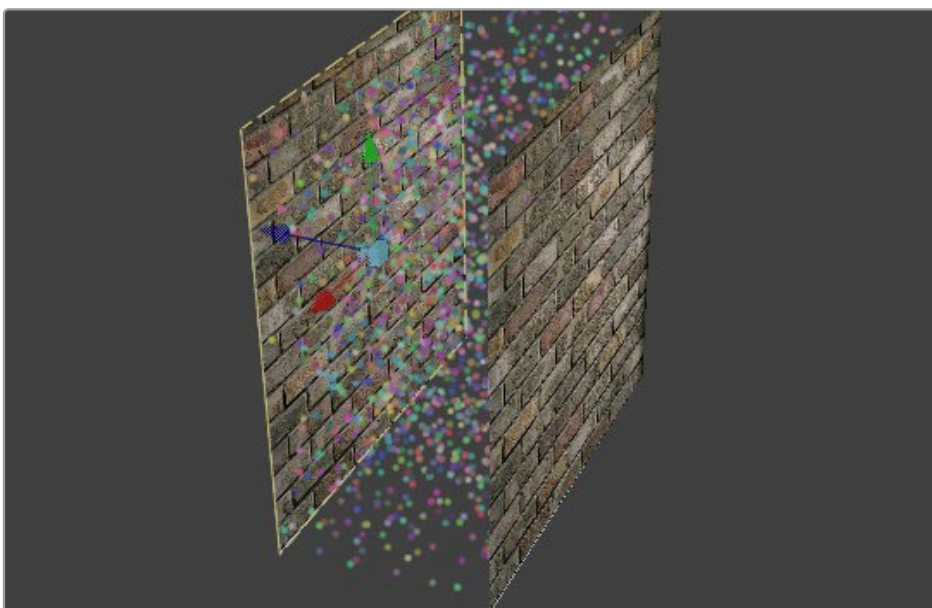
Limit By ObjectID

Selecting this checkbox allows the Object ID slider to select the ObjectID used as part of the region.

Style Tab

The Style tab exists in the pEmitter, pSpawn, pChangeStyle, and pImage Emitter. It controls the appearance of the particles, allowing the look of the particles to be designed and animated over time.

Style

The Style menu provides access to the various types of particles supported by the Particle Suite. Each style has its specific controls, as well as controls it will share with other styles.

— **Point Style:** This option produces particles precisely one pixel in size. Controls that are specific to Point style are Apply Mode and Sub Pixel Rendered.

— **Bitmap Style and Brush Style:** Both the Bitmap and Brush styles produce particles based on an image file. The Bitmap style relies on the image from another node in the node tree, and the Brush style uses image files in the Brushes directory. They both have numerous controls for affecting their appearance and animation, described below.

— **Blob Style:** This option produces large, soft spherical particles, with controls for Color, Size, Fade timing, Merge method, and Noise.

— **Line Style:** This style produces straight line-type particles with optional "falloff." The Size to Velocity control described below (under Size Controls) is often useful with this Line type. The Fade control adjusts the amount of falloff over the length of the line.

— **Point Cluster Style:** This style produces small clusters of single-pixel particles. Point Clusters are similar to the Point style; however, they are more efficient when a large quantity of particles is required. This style shares parameters with the Point style. Additional controls specific to Point Cluster style are Number of Points and Number Variance.

Style Options

The following options appear only on some of the styles, as indicated below.

Apply Mode (Point and Point Cluster)

This control applies only to 2D particles; 3D particle systems are not affected.

— **Add:** Overlapping particles are combined by adding together the color values of each particle.

— **Merge:** Overlapping particles are merged.

Sub Pixel Rendered (Point and Point Cluster)

This checkbox determines whether the point particles are rendered with Sub Pixel precision, which provides smoother-looking motion but blurrier particles that take slightly longer to render.

Number of Points and Variance (Point Cluster)

The value of this control determines how many points are in each Point Cluster.

Animate (Bitmap Style)

If the Bitmap source is a movie file or image sequence, this menu determines which frame is grabbed from the source and applied to newly-created particles.

— **Over Time:** All particles use the image produced by the Style Bitmap node at the current time, and change to each successive image together in step, as time increases. A particle created at frame 1 will contain the image at frame 1 of the Style Bitmap. At frame 2, the original particle will use the image from frame 2, and so will any new particles. All created particles will share the same bitmap image from their source at all times.

— **Particle Age:** Each particle animates through the sequence of images provided by the Style Bitmap node, independently of other particles. In other words, an individual particle's appearance is taken from the Style Bitmap node at successive times, indexed by its age.

— **Particle Birth Time:** New particles take the image from the Style Bitmap node at the current time and keep it unchanged until the end of the particle's lifespan. Thus, particles generated on a given frame will all have the same appearance and will stay that way.

Time Offset (Bitmap Style)

This control allows the Bitmap source frame to be offset in time from the current frame.

Time Scale (Bitmap Style)

This control scales the time range of the source bitmap images by a specified amount. For example, a scale of 2 will cause the particle created at frame 1 to be read from the bitmap source at frame 2.

Gain (Bitmap and Brush Style)

This control applies a gain correction to the image used as the bitmap. Higher values produce a brighter image, whereas lower values reduce both the brightness and the transparency of the image.

Style Bitmap (Bitmap Style)

This control appears when the Bitmap style is selected, along with an orange Style Bitmap input on the node's icon in the Node view. Connect a 2D node to this input to provide images to be used for the particles. You can do this on the Node view, or you may drag and drop the image source node onto the Style Bitmap control from the Node Editor or Timeline, or right-click on the control and select the desired source from the Connect To menu.

Brush (Brush Style)

This menu shows the names of any image files stored in the Brushes directory. The location of the Brushes directory is defined in the Preferences dialog, under Path Maps. The default is the Brushes subdirectory within Fusion's install folder. If no images are in this directory, the only option in the menu will be None, and no particles will be rendered.

Noise (Blob Style)

Increasing this control's value will introduce grain-type noise to the blobby particle.

Fade (Line Style)

The Fade control adjusts the falloff over the line particle's length.

The default value of 1.0 causes the line to fade out completely by the end of the length.

Color Controls

The Color Controls select the color and Alpha values of the particles generated by the emitter.
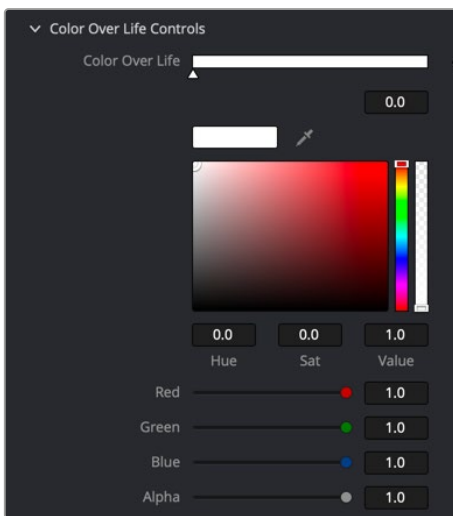
Color Variance

These range controls provide a means of expanding the colors produced by the pEmitter. Setting the Red variance range at -0.2 to +0.2 will produce colors that vary 20% on either side of the red channel, for a total variance of 40%. If the pEmitter is set to produce R0.5, G0.5, B0.5 (pure gray), the variance shown above will produce points with a color range between R0.3, G0.5, B0.5, and R0.7, G0.5, B0.5.

To visualize color space as values between 0-256 or as 0-65535, change the values used by Fusion using the Show Color As option provided in the General tab within the Preferences dialog.

Lock Color Variance

This checkbox locks the color variance of the particles. Unlocking this allows the color variance to be applied differently to each color channel, giving rise to a broader range of colors.

Particles Color Over Life controls

Color Over Life

This standard gradient control allows for the selection of a range of color values to which the particle will adhere over its lifetime.

The left point of the gradient represents the particle color at birth. The right point shows the color of the particle at the end of its lifespan.

Additional points can be added to the gradient control to cause the particle color to shift throughout its life.

This type of control can be useful for fire-type effects (for example, the flame may start blue, turn orange and end a darker red). The gradient itself can be animated over time by right-clicking on the control and selecting Animate from the contextual menu. All points on the gradient will be controlled by a single Color Over Life spline, which controls the speed at which the gradient itself changes. You may also use the From Image modifier, which produces a gradient from the range of colors in an image along a line between two points.

Position Nodes

This chapter details the Position nodes available in Fusion.

> The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.
>
> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Volume Fog [VlF]

The Volume Fog node

Volume Fog Node Introduction

The Volume Fog node is used to create sophisticated volumetric fog on images containing XYZ Position channels.

As opposed to 3D-rendered volumetric fog, it works on 2D images and delivers much faster results and interactive feedback when setting up the fog. See the "WPP Concept" section at the end of this chapter for further explanation of how this technology works and to learn about the required imagery.

Basic Node Setup

The Volume Fog node takes an image input; in the example below, it is a Renderer 3D with World Position enabled in the output channels. Another input is the 3D scene, which contains the camera. A Fast Noise node generates the fog texture.

Volume Fog Node Structure
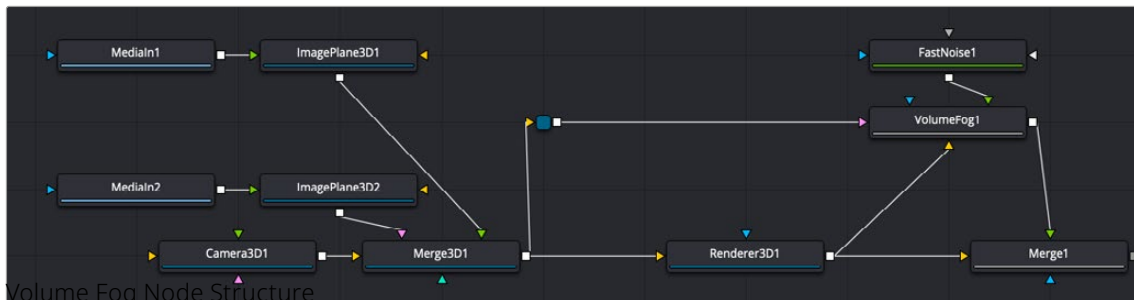
A 3D scene and a rendered scene connected to Volume Fog

Inputs

The following inputs appear on the Volume Fog node in the Node Editor.

— **Image:** The orange input accepts the primary image where the fog will be applied. This image contains a World Position Pass in the XYZ Position channels.

— **Fog Image:** The green Fog image input is for creating volumetric fog with varying depth and extent; a 2D image can be connected here. A good starting point is to use a Fast Noise at a small resolution of 256 x 256 pixels.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the fog to certain areas.

— **Scene Input:** The magenta scene input accepts a 3D scene containing a 3D Camera.

Inspector
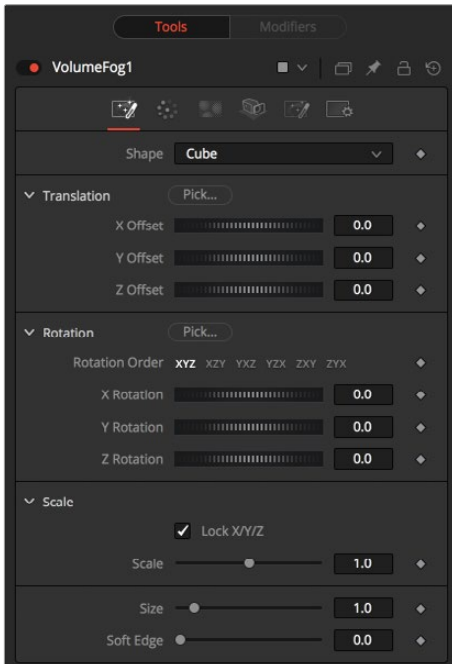
The Volume Fog Shape tab

Shape Tab

The Shape tab defines the size and location of the fog volume. You can either use the Pick buttons to select the location and orientation in the viewer or use the Translation, Rotation, and Scale controls.

Shape

This menu switches between a basic spherical or rectangular volume to be placed in your image. These volumes can then be further refined using the Fog image and effect mask.

Pick

Drag the Pick button into the viewer to select the XYZ coordinates from any 3D scene or 2D image containing XYZ values, such as a rendered World Pass, to position the center of the Volume object. When picking from a 2D image, make sure it's rendered in 32-bit float to get full precision.

X, Y, Z Offset

These controls can be used to position the center of the fog volume manually or can be animated or connected to other controls in Fusion.

Rotation Pick

Drag the Pick button into the viewer to select the rotational values from any 3D Scene or 2D image containing those values, like an XYZ-Normal-Pass, to reorient the fog volume.

When picking from a 2D image, like an XYZ Normal pass, make sure it's rendered in 32-bit float to get full precision and accurate rotational values.

X, Y, Z Rotation

Use these controls to rotate the fog volume around its center.

X, Y, Z Scale

Scale the fog volume in any direction from its center to refine further the overall Size value specified below.

Size

The overall size of the fog volume created.

Soft Edge

Controls how much the fog volume is faded toward the center from its perimeter to achieve a softer look.
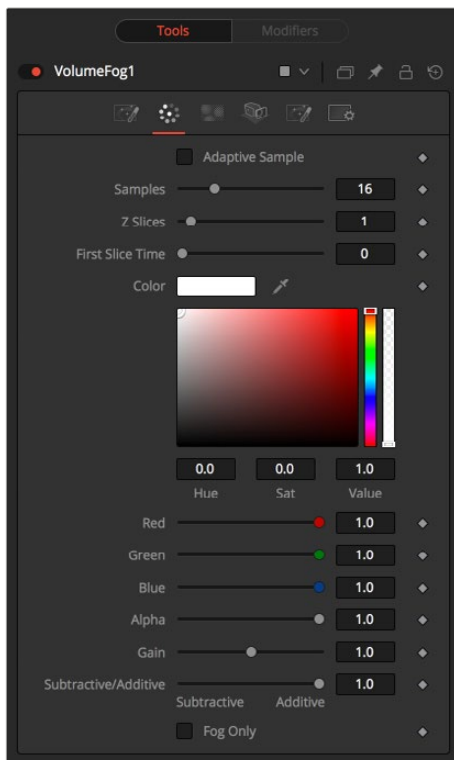
The Volume Fog Color tab

Color Tab

The Color tab controls the detail and color of the fog.

Adaptive Samples

Volumes images consist of multiple layers, so there may be 64 layers in a volume. This checkbox adjusts the rendering algorithm for how to best blend those layers.

Dither: Applies a form of noise to improve the blending and hide visible layer differences.

Samples

Determines how many times a "ray" shot into the volume will be evaluated before the final image is created. Not unlike raytracing, higher values lead to more detail inside the volume but also increase render times.

Z Slices

The higher the Z Slices value, the more images from the connected Fog image sequence will be used to form the depth of the volume.

You can, for example, use a Fast Noise with a high Seethe Rate to create such a sequence of images. Be careful with the resolution of the images. Higher resolutions can require a large amount of memory. As a rule of thumb, a resolution of 256 x 256 pixels with 256 Z Slices (i.e., forming a 256 x 256 x 256 cubic volume, which will use up to 256 MB for full color 32-bit float data) should give you a good starting point.

First Slice Time

Determines which frame of the Global Range is used to deliver the first slice from the connected fog image sequence.

Make sure that both Global In and Global Out, as well as the valid range of your source node, fall within the range of First Slice Time + Z Slices.

Color

Allows you to modify the color of the fog generated. This will multiply over any color provided by the connected Fog image.

Gain

Increases or decreases the intensity of the fog. More Gain will lead to a stronger glow and less transparency in the fog. Lower values let the fog appear less dense.

Subtractive/Additive Slider

Similar to the Merge node, this value controls whether the fog is composed onto the image in Additive or Subtractive mode, leading to a brighter or dimmer appearance of the fog.

Fog Only

This option outputs the generated fog on a black background, which then can be composited manually or used as a mask on a Color Corrector for further refinement.
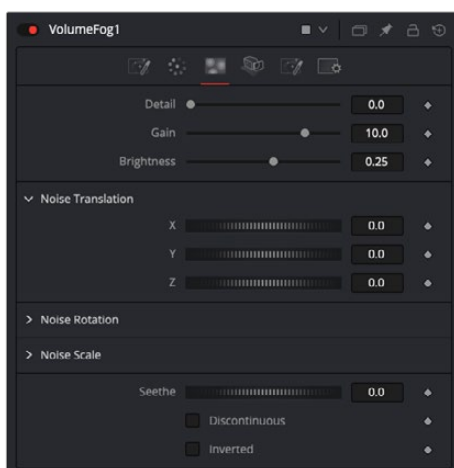
The Volume Fog Noise tab

Noise Tab

The Noise tab controls the shape and pattern of the noise added to the fog.

Detail

Increase the value of this slider to produce a greater level of detail in the noise result. Larger values add more layers of increasingly detailed noise without affecting the overall pattern. High values take longer to render but can produce a more natural result.

Gain

This control increases or decreases the brightest parts of the noise map.

Brightness

This control adjusts the overall brightness of the noise map, before any gradient color mapping is applied. In Gradient mode, this produces a similar effect to the Offset control.

Translation

Use the Translation coordinate control to pan and move the noise pattern.

Noise Rotation

Use the Rotation controls to orient the noise pattern in 3D.

Seethe

Adjust this thumbwheel control to interpolate the noise map against a different noise map. This will cause a crawling shift in the noise, like it was drifting or flowing. This control must be animated to affect the noise over time.

Discontinuous

Normally, the Noise function interpolates between values to create a smooth, continuous gradient of results. Enable this checkbox to create hard discontinuity lines along some of the noise contours. The result will be a dramatically different effect.

Inverted

Select this checkbox to invert the noise, creating a negative image of the original pattern. This is most effective when Discontinuous is also enabled.
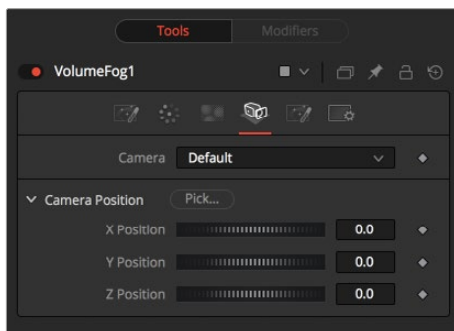
The Volume Fog Camera tab

Camera Tab

For a perfect evaluation of a fog volume, a camera or 3D scene can be connected to the Scene input of the node.

Camera

If multiple cameras are available in the connected Scene input, this menu allows the selection of the correct camera needed to evaluate the fog volume. Instead of connecting a camera, position values can be provided manually or by connecting the XYZ values to other controls.

Translation Pick

Drag the Pick button into the viewer to select XYZ coordinates from any 3D scene or 2D image containing XYZ values, like a rendered World Pass, to define the center of the camera. When picking from a 2D image, make sure it's rendered in 32-bit float to get full precision.

X, Y, Z Offset

These controls can be used to define the center of the camera manually or can be animated or connected to other controls in Fusion.
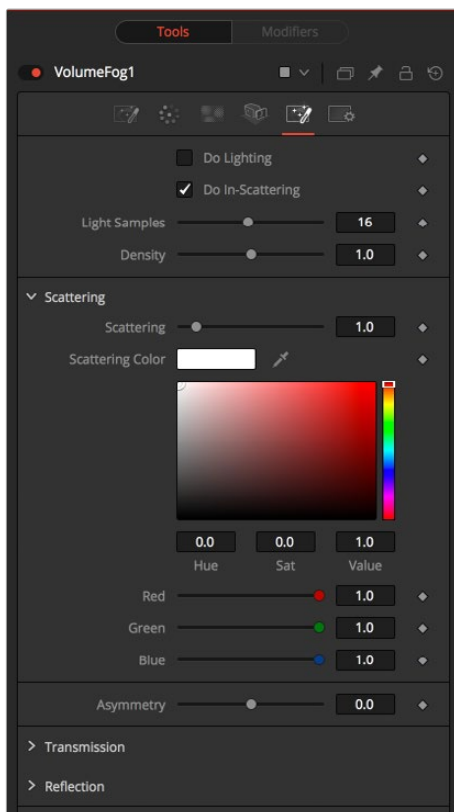
The Volume Fog Light tab

Light Tab

To utilize the controls in the Light tab, you must have actual lights in your 3D scene. Connect that scene, including Camera and Lights, to the 3D input of the node.

Do Lighting

Enables or disables lighting calculations. Keep in mind that when not using OpenCL (i.e., rendering on the CPU), these calculations may become a bit slow.

Do In-Scattering

Enables or disables light-scattering calculations. The volume will still be lit according to the state of the Do Lighting checkbox, but scattering will not be performed.

Light Samples

Determines how accurate the lighting is calculated. Higher values mean more accurate calculation at the expense of longer render times.

Density

This is similar to scattering in that it makes the fog appear thicker. With a high amount of scattering, though, the light will be scattered out of the volume before it has had much chance to travel through the fog, meaning it won't pick up a lot of the transmission color. With a high density instead, the fog still appears thicker, but the light gets a chance to be transmitted, thus picking up the transmission color before it gets scattered out. Scattering is affected by the light direction when Asymmetry is not 0.0. Density is not affected by light direction at all.

Scattering

Determines how much of the light bouncing around in the volume ends up scattering the light out of the fog. If the light scatters more, or more accurately, then there's a higher probability of the light being scattered out of the volume, hence less light is left to continue through the fog. This option can make the fog seem denser.

Asymmetry

Determines in what direction the light is scattered. A value of 0 produces uniform, or isotropic, scattering, meaning all directions have equal probability. A value greater than 0 causes "forward scattering," meaning the light is scattered more into the direction of the light rays. This is similar to what happens with water droplets in clouds. A value smaller than 0 produces "back scattering," where the light is more scattered back toward the original light source.

Transmission

Defines the color that is transmitted through the fog. The light that doesn't get scattered out will tend toward this color. It is a multiplier, though, so if you have a red light, but blue transmission, you won't see any blue.

Reflection

Changes the intensity of the light that is scattered out. Reflection can be used to modify the overall color before Emission is added. This will be combined with the color channels of the volume texture and then used to scale the values. The color options and the color channels of the volume texture are multiplied together, so if the volume texture were red, setting the Reflection color options to blue would not make the result blue. In such a case, they will multiply together to produce black.

Emission

This adds a bit of "glowing" to the fog, adding energy/light back into the calculation. If there are no lights in the scene, and the fog emission is set to be 1.0, the results are similar to no lighting, like turning off the Do Lighting option. Glowing can also be done while producing a different kind of look, by having a Transmission greater than 1. This, however, would never happen in the real world.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Position nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

EXAMPLES In these examples, we are looking at a volume from the outside. On the left, you see how the Volume Fog looks with straight accumulation. That means the Do Lighting option is turned off.
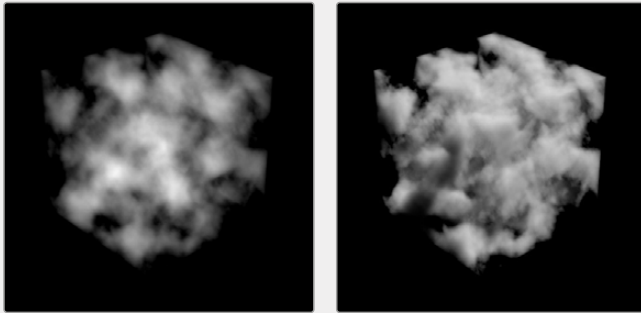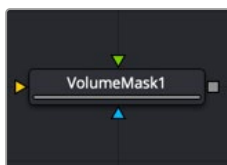

On the right, you see the same volume with lighting/scattering turned on, and a single point light.

Here, we have a slightly more complex Volume.

On the left with straight accumulation; in the middle with lighting, scattering, and a single point light; and on the right, the light in the scene has been moved, which also influences the look of the volume.

Volume Mask [VlM]

The Volume Mask node

Volume Mask Node Introduction

The Volume Mask node is used to create volumetric masks from images containing XYZ Position channels.

This can, for example, be used to isolate objects for color correction without the need to track or rotoscope the scene. See the "WPP Concept" section later in this chapter for further explanation on how this technology works and to learn about the required imagery.

Inputs

The following three inputs appear on the Volume Mask node in the Node Editor:

— **Image:** The orange image input accepts a 2D image containing a World Position Pass in the XYZ Position channels.

— **Mask Image:** An image can be connected to the green mask image input for refining the mask.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the volume mask to certain areas.

Basic Node Setup

Below, a MediaIn labeled RGBA contains the main rendered image from a 3D scene. A World Position Pass from a 3D-rendered scene is labeled WPP_PASS. The Channel Booleans node is used to map the X position pass to the red channel, Y to the green channel, and Z to the blue channel. The Volume Mask tool extracts the 3D position information from the World Position Pass to place a mask in 3D space. The mask is then used as an effect mask on a color corrector to selectively color correct objects in a 3D scene.
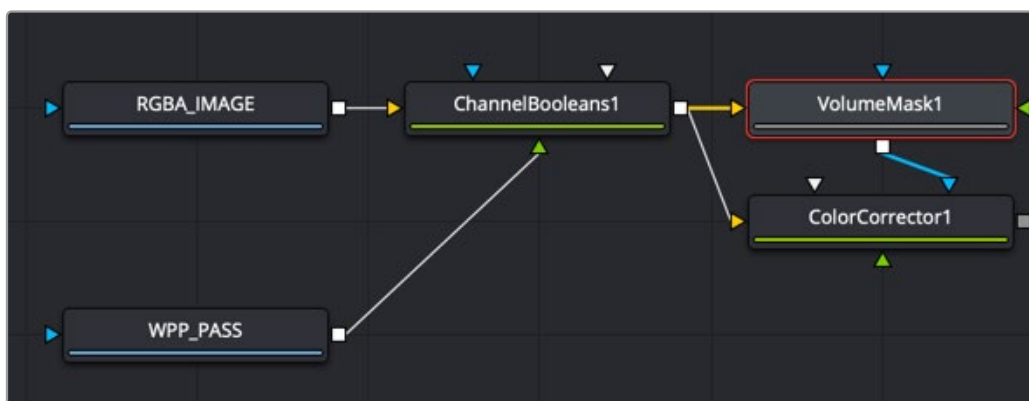
A Volume Mask tool takes advantage of World Position Pass for color correction in a 3D scene

Inspector
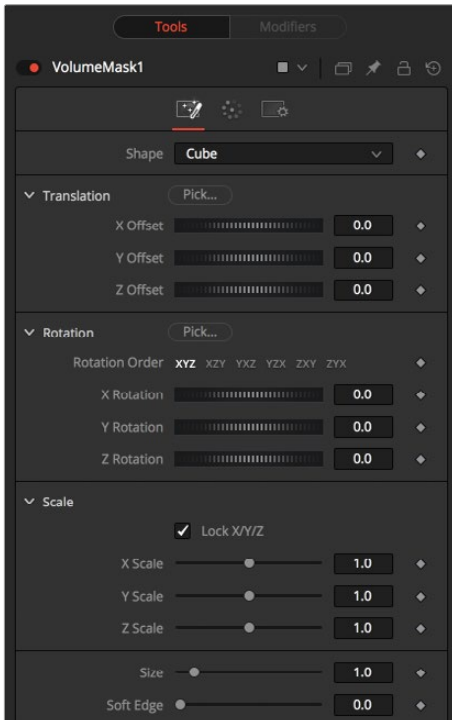
The Volume Mask Shape tab

Shape Tab

The Shape tab defines the size and location of the Volume Mask. You can either use the Pick buttons to select the location and orientation in the viewer or use the Translation, Rotation, and Scale controls.

Shape

This menu switches between a spherical or rectangular mask to be placed in your image. The mask can be further refined using the mask image input.

Translation Pick

Drag the Pick button into the viewer to select XYZ coordinates from any 3D scene or 2D image containing XYZ values, like a rendered World Pass, to position the center of the Volume Mask. When picking from a 2D image, make sure it's rendered in 32-bit float to get full precision.

X, Y, Z Offset

These controls can be used to position the center of the mask manually or can be animated or connected to other controls in Fusion.

Rotation Pick

Drag the Pick button into the viewer to select rotational values from any 3D scene or 2D image containing those values, like an XYZ Normal pass, to reorient the mask.

When picking from a 2D image, like an XYZ Normal pass, make sure it's rendered in 32-bit float, and use World Space coordinates to get full precision and the correct rotational values.

X, Y, Z Rotation

Use these controls to rotate the mask around its center.

X, Y, Z Scale

Scale the mask in any direction from its center to further refine the overall Size value specified below.

Size

The overall size, in X, Y, and Z, of the mask created.

Soft Edge

Controls how much the Volume is faded toward the center from its perimeter to achieve a softer look.
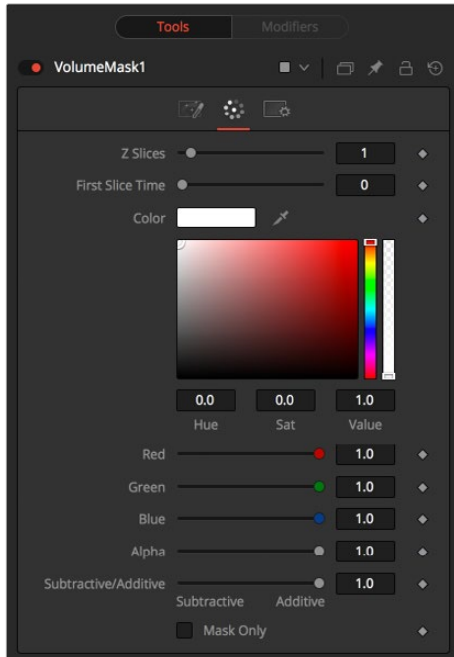
The Volume Mask Color tab

Color Tab

The Color tab controls the color and blending of the mask image.

Color

Allows you to modify the color of the generated Volume Mask. This will add to any color provided by the connected mask image.

Subtractive/Additive Slider

Similar to the Merge node, this value controls whether the mask is composed onto the image in Additive or Subtractive mode, leading to a brighter or dimmer appearance of the mask.

Mask Only

Outputs the generated mask on a black background, which then can be used as a mask on a Color Corrector for further refinement.
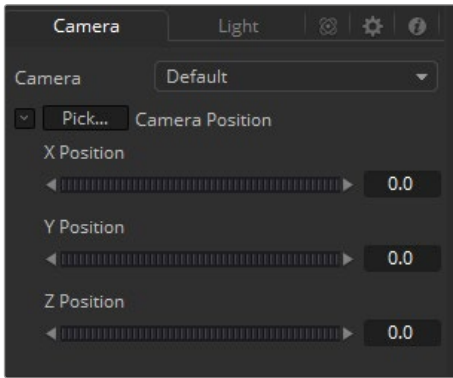
The Volume Mask Camera tab

Camera Tab

For a perfect evaluation of a Volume, a camera or 3D scene can be connected to the Scene input of the node.

Camera

If multiple cameras are available in the connected Scene input, this drop-down menu allows you to choose the correct camera needed to evaluate the Volume.

Instead of connecting a camera, position values can also be provided manually or by connecting the XYZ values to other controls.

Translation Pick

Drag the Pick button into the viewer to select XYZ coordinates from any 3D scene or 2D image containing XYZ values, like a rendered World Pass, to define the center of the camera.

When picking from a 2D image, make sure it's rendered in 32-bit float to get full precision.

X, Y, Z Offset

These controls can be used to define the center of the camera manually or can be animated or connected to other controls in Fusion.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Position nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Z to World Pos [Z2W]
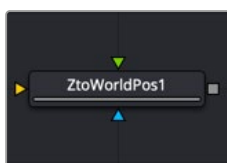

The Z to World Position node

Z to World Pos Node Introduction

The Z to World Pos node is used to either generate a World Position Pass from a Z channel and a 3D Camera or a Z channel from a World Position Pass and a 3D Camera.

Creating a World Position Pass from Z-depth can be useful when your 3D application is not capable of creating a WPP.

It can also be used when a 3D-tracking software outputs a per-pixel Z-depth together with the 3D Camera. Thus, the Volume Mask and Volume Fog could be applied to real-world scenes. The quality of the resulting WPP depends mainly on the quality of the incoming Z channel.

See the "WPP Concept" section for further explanation on how this technology works and to learn about the required imagery.

Inputs

The following inputs appear on the node tile in the Node Editor:

— **Image:** The orange image input accepts an image containing a World Position Pass or a Z-depth pass, depending on the desired operation.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the World Position Pass to certain areas.
— **Scene Input:** The magenta scene input accepts a 3D scene input containing a 3D Camera.

Basic Node Setup

Below, a MediaIn labeled RGBA contains the main rendered image from a 3D scene. A Z-depth pass from a 3D-rendered scene is labeled Z_PASS. The Channel Booleans node is used to map the Aux Z channel into either the red, green, or blue channel. The Z to World Position node is placed after the Channel Booleans node, and an imported 3D camera that matches the RGBA image is connected to the 3D camera input on the Z to World Position node. A Channel Booleans node is placed after the Z to World Position node, which can remap the X, Y, and Z positions for use in other nodes.
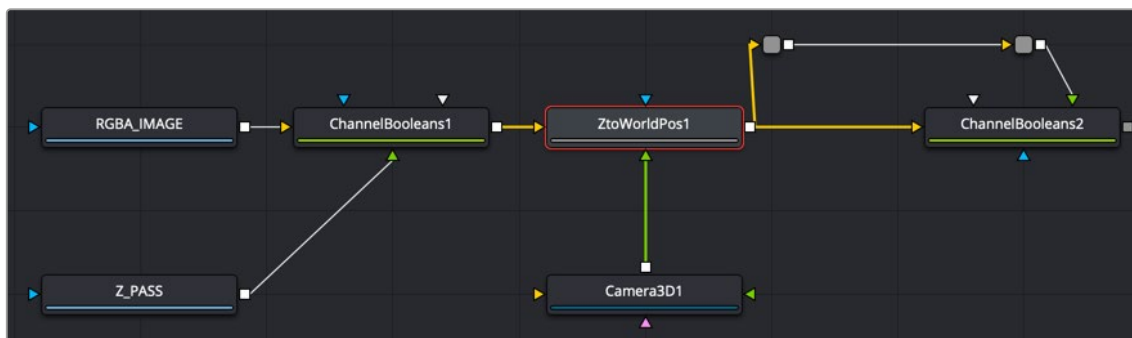
A Z to World Position node creates a World Position Pass from a Z-depth pass

Inspector
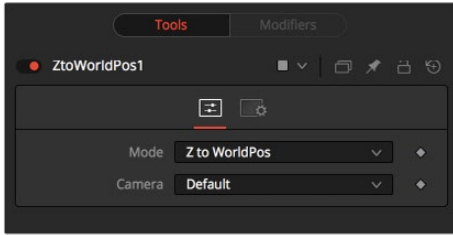
The Z to World Position Controls tab

Controls Tab

The Controls tab determines whether you are creating a World Position Pass or a Z channel. If there is more than one camera in the connected scene, this tab also selects the camera to use for the calculation.

Mode

This menu switches between creating a Z channel from a World Position Pass or vice versa.

Camera

If multiple cameras are available in the connected Scene input, this drop-down menu allows you to choose the correct camera needed to evaluate the image.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Position nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

WPP Concept

The Position nodes in Fusion offer an entirely new way of working with masks and Volumetrics for footage containing XYZ Position channels. Z to World offers the option to create those channels out of a Z channel and 3D Camera information. For this overview, we refer to the World Position Pass as WPP.

What Is a WPP?

The WPP interprets each pixel's XYZ position in 3D space as an RGB color value.

For instance, if a pixel sits at 0/0/0, the resulting pixel has an RGB value of 0/0/0 and thus will be black. If the pixel sits at 1/0/0 in the 3D scene, the resulting pixel is entirely red. Of course, if the coordinates of the pixel are something like -60/75/123, WPP interprets those values as RGB color values as well.

Due to the potentially enormous size of a 3D scene, the WPP channel should always be rendered in 32-bit floating point to provide the accuracy needed. The image below shows a 3D rendering of a scene with its center sitting at 0/0/0 in 3D Space and the related WPP. For better visibility, the WPP is normalized in this example.

Beauty | WPP
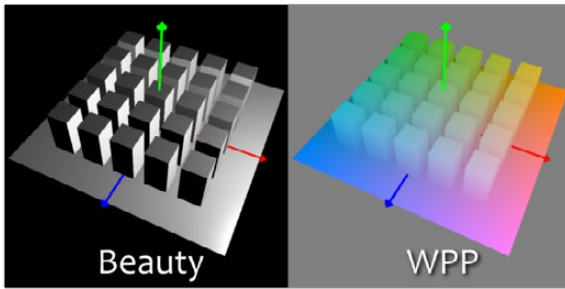
Different Coordinate Spaces

Rendering WPPs can occur in different Coordinate Spaces. These include World Space, Eye Space, and Object Space. The image below depicts how those different spaces look, although the nodes in Fusion require the WPP rendering to occur in World Space.

WorldSpace | EyeSpace | ModelSpace
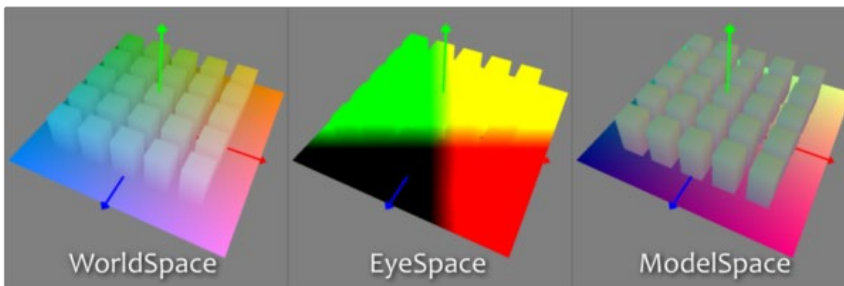
The Scene Input

The nodes offer a Scene input, which can either be a 3D camera or a 3D scene containing a camera. While the camera is vital for the Z to World node, Volume Mask and Volume Fog can generate their output without any camera attached or with the camera position set to 0/0/0.

However, connecting a camera that lines up with the original camera the WPP has been rendered from, or setting the camera's position manually, dramatically improves the accuracy and look of the resulting fog or mask.

Camera | No Camera
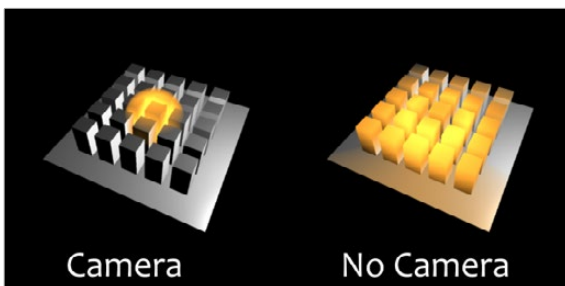
The "Invisible Sphere"

The example scene shown so far has an empty background, meaning there is nothing in the scene apart from the ground plane and the cubes.

If applying fog to a scene like that, which is larger than the ground plane, the result will look similar to the "w/o Sphere" example shown below because, with no WPP information outside the ground plane, the resulting value is 0/0/0, and the fog fills that area as well.

To get around that, you can add an invisible bounding sphere to your scene to create "dummy" WPP values to help the Fog node to create the correct volume as shown in the "with Sphere" example below.

The Common Controls

Nodes that handle Position operations share several identical controls in the Inspector. This section describes controls that are common among Position nodes.

Inspector
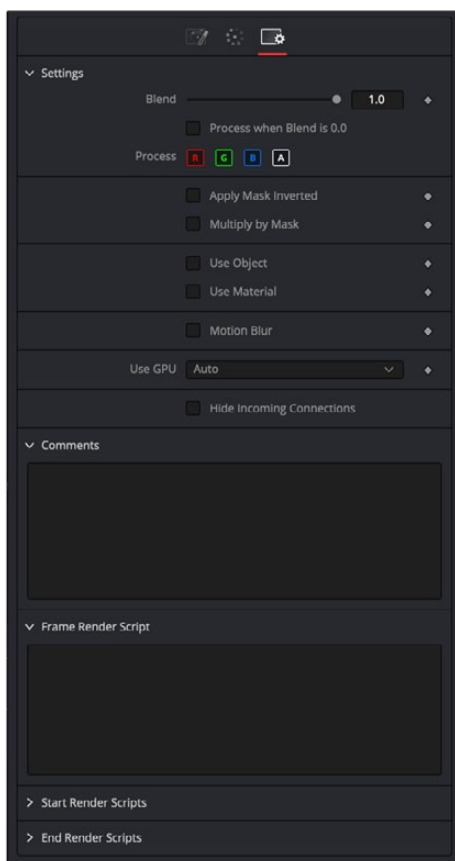
Position nodes Common Settings tab

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Position category. The controls are consistent and work the same way for each tool.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this will cause the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if the node is scripted to trigger a task, but the node's value is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the Red button on a Blur tool is deselected, the blur will first be applied to the image, and then the red channel from the original input will be copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this will generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option will cause the RGB values of the masked image to be multiplied by the mask channel's values. This will cause all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels will be used if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option is disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual

Object ID/Material ID (Sliders)

Use these sliders to select which ID will be used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the viewer. The image or sequence must have been rendered from a 3D software package with those channels included.

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 will cause Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one full frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows for the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off GPU hardware-accelerated rendering. Enabled uses the GPU hardware for rendering the node. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node will be displayed in the Inspector. Dragging a connected node from the node tree into the field will hide that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line will reappear.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Resolve Connect

This chapter details the single node found in the Resolve Connect category, available only in standalone Fusion Studio.

The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

External Matte Saver [EMS]

The External Matte Saver node

> **NOTE:** The Resolve Connect category and External Matte Saver node are available only in Fusion Studio.

External Matte Saver Node Introduction

The External Matte Saver node renders multiple mattes into multiple channels of an EXR file. This file is intended to import into DaVinci Resolve's Color page as an efficient way to deliver multiple mattes for color grading. To perform the same operation with a traditional Saver node, a Channel Boolean is needed to place each matte into a channel, and then name that channel in the Saver. It requires a bit more of a set up. This node streamlines the process by providing multiple inputs and naming the channel based on the node.

Inputs

By default, the node provides a single input for a 2D image you want to save as a matte.

— **Input:** Although initially there is only a single orange input for a matte to connect, the Inspector provides an Add button for adding additional inputs. Each input uses a new color, but all accept 2D RGBA images.

Basic Node Setup

The External Matte Saver node can be placed at the end of any node tree branch for saving mattes. Below, it is connected to the Delta Keyer as well as three other mattes. Each matte beyond the initial one is connected by clicking the Add button in the Inspector first, and then a new input is provided on the node.
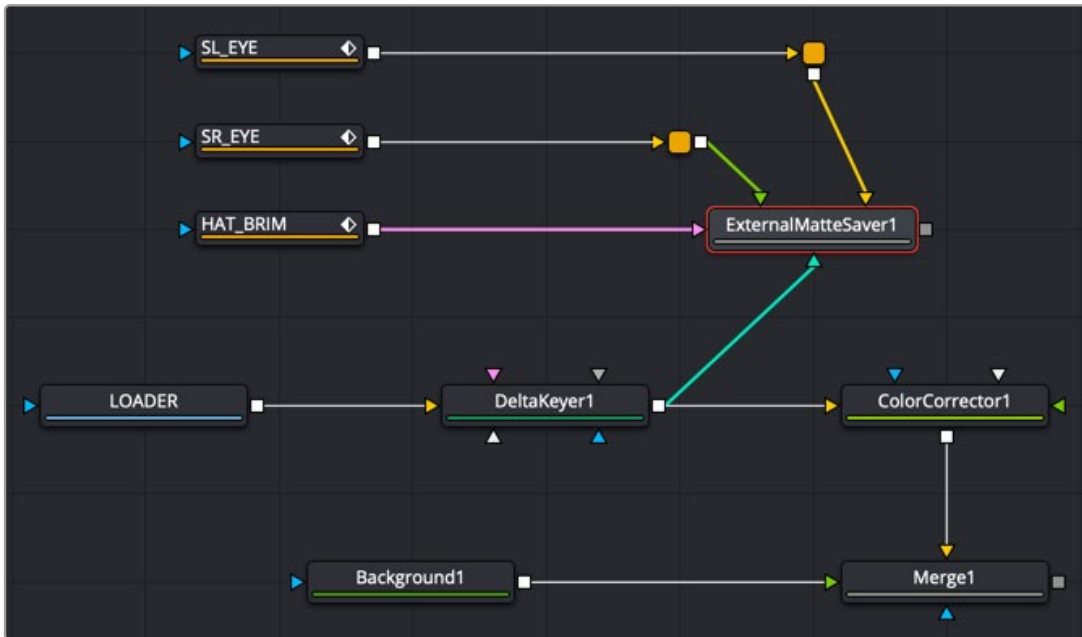
An External Matte Saver node added as a separate branch in a node tree to render the mattes

Inspector
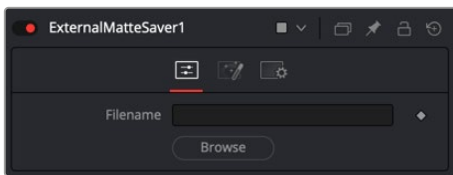
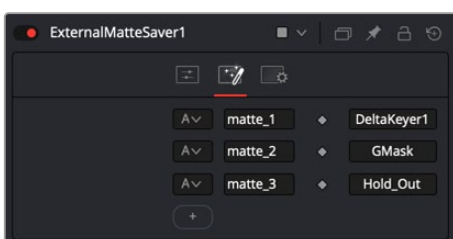

The External Matte Saver Controls tab

Controls Tab

The Controls tab is used to name the saved file and determine where on your hard drive the file
is stored.

Filename

Enter the name you want to use for the EXR file in the Filename field. At the end of the name, append
the .exr extension to ensure that the file is saved as an EXR file.

Browse

Clicking the Browse button opens a standard file browser window where you can select the location to
save the file.

The External Matte Saver Mattes tab

Mattes Tab

The Mattes tab is where you set up the number of mattes saved in the file, the name for each channel, and the RGBA channels saved from each input.

Channels menu

The Channels menu allows you to select which channels are saved in the matte. You can choose the alpha channel, the RGB channels, or the RGBA channels.

Channels Name

The Channels Name field allows you to customize the name of the matte channel you are saving. This name is displayed in DaVinci Resolve's Color page.

Node Name

The Node Name field displays the source of the matte. This is automatically populated when you connect a node to the input.

Add

Clicking the Add button adds an input on the node and another set of fields for you to configure and name the new matte channel.
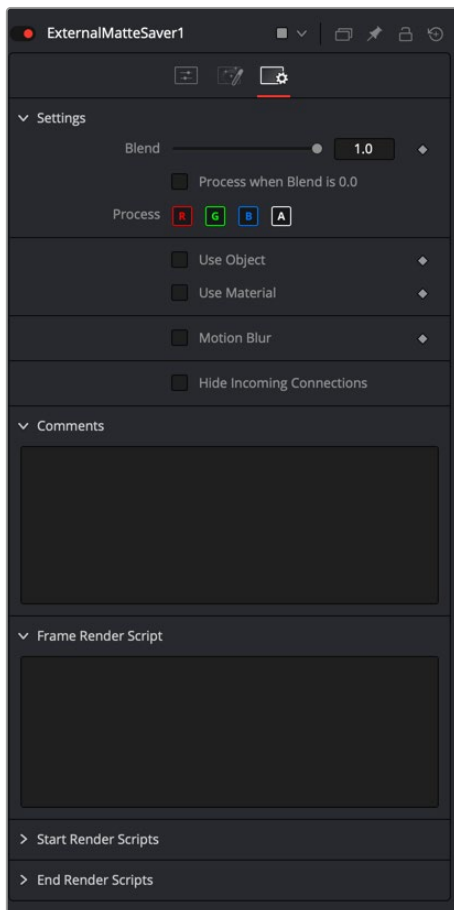
The External Matte Saver Settings tab

Settings Tab

The Settings Tab in the Inspector is similar to settings found in the Saver tool. The controls are consistent and work the same way as the Settings in other tools.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this causes the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if processing of this node is scripted to trigger another task, but the value of the node is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the Red button on a Blur tool is deselected, the blur is first applied to the image, and then the red channel from the original input is copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option causes the RGB values of the masked image to be multiplied by the mask channel's values. This causes all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object ID and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels are used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option is disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID is used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the viewer. The image or sequence must have been rendered from a 3D software package with those channels included.

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 causes Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one full frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows for the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node will be displayed in the Inspector. Dragging a connected node from the node tree into the field will hide that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line will reappear.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Shape Nodes

This chapter details the Shape nodes available in Fusion.

Contents

sBoolean

The sBoolean node

The sBoolean node combines or excludes overlapping areas of two shapes based on a menu of boolean operations.

Like almost all shape nodes, you can only view the sBoolean node's results through a sRender node.

External Inputs

The following inputs appear on the node's tile in the Node Editor. Except when using the subtract boolean operation, which shape you connect into which input does not matter.

— **Input1:** [orange, required] This input accepts the output of another shape node. This input is used as the base shape when the subtract boolean operation is chosen.

— **Input2:** [green, optional] This input accepts the output of another shape node. This input is used to cut the base shape hole when the subtract boolean operation is chosen.

Basic Node Setup

The sBoolean node is used to combine two shape nodes. The output of the sBoolean can then be output to another shape node or a sRender node for viewing.
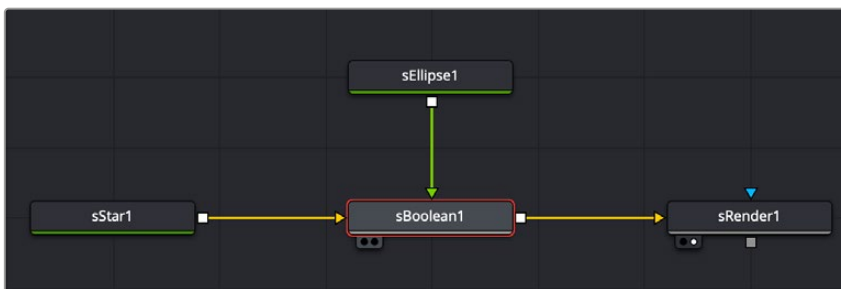
Star and ellipse shapes combined in an sBoolean node, then output to an sRender for viewing and further processing

Inspector
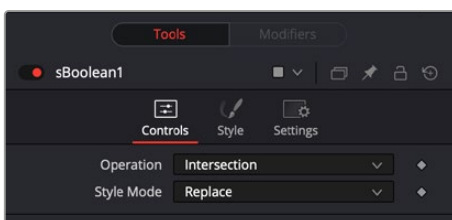
The sBoolean Controls tab

Controls

The Controls tab is primarily used to select the boolean operation that determines how the two shapes are combined.
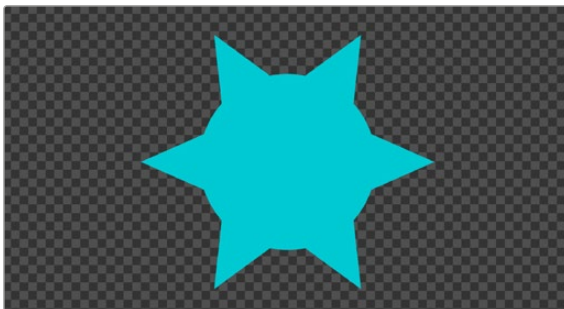
Operation

The operation menu includes four boolean operations:

— **Intersection:** Sometimes called an AND operation, this setting will only show areas where the two shapes overlap. The result is only where input 1 AND input 2 overlap.
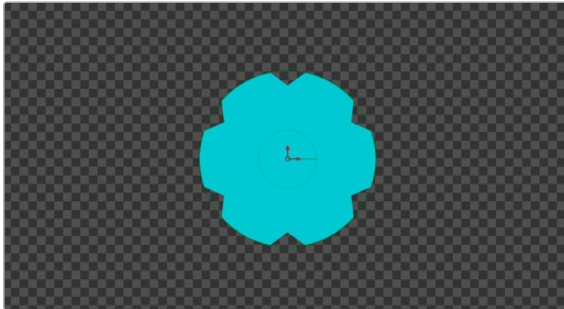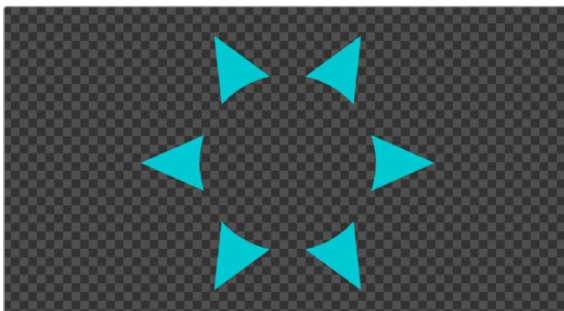

Star and ellipse shapes with an sBoolean
node set to intersection

— **Union:** Sometimes called an OR operation, this setting will only show areas where either of the two shapes exists. The result is where either input 1 OR input 2 exists. The Union setting is similar to the result of the sMerge node.

Star and ellipse shapes with an
sBoolean node set to union

— **Subtract:** Sometimes called a NOT operation, this setting outputs the shape of input 1 but eliminates the areas where input 2 overlaps. The result is input 1 minus input 2.

Star and ellipse shapes with an
sBoolean node set to subtract

— **Xor:** Sometimes called an AND NOT operation, this setting outputs the shape of input 1 or input 2 but eliminates the areas where they overlap. The result is (input 1 minus input 2) + (input 2 minus input 1).
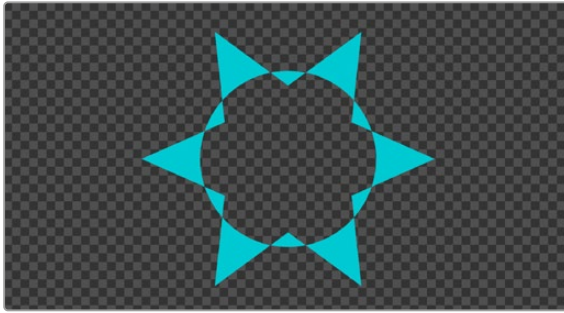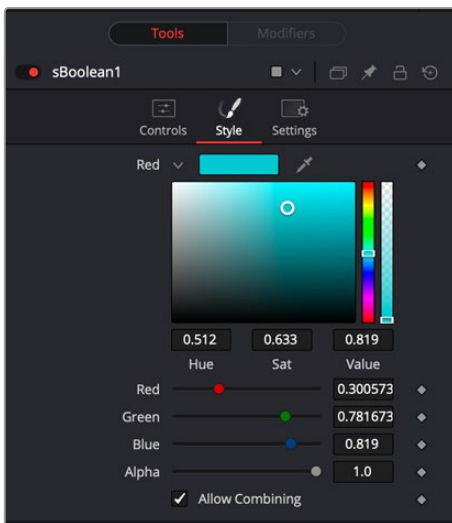

Star and ellipse shapes with an sBoolean node set to xor

Style Mode

The Style mode menu only includes one option. The Replace setting replaces the color and alpha level of the incoming shapes with the color set in the Style tab.

Style Tab

The sBoolean Style tab

Style

Any color assigned to the individual shape nodes is replaced by the color set using the Style tab controls.

Color

The color controls determine the color of the output shape from the sBoolean node. To choose a shape color, you can click the color disclosure arrow, use the color swatch, or drag the eyedropper into the viewer to select a color from an image. The RGBA sliders or number fields can be used to enter each color channel's value or the strength of the alpha channel.

Allow Combining

When this checkbox is enabled, the alpha channel value is maintained even when passing through other nodes downstream that may cause the shape to overlap with copies of itself. When disabled, the alpha channel value may increase when the shape overlaps itself.

For instance, if an ellipse's alpha channel is set to .5, enabling the Allow Combining checkbox maintains that value even if the shape passes through a duplicate or grid node that causes the shape to overlap. Disabling the checkbox causes the alpha channel values to be compounded at each overlapping area. When using the sBoolean node, the individual shape node checkboxes are ignored, and the sBoolean node's checkbox determines the alpha channel's behavior.
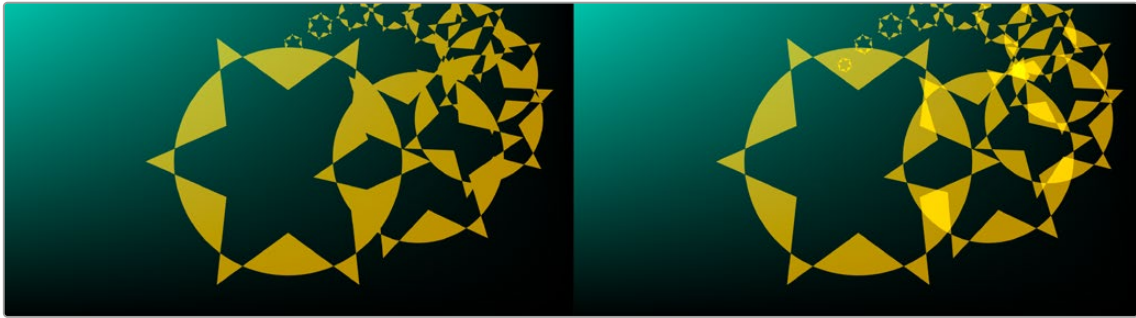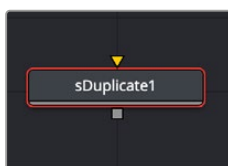

Allow Combining Enabled (left), Allow Combining Disabled (right)

Common Controls

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

sDuplicate

The sDuplicate node

The sDuplicate node creates copies of the input shape, offsetting each copy's position, size, and rotation. Like almost all shape nodes, you can only view the sDuplicate node's results through a sRender node.

External Inputs

The following input appears on the node's tile in the Node Editor:

— **Input1:** [orange, required] This input accepts the output of another Shape node. The shape connected to this input is copied and offset based on the controls in the Inspector.

Basic Node Setup

The sDuplicate node takes a single input that is most often from a single or compound shape from a sMerge or sBoolean node. The sDuplicate node creates copies of the incoming shape and offsets them to create a pattern. The output of the sDuplicate can then be output to another shape node or to a sRender node for viewing or compositing into the greater node tree.

An sEllipse shape connected to an sDuplicate and then output to an sRender for viewing and combining with other elements

Inspector

The sDuplicate Controls tab

Controls

The Controls tab is used to determine the number of copies and set their position, size, and rotation offset.

Copies

This slider determines the number of copies created by the node. The number does not include the original shape, so entering a value of five will produce five copies plus the original.

X and Y Offset

These sliders set the X and Y distance between each of the copies. Each copy is offset from the previous copy by the value entered in the X and Y number fields. The copies all start at 0, the center of the original shape, and are offset from there. Using Fusion's normalized coordinate system, entering X Offset at 0.5 would move each copy half the frame's width to the right. Entering -1.0 would move each copy to the left by the width of the frame.

X and Y Size

Sets the X and Y size offset based on the previous shape size. For instance, an X and Y value of 1.0 creates copies identical in size to the original but. Entering a value of X and Y of 0.5 will cause each copy to be half the size of the copy before it.

Axis Mode

The Axis mode menu provides four options for determining how each copy determines its rotational pivot point.

— **Absolute:** Allows you to set an X and Y position for the axis of rotation based on the original shape's location. The axis of rotation is then copied and offset with each duplicated shape.
— **Origin Relative:** Each copy uses its center point as its axis of rotation.
— **Origin Absolute:** Each copy uses the center of the original shape as its axis of rotation.
— **Progressive:** Compounds each shape copy by progressively transforming each copy based on the previous shape's position, rotation, and scale.

X and Y Pivot

The X and Y pivot controls are displayed when the Axis mode is set to Absolute. You can use these position controls to place the axis of rotation.

Rotation

Determines an offset rotation applied to each copy. The rotation is calculated from the offset rotation of the previous copy. To rotate all copies identically, use the Angle parameter on the original shape or use a sTransform node.

Common Controls

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

sEllipse
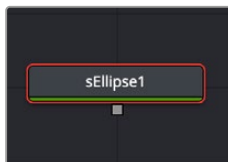
The sEllipse node

The sEllipse node is used to create circular shapes. Like almost all shape nodes, you can only view the sEllipse node's results through a sRender node.

External Inputs

This node generates shapes and does not have any inputs.

Basic Node Setup

The sEllipse node is a shape generator, meaning it generates a shape and therefore has no input. The output of the sEllipse can go into a sRender for viewing and further compositing or, more likely, connect to another shape node like sGrid or sDuplicate.

An sEllipse node connecting to an sGrid node, and then viewed using an sRender node

Inspector
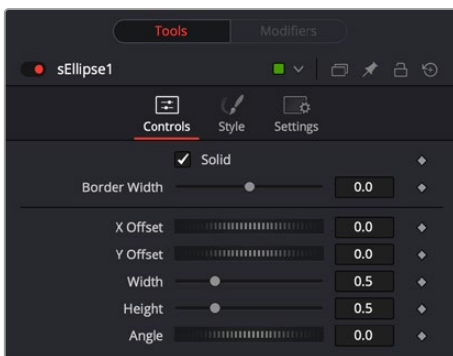
The sEllipse Controls tab

Controls

The Controls tab is used to define the elliptical shape characteristics, including fill, border, size, and position.

Solid

When enabled, the Solid checkbox fills the elliptical shape with the color defined in the Style tab. When disabled, an outline created by the Border Width control is displayed, and the center is made transparent.

Border Width

This parameter expands or contracts the border around the shape. Although it can be used when the Solid checkbox is enabled, it is primarily used to determine the outline thickness when the checkbox is disabled.

Cap style

When the Solid checkbox is disabled, three cap style options are displayed. The cap styles can create lines with flat, rounded, or squared ends. Flat caps have flat, squared ends, while rounded caps have semi-circular ends. Squared caps have projecting ends that extend half the line width beyond the end of the line.

The caps are not visible unless the length is below 1.0.

Position

The position parameter is only displayed when the Solid checkbox is disabled. It allows you to position the starting point of the shape. When used in conjunction with the length parameter, it positions the gap in the ellipse outline.

Length

The length parameter is only displayed when the Solid checkbox is disabled. A length of 1.0 is a closed shape. Setting the length below 1.0 creates an opening or gap in the outline. Keyframing the length parameters allows you to create write-on style animations.

X and Y Offset

These parameters are used to position the shape left, right, up, and down in the frame. The shape starts in the center of the frame, and the parameters are used to offset the position. The offset coordinates are normalized based on the width of the frame. An X offset of 0.0 is centered, and a value of 0.5 places the center of the shape directly on the right edge of the frame.
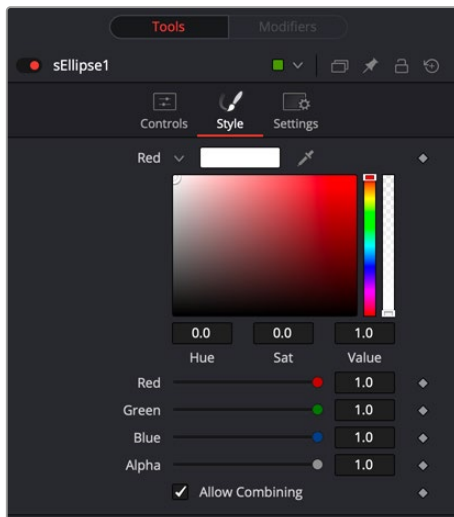
Width/Height

The width and height determine the vertical and horizontal size of the ellipse. If the values are identical, then you have a perfect circle.

Angle

The angle rotates the shape, which on a perfect circle doesn't change the image all that much, but if you create an oval or an outline with a short length, you can rotate the shape based on the center axis.

Style Tab

The sEllipse Style tab

Style

The Style tab is used to assign a color to the shape and control its transparency.
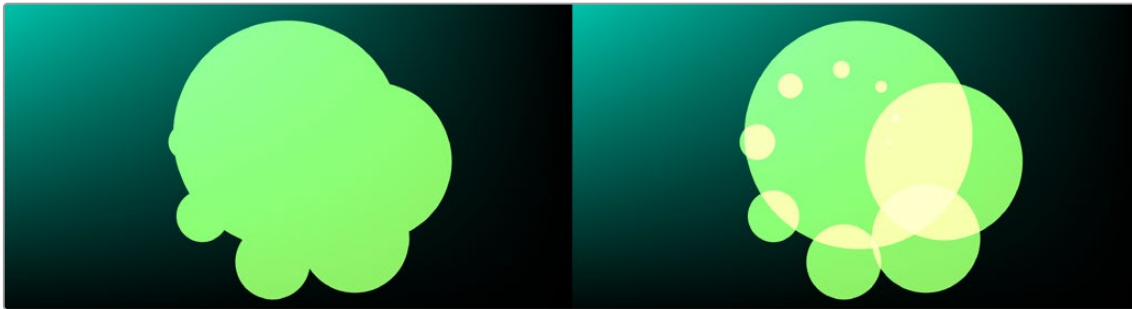
Color

The color controls determine the color of the fill and border. To choose a shape color, you can click the color disclosure arrow, use the color swatch, or drag the eyedropper into the viewer to select a color

from an image. The RGBA sliders or number fields can be used to enter each color channel's value or the strength of the alpha channel.

Allow Combining

When this checkbox is enabled, the alpha channel value is maintained even when passing through other nodes downstream that may cause the shape to overlap with copies of itself. When disabled, the alpha channel value may increase when the shape overlaps itself.

For instance, if an ellipse's alpha channel is set to .5, enabling the Allow Combining checkbox maintains that value even if the shape passes through a Duplicate or Grid node that causes the shape to overlap. Disabling the checkbox causes the alpha channel values to be compounded at each overlapping area.
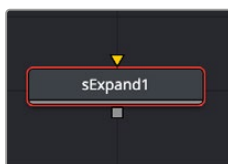

Allow Combining Enabled (left), Allow Combining Disabled (right)

Common Controls

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

sExpand

The sExpand node

The sExpand node is used to dilate or erode shapes. Like almost all Shape nodes, you can only view the sExpand node's results through a sRender node.

External Inputs

The following input appears on the node's tile in the Node Editor.

— **Input1:** [orange, required] This input accepts the output of another shape node. This shape or compound shape connected to this input is either eroded or dilated.

Basic Node Setup

The sExpand node takes a single input that is most often from a compound shape. However, it can be used on single shapes like sStars and sNgons. The output of the sExpand can then be output to another shape node or to a sRender node for viewing or compositing into the greater node tree.
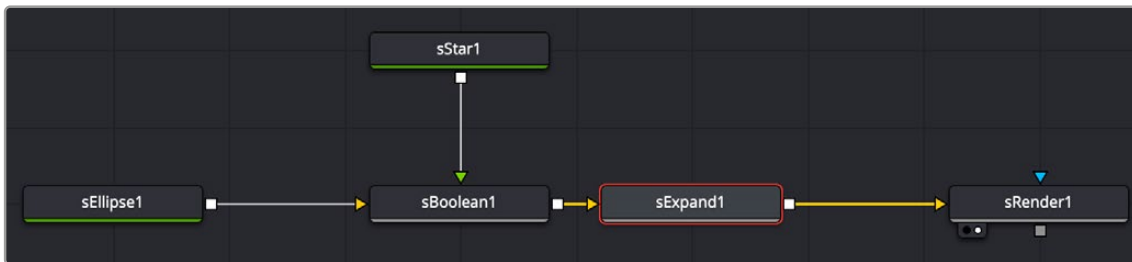
Star and ellipse shapes combined in an sBoolean node, then output to an sExpand for dilating or eroding

Inspector
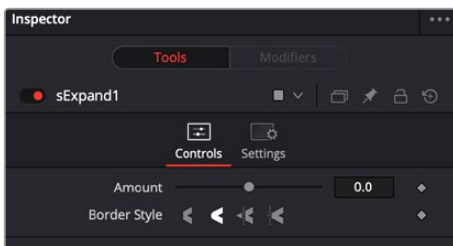
The sExpand Controls tab

Controls

The Controls tab includes all of the parameters for the sExpand node.

Amount

A positive value dilates the shape while a negative value erodes it.

Border Style

The border style controls how the expanded or contracted shapes join at the corners. There are four styles provided as options. Bevel squares off the corners. Round creates rounded corners. Miter and Miter Clip maintain pointed edges, until a certain threshold. The Threshold is set by the Miter limit slider.

Miter Limit

The Miter parameter is only displayed when the Miter or Miter Clip border style is selected. The miter limit determines when the pointed edges become beveled based on the shape's thickness.

Common Controls

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

sGrid
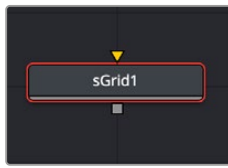
The sGrid node

The sGrid node replicates the shape on an X and Y grid and adds the ability to offset the rows and columns. Like almost all Shape nodes, you can only view the sGrid node's results through a sRender node.

External Inputs

The following input appears on the node's tile in the Node Editor.

— **Input1:** [orange, required] This input accepts the output of another Shape node. The shape connected to this input is replicated on a custom grid.

Basic Node Setup

The sGrid node takes a single input from a single or compound shape from am sMerge or sBoolean node. The sGrid node places the incoming shape on a grid of rows and columns. The output of the sGrid can then be output to another Shape node or a sRender node for viewing or compositing into the greater node tree.

An sEllipse shape connected to an sGrid and then output to an sRender
for viewing and combining with other elements

Inspector
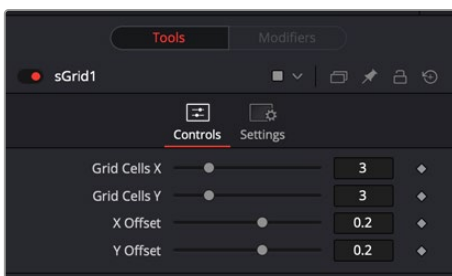
The sGrid Controls tab

Controls

The Controls tab is used to determine the number of grid cells and their offset position.

Grid Cells X and Y

These parameters set the number of cells on the grid, both horizontally and vertically. For instance, entering 5 in the X and Y number field creates five rows of the shape and five columns.

X and Y Offset

Sets the X and Y distance between the rows and columns. An offset value of 0.0 will have all the rows and columns on top of each other. Entering X Offset at 1.0 would spread the columns the width to the frame.

Common Controls

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

sJitter

The sJitter node

The sJitter node is most often used to randomly position an array of shapes generated from a sGrid or sDuplicate node. However, it includes an auto-animating random mode that can be used to distort and randomly jitter single shapes.

Like almost all Shape nodes, you can only view the sJitter node's results through a sRender node.

External Inputs

The following input appears on the node's tile in the Node Editor.

— **Input1:** [orange, required] This input accepts the output of another Shape node. The shape connected to this input is offset, distorted, and animated based on the sJitter node settings.

Basic Node Setup

The sJitter node takes an array of shapes from a sGrid or sDuplicate node and randomly changes their position, size, and rotation. The output of the sExpand can then be output to another Shape node or to a sRender node for viewing or compositing into the greater node tree.
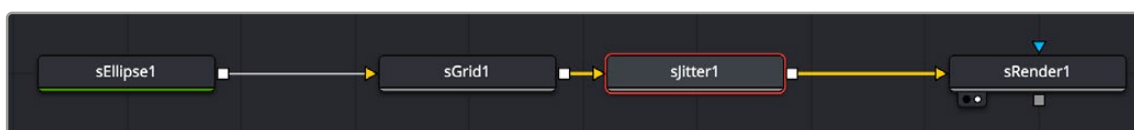
An array of shapes created by the sGrid node input into an sJitter
node to randomly offset or scale the shapes

Inspector
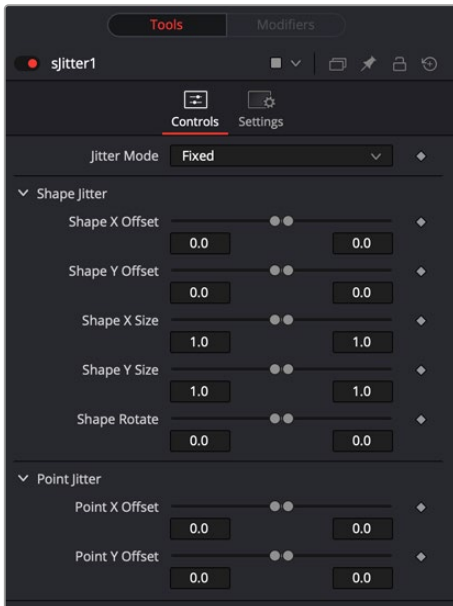
The sJitter Controls tab

Controls

The Controls tab offers range sliders that determine the variation amount for offset, size, and rotation. The Point Jitter parameters are used to offset the invisible points that create the vector shapes.

Jitter Mode

The Jitter Mode menu allows you to choose between static position and size offsets or enabling an auto-animation mode. Leaving the default Fixed selection allows you to offset a grid of shapes, animating with keyframes or modifiers if needed. The Random menu selection auto-animates the parameters based on the range you define using the range sliders. If all the range sliders are left in the default position, no random animation is created. Increasing the range on any given parameter will randomly animate that parameter between the range slider values.

Shape X and Y Offset

These parameters set the horizontal and vertical offset from the shape array's original position. This is done randomly, so not all shapes in the array will offset by the same amount.

Shape X and Y Size

These parameters set the horizontal and vertical scaling for each shape in an array. The left range value decreases the scale, and the right range value increases the scale. This is done randomly, so not all shapes in the array will scale by the same amount.

Shape Rotate

This parameter rotates each shape in an array.

Point Jitter

The X and Y Point Jitter parameters use the vector control points to distort the shape. This can be used to give a distressed appearance to ellipses or wobbly animation to other shapes.

Common Controls

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

sMerge

The sMerge node

The sMerge node combines shapes similar to a standard Merge node, except the sMerge node can accept more than two shape inputs.

Like almost all Shape nodes, you can only view the sMerge node's results through a sRender node.

External Inputs

The node displays only two inputs first, but as each shape node is connected, a new input appears on the node, assuring there is always one free to add a new shape into the composite.

— **Input[#]:** These multi-colored inputs are used to connect multiple Shape node. There is no limit to the number of inputs this node can accept. The node dynamically adds more inputs as needed, ensuring that there is always at least one input available.

Basic Node Setup

The sMerge node is used to combine two Shape nodes. In terms of layering, each subsequent input is placed over the previous input. For instance, the first shape connected to the orange input is the bottom-most shape, the green input is layered over it, and if a third shape is connected to the pink input, that is the topmost layer.
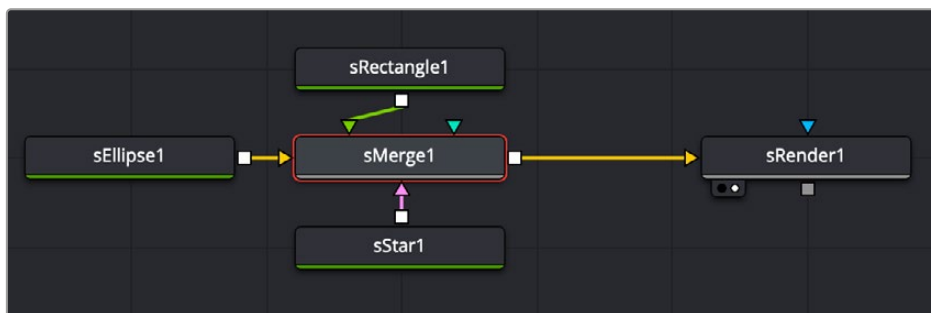
Three shapes are combined in an sMerge node, then output to
an sRender for viewing and further processing.
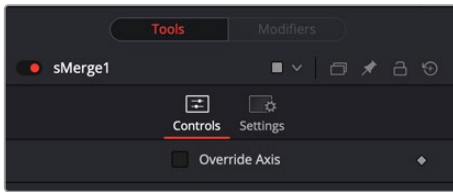
Inspector

The sMerge Controls tab

Controls

The only control for the sMerge node is the Override Axis checkbox, which overrides the shape's axis.

Common Controls

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.
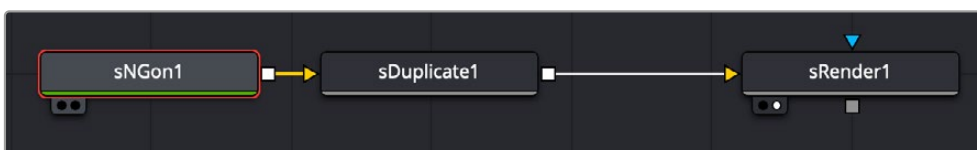
sNGon

The sNGon node

The sNGon node is used to create multi-sided shapes like triangles, pentagons, and octagons. Like almost all Shape nodes, you can only view the sNGon node's results through a sRender node.

External Inputs

This node generates shapes and does not have any inputs.

Basic Node Setup

The sNGon node is a shape generator, meaning it generates a shape and therefore has no inputs. The output of the sNGon can go into a sRender for viewing and further compositing or, more likely, connect to another Shape node like sGrid or sDuplicate.

An sNGon node connecting to an sDuplicate node, and then viewed using an sRender node.

Inspector
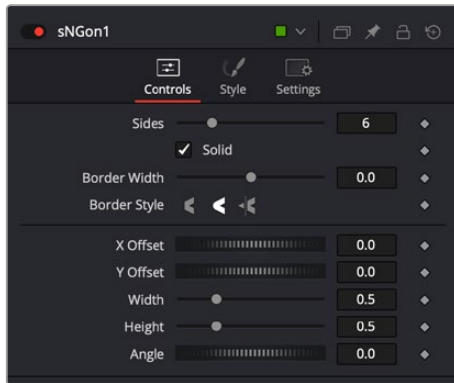
The sNGon Controls tab

Controls

The Controls tab is used to define multi-sided shape characteristics, including fill, border, size, and position.

Solid

When enabled, the Solid checkbox fills the NGon shape with the color defined in the Style tab. When disabled, an outline created by the Border Width control is displayed, and the center is made transparent.

Border Width

This parameter expands or contracts the border around the shape. Although it can be used when the Solid checkbox is enabled, it is primarily used to determine the outline thickness when the checkbox is disabled.

Border Style

The Border Style parameter controls how the sides of the NGon join at the corners. There are three styles provided as options. Bevel squares off the corners. Round creates rounded corners. Miter maintains pointed corners.

Cap style

When the Solid checkbox is disabled, three cap style options are displayed. The cap styles can create lines with flat, rounded, or squared ends. Flat caps have flat, squared ends, while rounded caps have semi-circular ends. Squared caps have projecting ends that extend half the line width beyond the end of the line.

The caps are not visible unless the length is below 1.0.

Position

The Position parameter is only displayed when the Solid checkbox is disabled. It allows you to position the starting point of the shape. When used in conjunction with the Length parameter, it positions the gap in the outline.

Length

The Length parameter is only displayed when the Solid checkbox is disabled. A length of 1.0 is a closed shape. Setting the length below 1.0 creates an opening or gap in the outline. Keyframing the Length parameters allows you to create write-on style animations.

X and Y Offset

These parameters are used to position the shape left, right, up, and down in the frame. The shape starts in the center of the frame, and the parameters are used to offset the position. The offset coordinates are normalized based on the width of the frame. So, an X offset of 0.0 is centered and a value of 0.5 places the center of the shape directly on the right edge of the frame.
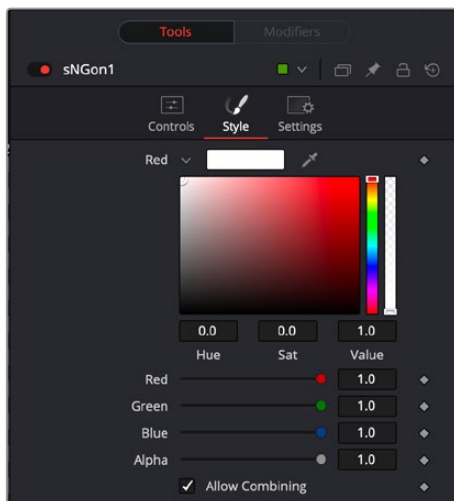
Width/Height

The Width and Height parameters determine the vertical and horizontal size of the ellipse. If the values are identical, then all sides are of equal length.

Angle

The Angle parameter rotates the shape based on the center axis.

Style Tab

The sNGon Style tab

Style

The Style tab is used to assign a color to the shape and control its transparency.

Color

The Color controls determine the color of the fill and border. To choose a shape color, you can click the color disclosure arrow, use the color swatch, or drag the eyedropper into the viewer to select a color from an image. The RGBA sliders or number fields can be used to enter each color channel's value or the strength of the alpha channel.

Allow Combining

When this checkbox is enabled, the alpha channel value is maintained even when passing through other nodes downstream that may cause the shape to overlap with copies of itself. When disabled, the alpha channel value may increase when the shape overlaps itself.

For instance, if a NGon alpha channel is set to .5, enabling the Allow Combining checkbox maintains that value even if the shape passes through a duplicate or grid node that causes the shape to overlap. Disabling the checkbox causes the alpha channel values to be compounded at each overlapping area.
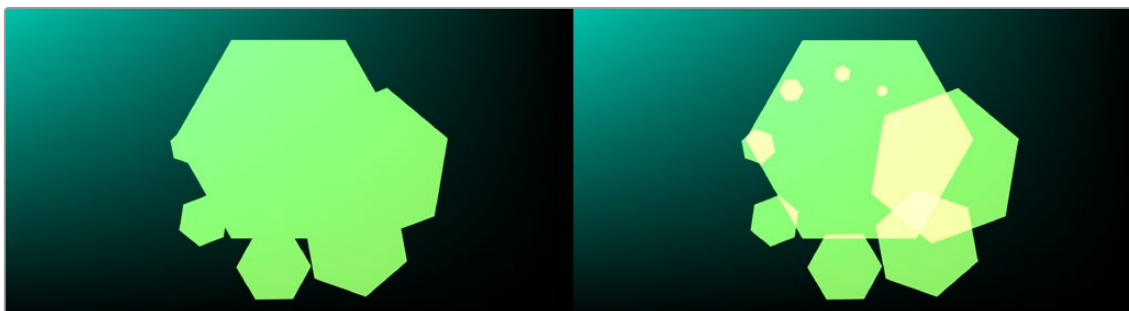
Allow Combining Enabled (left), Allow Combining Disabled (right)

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

sOutline

The sOutline node

The sOutline node is used to create outlines from merged or boolean compound shapes. The individual shapes retain their own style, color, size, position, and other characteristics. The only difference is the border thickness, border style, position, and length are applied to all incoming shapes uniformly in the sOutline node.

Like almost all shape nodes, you can only view the sOutline node's results through a sRender node.

External Inputs

The following input appears on the node's tile in the Node Editor:

— **Input1:** [orange, required] This input accepts the another shape node's output, but more likely a compound shape from a sMerge or sBoolean. An outline is created from the compound shape connected to this input.

Basic Node Setup

The sOutline node takes a single input that is most often from a compound shape, however, it can sometimes be useful on single shapes to create double outlines. The output of the sOutline can then be output to another Shape node or to a sRender node for viewing or compositing into the greater node tree.

A compound shape from an sBoolean node is connected to an sOutline for creating a complex outlined shape

Inspector
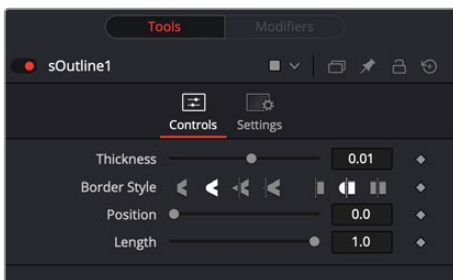
The sOutline Controls tab

Controls

The Controls tab is used to define the outline thickness, border and cap style, position, and length that is applied to the compound shape connected to the input.

Thickness

This parameter controls the width of the outline.

Border Style

The Border Style parameter controls how the outline joins at the corners. There are three styles provided as options. Bevel squares off the corners. Round creates rounded corners. Miter maintains pointed corners.

Cap style

Three Cap Style options are used to create lines with flat, rounded, or squared ends. Flat caps have flat, squared ends, while rounded caps have semi-circular ends. Squared caps have projecting ends that extend half the line width beyond the end of the line.

The caps are not visible unless the length is below 1.0.

Position

The Position parameter allows you to position the starting point of the shape. When used in conjunction with the Length parameter, it positions the gap in the outline.

Length

The Length parameter controls the end position of the outline. A length of 1.0 is a closed shape. Setting the length below 1.0 creates an opening or gap in the outline. Keyframing the Length parameters allows you to create write-on style animations.

Common Controls

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

sRectangle
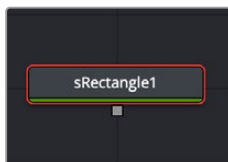
The sRectangle node

The sRectangle node is used to create rectangular shapes. Like almost all shape nodes, you can only view the sRectangle node's results through a sRender node.

External Inputs

This node generates shapes and does not have any inputs.

Basic Node Setup

The sRectangle node is a shape generator, meaning it generates a shape and therefore has no inputs. The output of the sRectangle can go into a sRender for viewing and further compositing or, more likely, connect to another Shape node like sGrid or sDuplicate.
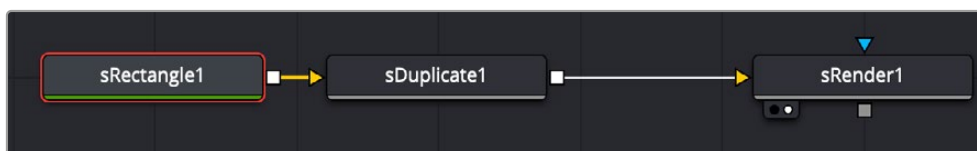
An sRectangle node connecting to an sDuplicate node,
and then viewed using an sRender node

Inspector

The sRectangle Controls tab

Controls

The Controls tab is used to define the rectangle characteristics, including fill, border, size, and position.

Solid

When enabled, the Solid checkbox fills the rectangle shape with the color defined in the Style tab. When disabled, an outline created by the Border Width control is displayed, and the center is made transparent.

Border Width

This parameter expands or contracts the border around the shape. Although it can be used when the Solid checkbox is enabled, it is primarily used to determine the outline thickness when the checkbox is disabled.

Border Style

The Border Style parameter controls how the sides of the rectangle join at the corners. There are three styles provided as options. Bevel squares off the corners. Round creates rounded corners. Miter maintains pointed corners.

Cap style

When the Solid checkbox is disabled, three Cap Style options are displayed. The cap styles can create lines with flat, rounded or squared ends. Flat caps have flat, squared ends, while rounded caps have semi-circular ends. Squared caps have projecting ends that extend half the line width beyond the end of the line.

The caps are not visible unless the length is below 1.0.

Position

The Position parameter is only displayed when the Solid checkbox is disabled. It allows you to position the starting point of the shape. When used in conjunction with the Length parameter, it positions the gap in the outline.

Length

The Length parameter is only displayed when the Solid checkbox is disabled. A length of 1.0 is a closed shape. Setting the length below 1.0 creates an opening or gap in the outline. Keyframing the Length parameters allows you to create write-on style animations.

X and Y Offset

These parameters are used to position the shape left, right, up, and down in the frame. The shape starts in the center of the frame, and the parameters are used to offset the position. The offset coordinates are normalized based on the width of the frame. So an X offset of 0.0 is centered and a value of 0.5 places the center of the shape directly on the right edge of the frame.

Width/Height

The Width and Height parameters determine the vertical and horizontal size of the rectangle. If the values are identical, then you have a square.
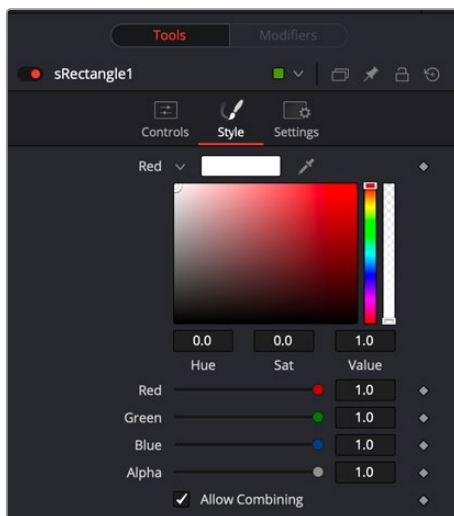
Corner Radius

This parameter determines if the corners of the rectangle are sharp or rounded. A value of 0.0 produces sharp corners, while a value of 1.0 will create a circle from a staring square shape or a pill shape from a rectangle.

Angle

The Angle parameter rotates the shape based on the center axis.

Style Tab

The sRectangle Style tab

Style

The Style tab is used to assign color to the shape and control its transparency.

Color

The Color parameter controls determine the color of the fill and border from the sRectangle node. To choose a shape color, you can click the color disclosure arrow and use the color swatch, or drag the eye dropper into the viewer to select a color from an image. The RGBA sliders or number fields can be used to enter the value of each color channel or the strength of the alpha channel.

Allow Combining

When this checkbox is enabled, the alpha channel value is maintained even when passing through other nodes downstream that may cause the shape to overlap with copies of itself. When disabled, the alpha channel value may increase when the shape overlaps itself. For instance, if a rectangle alpha channel is set to .5, enabling the Allow Combining checkbox maintains that value even if the shape passes through a duplicate or grid node that causes the shape and alpha channel to overlap. Disabling the checkbox causes the alpha channel values to be compounded at each overlapping area.
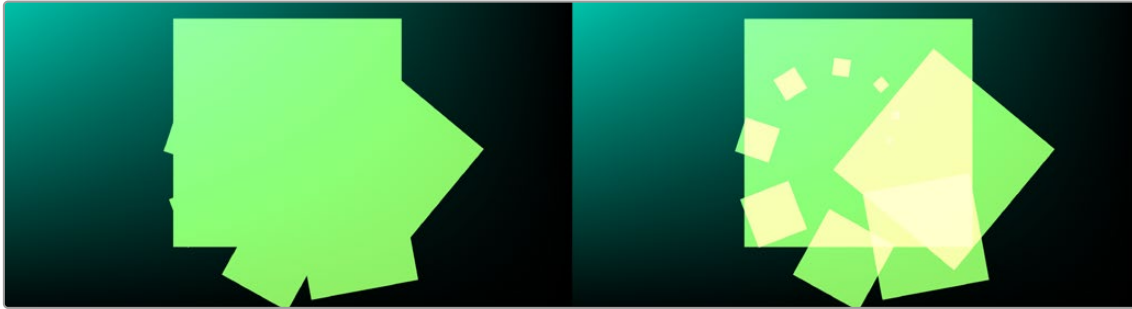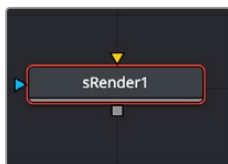

Allow Combining Enabled (left), Allow Combining Disabled (right)

Common Controls

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

sRender

The sRender node

The sRender node converts the vector shapes to an image. The output of the sRender allows the vector shapes to be integrated with other elements in a composite.

Inputs

There is one input on the Background node for an Effect Mask input.

— **Input1:** [orange, required] This input accepts the output of your final shape node. A rendered bitmap image is created from the sRender node for compositing into the rest of your comp.

— **Effect Mask:** The optional blue effect mask input accepts a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the displayed area to only those pixels within the mask.

Basic Node Setup

The sRender node is always placed at the end of a string of Shape nodes. The output connects to other 2D nodes like a Soft Glow node.

Multiple Shape nodes connected to the sRender node and then processed and composited with a title

Inspector

The sRender Image tab

Image Tab

The controls in this tab are used to set the resolution, color depth, and pixel aspect of the image produced by the sRender node.

Process Mode

Use this menu control to select the Fields Processing mode used by Fusion to render the resulting image. The default Full Frames option is appropriate for progressive formats.

Width/Height

This pair of controls are used to set the Width and Height dimensions of the image to be created by the sRender node.

Pixel Aspect

This control is used to specify the Pixel Aspect ratio of the created images. An aspect ratio of 1:1 would generate a square pixel with the same dimensions on either side (like a computer display monitor), and an aspect of 0.9:1 would create a slightly rectangular pixel (like an NTSC monitor).

> **NOTE:** Right-click on the Width, Height, or Pixel Aspect controls to display a menu listing the file formats defined in the preferences Frame Format tab. Selecting any of the listed options will set the width, height, and pixel aspect to the values for that format, accordingly.

Auto Resolution

When this checkbox is selected, the width, height, and pixel aspect of the image created by the node will be locked to values defined in the composition's Frame Format preferences. If the Frame Format preferences change, the resolution of the image produced by the node will change to match. Disabling this option can be useful to build a composition at a different resolution than the eventual target resolution for the final render.

Depth

The Depth button array is used to set the pixel color depth of the image created by the Creator node. 32-bit pixels require 4X the memory of 8-bit pixels but have far greater color accuracy. Float pixels allow high dynamic range values outside the normal 0..1 range, for representing colors that are brighter than white or darker than black.

Source Color Space

You can use the Source Color Space menu to set the Color Space of the footage to help achieve a linear workflow. Unlike the Gamut tool, this doesn't perform any actual color space conversion, but rather adds the source space data into the metadata, if that metadata doesn't exist. The metadata can then be used downstream by a Gamut tool with the From Image option, or in a Saver, if explicit output spaces are defined there. There are two options to choose from:

— **Auto:** Automatically reads and passes on the metadata that may be in the image.
— **Space:** Displays a Color Space Type menu where you can choose the correct color space of the image.

Source Gamma Space

Using the Curve Type menu, you can set the Gamma Space of the footage and choose to remove it by way of the Remove Curve check box when working in a linear workflow. There are three choices in the Curve Type menu

— **Auto:** Automatically reads and passes on the metadata that may be in the image.
— **Space:** Displays a Gamma Space Type menu where you can choose the correct gamma curve of the image.
— **Log:** Brings up the Log/Lin settings, similar to the Cineon tool. For more information, see Chapter 98, "Film Nodes." in the DaVinci Resolve Reference Manual, or Chapter 37 in the Fusion Reference Manual.

Remove Curve

Depending on the selected Gamma Space or on the Gamma Space found in Auto mode, the Gamma Curve is removed from, or a log-lin conversion is performed on, the material, effectively converting it to a linear output space.

Common Controls

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

sStar
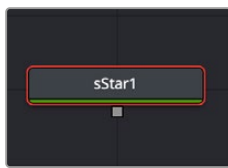
The sStar node

The sStar node is used to create multi-point star shapes. Like almost all Shape nodes, you can only view the sStar node's results through a sRender node.

External Inputs

This node generates shapes and does not have any inputs.

Basic Node Setup

The sStar node is a shape generator, meaning it generates a shape and therefore has no inputs. The output of the sStar can go into a sRender for viewing and further compositing or, more likely, connect to another shape node like sGrid or sDuplicate.
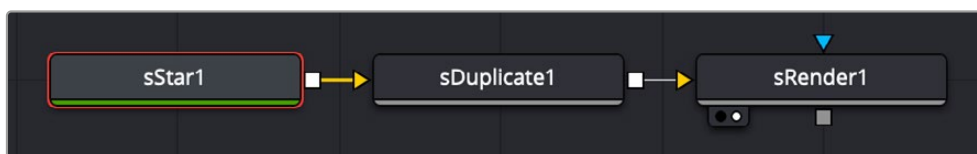
An sStar node connecting to an sDuplicate node, and then viewed using an sRender node

Inspector

The sStar Controls tab

Controls

The Controls tab is used to define the star shape's characteristics, including number of points, depth, fill, border, size, and position.
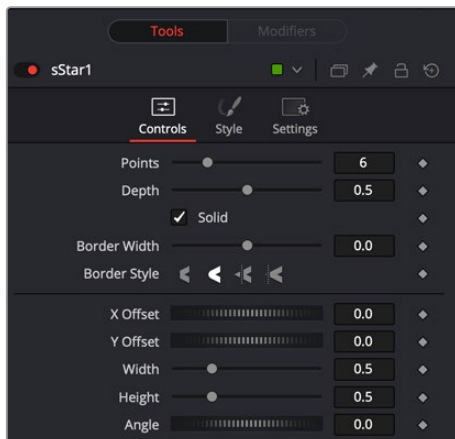
Points

This slider determines the number of points or arms on the star.

Depth

The depth slider controls the inner radius or width of the arms. A depth of 0.001 makes hair-thin arms, while a depth of 1.0 makes a faceted circle.

Solid

When enabled, the Solid checkbox fills the star shape with the color defined in the Style tab. When disabled, an outline created by the Border Width control is displayed, and the center is made transparent.

Border Width

This parameter expands or contracts the border around the shape. Although it can be used when the Solid checkbox is enabled, it is primarily used to determine the outline thickness when the checkbox is disabled.

Border Style

The Border Style parameter controls how the sides of the star join at the corners. There are three styles provided as options. Bevel squares off the corners. Round creates rounded corners. Miter maintains pointed corners.

Cap style

When the Solid checkbox is disabled, three cap style options are displayed. The cap styles can create lines with flat, rounded or squared ends. Flat caps have flat, squared ends while rounded caps have semi-circular ends. Squared caps have projecting ends that extend half the line width beyond the end of the line.

The caps are not visible unless the length is below 1.0.

Position

The Position parameter is only displayed when the Solid checkbox is disabled. It allows you to position the starting point of the shape. When used in conjunction with the length parameter, it positions the gap in the outline.

Length

The Length parameter is only displayed when the Solid checkbox is disabled. A length of 1.0 is a closed shape. Setting the length below 1.0 creates an opening or gap in the outline. Keyframing the Length parameters allows you to create write-on style animations.

X and Y Offset

These parameters are used to position the shape left, right, up, and down in the frame. The shape starts in the center of the frame, and the parameters are used to offset the position. The offset coordinates are normalized based on the width of the frame. So an X offset of 0.0 is centered and a value of 0.5 places the center of the shape directly on the right edge of the frame.

Width/Height

The Width and Height parameters determine the vertical and horizontal size of the star. If the values are identical, then all arms of the star are of equal length.

Angle

The Angle parameter rotates the shape based on the center axis.
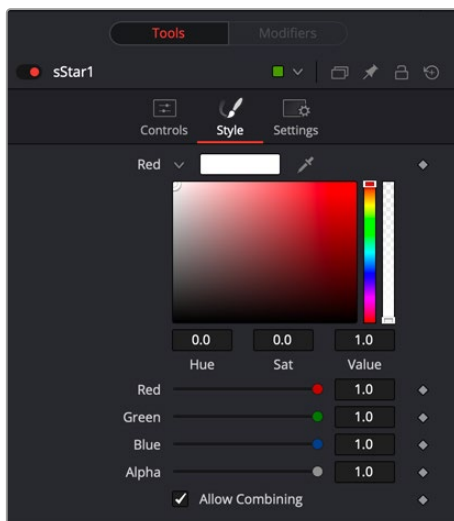
Style Tab

The sStar Style tab

Style

The Style tab is used to assign color to the shape and control its transparency.

Color

The Color controls determine the color of the fill and border from the sStar node. To choose a shape color, you can click the color disclosure arrow and use the color swatch, or drag the eye dropper into the viewer to select a color from an image. The RGBA sliders or number fields can be used to enter the value of each color channel or the strength of the alpha channel.

Allow Combining

When this checkbox is enabled, the alpha channel value is maintained even when passing through other nodes downstream that may cause the shape to overlap with copies of itself. When disabled, the alpha channel value may increase when the shape overlaps itself. For instance, if a star alpha channel is set to .5, enabling the Allow Combining checkbox maintains that value even if the shape passes through a duplicate or grid node that causes the shape and alpha channel to overlap. Disabling the checkbox causes the alpha channel values to be compounded at each overlapping area.

Allow Combining Enabled (left), Allow Combining Disabled (right)

Common Controls

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

sTransform

The sTransform node

The sTransform node is used to add an additional set of transform controls to the existing controls that are contained in Shape nodes. These additional transforms can be used to create hierarchical animations. For instance, you can use a sStar's built-in Angle control to spin the star around. The star can then be output to an sTransform node. The rotation control in the sTransform can be used to orbit the star around the frame.

Like almost all Shape nodes, you can only view the sStar node's results through a sRender node.

External Inputs

The following input appears on the node's tile in the Node Editor:

— **Input1:** [orange, required] This input accepts the output of another Shape node. The shape connected to this input is moved, scaled, and rotated based on the sTransform settings.
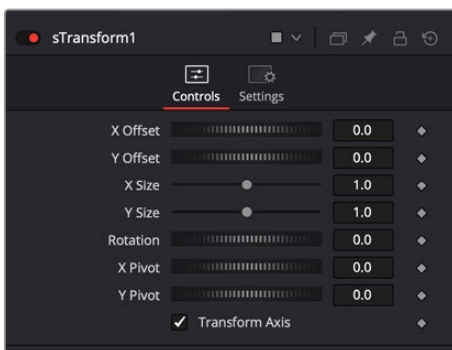
Basic Node Setup

The sTransform node takes the input from another Shape node and adds another set of transforms or hierarchical animation. The output of the sTransform can go into a sRender for viewing and further compositing.
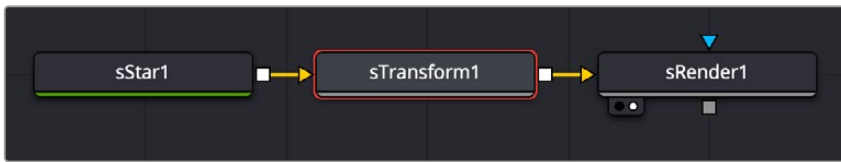
An sStar node connecting to an sTransform node, and then viewed using an sRender node

Inspector

The sTransform Controls tab

Controls

The Controls tab is used to define the add a set of transform controls to the incoming shape.

X and Y Offset

These parameters are used to position the shape left, right, up, and down in the frame. The shape starts in the center of the frame, and the parameters are used to offset the position. The offset coordinates are normalized based on the width of the frame. So an X offset of 0.0 is centered and a value of 0.5 places the center of the shape directly on the right edge of the frame.

X and Y Size

The X and Y Size determine the vertical and horizontal scaling of the incoming shape. If the values are different then the shape will be skewed from its original design.

Rotation

The dial rotates the shape based on the pivot controls.

X and Y Pivot

These parameters position the axis of rotation for the incoming shape. The pivot point is visible in the viewer as a red X. The X can be dragged in the viewer for positioning.

Transform Axis

Check this box to apply the transform to the shape's axis.

Common Controls

Settings tab

The Settings tab in the Inspector is common to all Shape nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Common Controls

Nodes that handle Shape operations share a number of identical controls in the Inspector. This section describes controls that are common amongst Shape nodes.

Settings Tab

The Settings tab in the Inspector can be found on every Shape node. Most of the controls listed here are only found in the sRender node but a few are common to all Shape nodes.

Blend (sRender only)

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming. Normally, this causes the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0 (sRender only)

The tool is processed even when the input value is zero. This can be useful if processing of this node is scripted to trigger another task but the value of the node is set to 0.0.

Red/Green/Blue/Alpha Channel Selector (sRender only)

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the red button on a Blur tool is deselected, the blur is first applied to the image, then the red channel from the original input is copied back over the red channel of the result.

There are some exceptions, such as tools where deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Control tabs are identical.

Apply Mask Inverted (sRender only)

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply By Mask (sRender only)

Selecting this option causes the RGB values of the masked image to be multiplied by the mask channel's values. This causes all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Motion Blur (sRender only)

— **Motion Blur:** This toggles the rendering of motion blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 causes Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one whole frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows for the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Use GPU (sRender only)

The user GPU menu has three settings. Setting the menu to Disable turns off hard accelerated rendering using the graphics card in your computer. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node will be displayed in the Inspector. Dragging a connected node from the node tree into the empty field will hide that incoming connection line as long as the node is not selected in the node tree.. When the node is selected in the node tree the line will reappear.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower left corner of the node when the full tile is displayed or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more information on scripting nodes, see the Fusion scripting documentation.

Chapter 116

Stereo Nodes

This chapter details the Stereo nodes available in Fusion. Stereoscopic nodes are available only in Fusion Studio and DaVinci Resolve Studio.

> The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.
>
> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Anaglyph [Ana]

The Anaglyph node

> **NOTE:** The Anaglyph node is available only in Fusion Studio and DaVinci Resolve Studio.

Anaglyph Node Introduction

The Anaglyph node is used to create stereoscopic images by combining separate left eye and right eye images. It is most commonly used at the end of a stereoscopic workflow to display or deliver the final result.

Inputs

The three inputs on the Anaglyph node are the left eye input, right eye input, and effect mask.

— **Left Eye Input:** The orange input is used to connect the 2D image representing the left eye in the stereo comp.

— **Right Eye Input:** The green input is used to connect the 2D image representing the right eye in the stereo comp.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the stereoscopic creation to only those pixels within the mask.

Basic Node Setup

The Anaglyph node is usually placed at the end of a stereoscopic node tree to display the final result.

When using separate images for the left and right eye, the left eye image is connected to the orange input, and the right eye image is connected to the green input of the node. When using either horizontally or vertically stacked images containing both left eye and right eye information, these only connect to the orange input.

An Anaglyph node using separate left and right eye inputs

Inspector
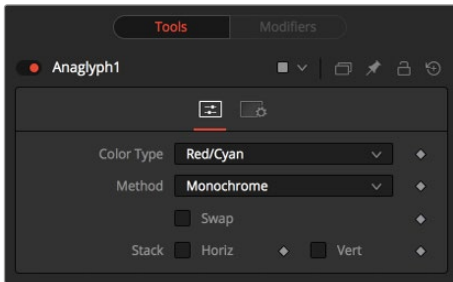
The Anaglyph Controls tab

Controls Tab

Using the parameters in the Controls tab, the separate images are combined to create a stereoscopic output.

Color Type Menu

The Color Type menu allows you to choose between different color encodings to fit your preferred display device. To match your stereo glasses, you can choose between Red/Cyan, Red/Green, Red/Blue, Amber/Blue, and Green/Magenta encoding; Red/Cyan is the most commonly used.
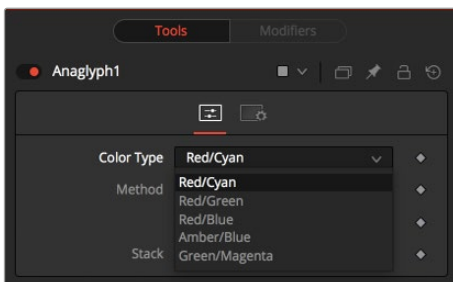
The Anaglyph Color Type menu

Method

In addition to the color used for encoding the image, you can also choose five different methods from the Method menu: Monochrome, Half-color, Color, Optimized, and Dubois. These methods are described below.
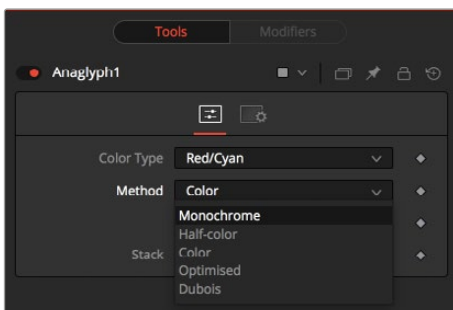
The Anaglyph Method menu

— **Monochrome:** Assuming you are using a Red/Cyan Color Type, the left eye contains the luminance of the left image and is placed in the output of the red channel. The right eye contains the luminance of the right image and is placed in the output green and blue channels.

— **Half-Color:** Assuming you are using a Red/Cyan Color Type, the left eye contains the luminance of the left image and is placed in the output of the red channel. The right eye contains the color channels from the right image that match the glasses' color for that eye.
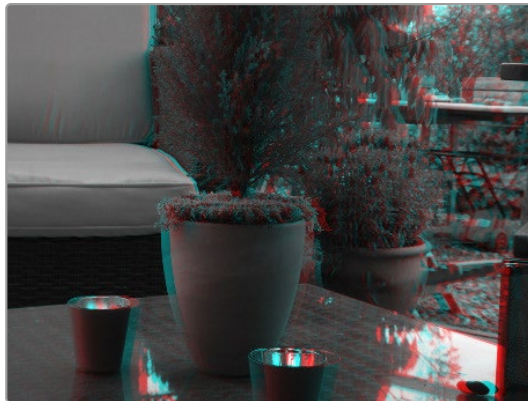

Monochrome

Half-Color

— **Color:** The left eye contains the color channels from the left image that match the glasses' color for that eye. The right eye contains the color channels from the right image that match the glasses' color for that eye.

— **Optimized:** Used with red/cyan glasses, for example, the resulting brightness of what shows through the left eye is substantially less than the brightness of the right eye. Using typical ITU-R 601 ratios for luminance as a guide, the red eye would give 0.299 brightness, while the cyan eye would give 0.587+0.114=0.701 brightness—over twice as bright. The difference in brightness between the eyes can produce what are referred to as retinal rivalry or binocular rivalry, which can destroy the stereo effect. The Optimized method generates the right eye in the same fashion as the Color method. The left eye also uses the green and blue channels but in combination with increased brightness that reduces retinal rivalry. Since it uses the same two channels from each of the source images, it doesn't reproduce the remaining one. For example, 1.05× the green and 0.45× the blue channels of the left image is placed in the red output channel, and the green and blue channels of the right image are placed in the output green and blue channels. Red from both the left and right images is not used.
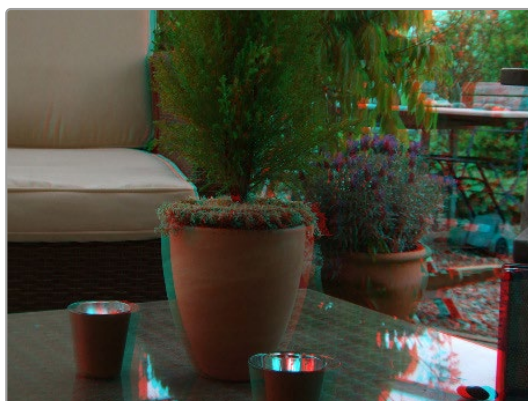
Color

Optimized

— **Dubois:** Images with fairly saturated colors can produce retinal rivalry with the Half-color, Color, and Optimized methods because the color is visible in only one eye. For example, with red/cyan glasses, a saturated green object looks black in the red eye, and green in the cyan eye. The Dubois method uses the spectral characteristics of (specifically) red/cyan glasses and CRT (Trinitron) phosphors to produce a better anaglyph and in the end, tends to reduce retinal rivalry caused by such color differences in each eye. It also tends to reduce ghosting produced when one eye 'leaks' into the other eye. The particular calculated matrix in Fusion is designed for red/cyan glasses and isn't available for other glasses types. Since it is also derived from CRT color primaries, it may not give the best results with a common LCD (though it'll still likely produce less retinal rivalry and ghosting than the other methods).

Dubois

Swap Eyes

Allows you to swap the left and right eye inputs easily.

Horiz Stack

Takes an image that contains both left and right eye information stacked horizontally. These images are often referred to as "crosseyed" or "straight stereo" images. You only need to connect that one image to the orange input of the node. It then creates an image half the width of the original input, using the left half of the original image for the left eye and the right half of the original image for the right eye. Color encoding takes place using the specified color type and method.

Vert Stack

Takes an image that contains both left and right eye information stacked vertically. You only need to connect that one image to the orange input of the node. It then creates an image half the height of the original input, using the bottom half of the original image for the left eye and the top half of the original image for the right eye. Color encoding takes place using the specified color type and method.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Stereo nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Combiner [Com]

The Combiner node

> **NOTE:** The Combiner node is available only in Fusion Studio and DaVinci Resolve Studio.

Combiner Node Introduction

The Combiner node takes two stereoscopic inputs and creates so-called "stacked images" with the left and right eye, either side by side or on top of each other. Stereoscopic nodes are available only in Fusion Studio and DaVinci Resolve Studio.

Inputs

The two inputs on the Combiner node are used to connect the two images that get combined in a stacked stereo image.

— **Image 1 Input:** The orange input is used to connect the 2D image representing the left eye in the stereo comp.

— **Image 2 Input:** The green input is used to connect the 2D image representing the right eye in the stereo comp.
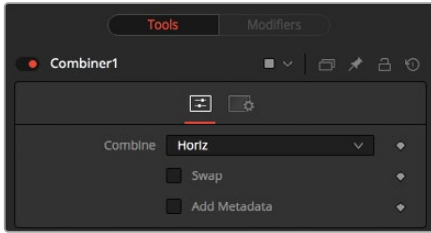
Basic Node Setup

Below, a left eye image and right eye image are connected to the Combiner node to create a single stacked stereo image. It can be more efficient to render out the stacked stereo images as EXR files than to generate the disparity on-the-fly.

Left and right eye images are connected into a Combiner node to generate a stacked stereo image.

Inspector

The Combiner Controls tab

Controls Tab

To stack the images, the left eye image is connected to the orange input, and the right eye image is connected to the green input of the node.

Combine

The Combine menu provides three options for how the two images are made into a stacked stereo image.

— **None:** No operation will take place. The output image is identical to the left eye input.

— **Horiz:** Both images will be stacked horizontally, or side-by-side, with the image connected to the left eye input on the left. This will result in an output image double the width of the input image.

— **Vert:** Both images will be stacked vertically, or on top of each other, with the image connected to the left eye input on the bottom. This will result in an output image double the height of the input image.

Swap Eyes

Allows you to easily swap the left and right eye input.

Add Metadata

Metadata is carried along with the images and can be added to the existing metadata using this checkbox. To view Metadata, use the viewer's SubView menu set to Metadata.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Stereo nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Disparity [Dis]

The Disparity node

> **NOTE:** The Disparity node is available only in Fusion Studio and DaVinci Resolve Studio.

Disparity Node Introduction

Disparity generates the left/right shift between the frames in a stereo pair. It also generates the vertical disparity between the left/right images, which is usually a lot smaller than the horizontal disparity and ideally should be 0 to minimize viewing discomfort. When viewing the output of the Disparity node in the views, the human eye can distinguish quality/detail in the Disparity map better by looking at either the grayscale X disparity or Y disparity rather than looking at the combined XY disparity as a Red/Green color image.

The generated disparity is stored in the output image's Disparity aux channel, where the left image contains the left > right disparity, and the right image contains the right > left disparity. Because disparity works based on matching regions in the left eye to regions in the right eye by comparing colors and gradients of colors, colors in the two eyes must be as similar as possible. Thus, it is a good idea to color correct ahead of time. It is also a good idea to crop away any black borders around the frames, as this confuses the disparity tracking (and also causes problems if you are using the Color Corrector's histogram match ability to do the color matching).

In Stack mode, left and right outputs deliver the same image. If the left and right images have a global vertical offset larger than a few pixels, it can help the disparity tracking algorithm if you vertically align features in the left/right eyes ahead of time using a Transform node. Small details tend to get lost in the tracking process when you have a large vertical offset between left/right eyes.

Consider using a SmoothMotion node to smooth your disparity channel. This can help reduce time-dependent flickering when warping an eye. Also, think about whether you want to remove lens distortion before computing disparity. If you do not, your Disparity map becomes a combined Disparity and Lens Distortion map. This can have advantages and disadvantages.

One disadvantage is that if you then do a vertical alignment, you are also removing lens distortion effects. When trying to reduce the computation time, start first with adjusting the Proxy and Number of Iterations sliders.

The Disparity node does not support RoI or DoD.

Inputs

The two inputs on the Disparity node are used to connect the left and right images.

— **Left Input:** The orange input is used to connect either the left eye image or the stacked image.

— **Right Input:** The green input is used to connect the right eye image. This input is available only when the Stack Mode menu is set to Separate.

Outputs

Unlike most nodes in Fusion, Disparity has two outputs for the left and right eye.

Left Output: This holds the left eye image with a new disparity channel, or a Stacked Mode image with a new disparity channel.

Right Output: This holds the right eye image with a new disparity channel. This output is visible only if Stack Mode is set to Separate.

Basic Node Setup

Below, a left eye image and right eye image are connected to the Disparity node. The Disparity node then outputs each eye to Saver nodes. It can be more efficient to render out the stereo images as EXR files than to generate the disparity on-the-fly.
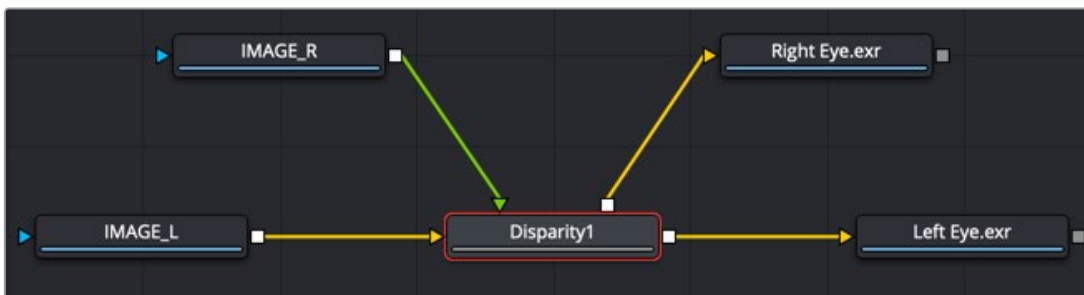
Left and right eye images are connected into a Disparity node to generate and render out a stereo image

Inspector

The Disparity Controls tab

Proxy (for Tracking)

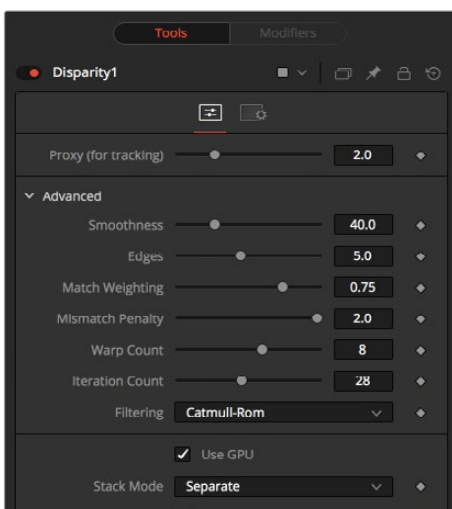The input images are resized down by the proxy scale, tracked to produce the disparity, and then the resulting disparities are scaled back up. This option is purely to speed up the calculation of the disparity, which can be slow. The computational time is roughly proportional to the number of pixels in the image. This means a proxy scale of 2 gives a 4x speedup, and a proxy scale of 3 gives a 9x speedup. In general, 1:1 proxy will give the most detailed flow, but keep in mind that this is highly dependent on the amount of noise and film grain. If noise is present in large quantities, it can completely obliterate any gains moving from 2:1 to 1:1 proxy. In some situations, it can even make things worse. You can think of the Proxy setting as acting as a simplistic low-pass filter for removing noise/grain.

Advanced

The Advanced settings section has parameter controls to tune the Disparity map calculations. The default settings have been chosen to be the best default values from experimentation with many different shots and should serve as a good standard. In most cases, tweaking of the Advanced settings is not needed.

Smoothness

This controls the smoothness of the disparity. Higher values help deal with noise, while lower values bring out more detail.

Edges

This slider is another control for smoothness but applies it based on the color channel. It tends to have the effect of determining how edges in the disparity follow edges in the color images. When it is set to a lower value, the disparity becomes smoother and tends to overshoot edges. When it is set to a higher value, edges in the disparity align more tightly with the edges in the color images, and details from the color channels start to slip into the disparity, which is not usually desirable.

As a rough guideline, if you are using the disparity to produce a Z channel for post effects like depth of field, experiment with higher values, but if you are using the disparity to do interpolation, you might want to keep the values lower.

In general, if the Edges slider is set is too high, there can be problems with streaked out edges when the disparity is used for interpolation.

Match Weight

This controls how matching is done between neighboring pixels in the left image and neighboring pixels in the right image. When a lower value is used, large structural color features are matched. When higher values are used, small sharp variations in the color are matched. Typically, a good value for this slider is in the [0.7, 0.9] range. Setting this option higher tends to improve the matching results in the presence of differences due to smoothly varying shadows or local lighting variations between the left and right images. You should still color match the initial images so they are as similar as possible; this option tends to help with local variations (e.g., lighting differences due to light passing through a mirror rig).

Mismatch Penalty

This controls how the penalty for mismatched regions grows as they become more dissimilar. The slider provides a choice between a balance of Quadratic (lower values) and Linear (higher values) penalties. Lower value Quadratic settings strongly penalize large dissimilarities, while higher value Linear settings are more robust to dissimilar matches. Moving this slider toward lower tends to give a disparity with more small random variations in it, while higher values produce smoother, more visually pleasing results.

Warp Count

Turning down the Warp Count makes the disparity computations faster. In particular, the computational time depends linearly upon this option. To understand what this option does, you need to understand that the Disparity algorithm progressively warps the left image until it matches with the right image. After some point, convergence is reached, and additional warps are just a waste of computational time. The default value in Fusion is set high enough that convergence should always be reached. You can tweak this value to speed up the computations, but it is good to watch how the disparity is degrading in quality at the same time.

Iteration Count

Turning down the Iteration Count makes the disparity computations faster. In particular, the computational time depends linearly upon this option. Just like adjusting Warp Count, at some point adjusting this option higher will yield diminishing returns and will not produce significantly better results. By default, this value is set to something that should converge for all possible shots and can be tweaked lower fairly often without reducing the disparity's quality.

Filtering

This menu determines the filtering operations used during flow generation. Catmull-Rom filtering will produce better results, but at the same time, it increases the computation time steeply.

Stack Mode

This menu determines how the input images are stacked.

When set to Separate, the Right Input and Output will appear, and separate left and right images must be connected.

Swap Eyes

Enabling this checkbox causes the left and right images to swap.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Stereo nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Disparity To Z [D2Z]

The DisparityToZ node

> **NOTE:** The Disparity to Z node is available only in Fusion Studio and DaVinci Resolve Studio.

Disparity to Z Node Introduction

Disparity To Z takes a 3D camera and an image containing a disparity channel as inputs, and outputs the same image but with a newly computed Z channel.

Optionally, this node can output Z into the RGB channels. Ideally, either a stereo Camera 3D or a tracked stereo camera is connected into Disparity To Z. However, if no camera is connected, the node provides artistic controls for determining a Z channel. The depth created by this node can be used for post effects like fogging or depth of field (DoF).

The Z values produced become more incorrect the larger (negative) they get. The reason is that disparity approaches a constant value as Z approaches -infinity. So Z = -1000 and Z = -10000 and Z = -100000 may map to D=142. 4563 and D=142. 4712 and D=142. 4713. As you can see, there is only 0.0001 in D to distinguish between 10,000 and 100,000 in Z. The maps produced by disparity are not accurate enough to make distinctions like this.

Inputs

The three inputs on the Disparity To Z node are used to connect the left and right images and a camera node.

— **Left Input:** The orange input is used to connect either the left eye image or the stack image.

— **Right Input:** The green input is used to connect the right eye image. This input is available only when the Stack Mode menu is set to Separate.

— **Stereo Camera:** The magenta input is used to connect a stereo camera node.

Outputs

Unlike most nodes in Fusion, Disparity To Z has two outputs for the left and right eye.

— **Left Output:** This holds the left eye image with a new Z channel, or a Stacked Mode image with a new disparity channel.

— **Right Output:** This holds the right eye image with a new Z channel. This output is visible only if Stack Mode is set to Separate.
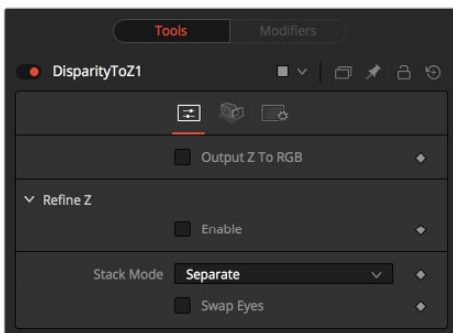
Basic Node Setup

Disparity To Z takes a 3D camera and stereo images containing a disparity channel as inputs. The output is an image with a newly computed Z channel.

A Disparity to Z node creates an image with a Z channel.

Inspector

The Disparity To Z Controls tab

Controls Tab

In addition to outputting Z values in the Z channel, this tab promotes the color channels to float32 and outputs the Z values into the color channels as {z, z, z, 1}. This option is useful to get a quick look at the Z channel.

> **NOTE:** Z values are negative, becoming more negative the further you are from the camera. The viewers only show 0.0 to 1.0 color, so to visualize other data it has to be converted via a normalization method to fit in a display 0-1 range. To do this, right-click in the viewer and choose Options > Show Full Color Range.

Output Z to RGB

Rather than keeping the Z values within the associated aux channel only, they will be copied into the RGB channels for further modification with any of Fusion's nodes.

Refine Z

The Enable checkbox refines the depth map based upon the RGB channels. The refinement causes edges in the flow to align more closely with edges in the color channels. The downside is that unwanted details in the color channels start to show up in the flow. You may want to experiment with using this option to soften out harsh edges for Z-channel post effects like depth of field or fogging.

HiQ Only

Activating this checkbox causes the Refine Z option to process only when rendering is set to High Quality. You can ensure High Quality is enabled by right-clicking to the left or right of the transport controls in the main toolbar.

Strength

Increasing this slider does two things. It smooths out the depth in constant color regions and moves edges in the Z channel to correlate with edges in the RGB channels.

Increasing the refinement has the undesirable effect of causing texture in the color channel to show up in the Z channel. You will want to find a balance between the two.

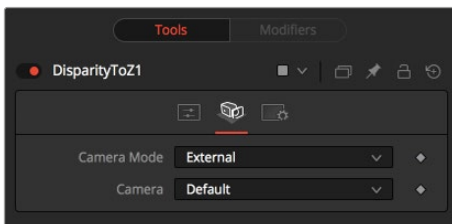**Radius**

This is the radius of the smoothing algorithm.

Stack Mode

This menu determines how the input images are stacked.

When set to Separate, the Right Input and Output will appear, and separate left and right images must be connected.

Swap Eyes

Enabling this checkbox causes left and right images to be swapped.

The Disparity To Z Camera tab

Camera Tab

If you need correct real-world Z values because you are trying to match some effect to an existing scene, you should use the External Camera options to get precise Z values back. If any Z-buffer will suffice and you are not that particular about the exact details of how it is offset and scaled, or if there is no camera available, the Artistic option might be helpful.

— **External Mode:** An input is available on the node to connect an existing stereo Camera 3D. This can either be a single stereo Camera 3D (i.e., its eye separation is set to non-zero), or a pair of (tracked) Camera 3Ds connected via the Camera 3D > Stereo > Right Camera input.

— **Artistic Mode:** If you do not have a camera, you can adjust these controls to produce an "artistic" Z channel whose values will not be physically correct but will still be useful. To reconstruct the Disparity > Z Curve, pick (D, Z) values for a point in the foreground and a point in the background.

> **TIP:** If artistic mode is a little too "artistic" for you and you want more physically-based parameters to adjust (e.g., convergence and eye separation), create a dummy Camera 3D, connect it into the Disparity To Z > Camera input, and then fiddle with the Camera 3D's controls.

Foreground Disparity (Pick from Left Eye)

When the camera Mode is set to Artistic, a Foreground Disparity slider is available. This is the disparity for the closest foreground object. It will get mapped to the depth value specified by the Foreground Depth control. Any objects with disparity outside of the range [ForegroundDisparity, BackgroundDisparity] will have their disparity values clipped to this range leading to flat areas in the Z channel, so make sure that you pick values that enclose the actual disparity range.

Background Disparity (Pick from Left Eye)

When the camera Mode is set to Artistic, a Background Disparity is available. This is the disparity for the furthest background object. It will get mapped to the depth value specified by the Background Depth control. One way to think of this input is as the upper limit to disparity values for objects at -infinity. This value should be for the left eye. The corresponding value in the right eye will be the same in magnitude but negative.

Foreground Depth

This is the depth to which Foreground Disparity will be mapped. Think of this as the depth of the nearest object. Note that values here are positive depth.

Background Depth

This is the depth to which Background Disparity will be mapped. Think of this as the depth of the most distant object.

Falloff

Falloff controls the shape of the depth curve between the requested foreground and background depths. When set to Hyperbolic, the disparity-depth curve behaves roughly like depth = constant/disparity. When set to Linear, the curve behaves like depth = constant * disparity. Hyperbolic tends to emphasize Z features in the foreground, while linear gives foreground/background features in the Z channel equal weighting.

Unless there's a specific reason, choose Hyperbolic, as it is more physically accurate, while Linear does not correspond to nature and is purely for artistic effect.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Stereo nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Global Align [GA]

The GlobalAlign node

> **NOTE:** The Global Align node is available only in Fusion Studio and DaVinci Resolve Studio.

Global Align Node Introduction

As opposed to Stereo Align, this node does not utilize optical flow at all. It's meant as a fast and convenient way to do simple stereo alignment for both X and Y as well as rotation.

Global Align comes in handy at the beginning of the node chain to visually correct major differences between the left and right eye before calculating Disparity.

Manual correction of large discrepancies between left and right, as well as applying an initial color matching, helps Disparity generate more accurate results.

Inputs

The two inputs on the Global Align node are used to connect the left and right images.

— **Left Input:** The orange input is used to connect either the left eye image or the stack image.
— **Right Input:** The green input is used to connect the right eye image. This input is available only when the Stack Mode menu is set to Separate.

Outputs

Unlike most nodes in Fusion, Global Align has two outputs for the left and right eye.

— **Left Output:** This outputs the newly aligned left eye image.
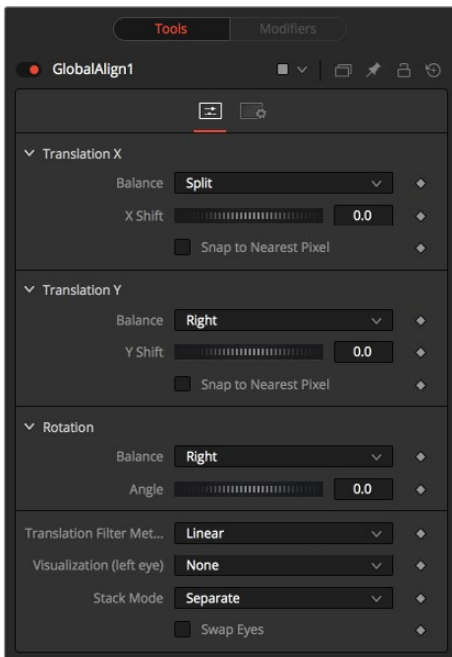— **Right Output:** This outputs the newly aligned right eye image.

Basic Node Setup

Global Align is typically placed at the beginning of the node tree. Below it is inserted between the left and right eye images and the Disparity node to visually correct major differences.

A Global Align node used to manually correct left and right eye discrepancies

Inspector

The Global Align Controls tab

Controls Tab

The Controls tab includes translation and rotation controls to align the stereo images manually.

Translation X and Y

— **Balance:** Determines how the global offset is applied to the stereo footage.

— **None:** No translation is applied.

— **Left Only:** The left eye is shifted, while the right eye remains unaltered.

— **Right Only:** The right eye is shifted, while the left eye remains unaltered.

— **Split Both:** Left and right eyes are shifted in opposite directions.

Snap to Nearest Pixel

While adjusting the X or Y shift dial, this option ensures that the image is shifted in full pixel amounts only to maintain optimum quality. This avoids sub-pixel rendering of the image, which could result in subtle blurring.

Rotation

— **Balance:** Determines how the global rotation is applied to the stereo footage.

— **None:** No rotation is applied.

— **Left Only:** The left eye is rotated, while the right eye remains unaltered.

— **Right Only:** The right eye is rotated, while the left eye remains unaltered.

— **Split Both:** Left and right eyes are rotated in opposite directions.

Angle

This dial adjusts the angle of the rotation. Keep in mind that the result depends on the Balance settings. If only rotating one eye by, for example, 10 degrees, a full 10-degree rotation will be applied to that eye.

When applying rotation in Split mode, one eye will receive a -5 degree and the other eye a +5 degree rotation.

Translation Filter Method

This menu chooses the filter method that delivers the best results depending on the content of your footage.

Visualization

This control allows for different color encodings of the left and right eye to conveniently examine the results of the above controls without needing to add an extra Anaglyph or Combiner node.

Set this to None for final output.

Stack Mode

Determines how the input images are stacked.

When set to Separate, the right input and output will appear, and separate left and right images must be connected.

Swap Eyes

With Stacked Mode, image stereo pairs' left and right images can be swapped.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Stereo nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

New Eye [NE]

The New Eye node

> **NOTE:** The New Eye node is available only in Fusion Studio and DaVinci Resolve Studio.

New Eye Node Introduction

The New Eye node constructs a new image by interpolating between two existing stereo images using the embedded disparity channels. This node can also be used to replace one view with a warped version of the other. In Stack Mode, L and R outputs will output the same image.

You can map the left eye onto the right eye and replace it. This can be helpful when removing errors from certain areas of the frame.

New Eye does not interpolate the aux channels but instead destroys them. In particular, the disparity channels are consumed/destroyed. Add another Disparity node after the New Eye if you want to generate Disparity for the realigned footage.

Inputs

The two inputs on the New Eye node are used to connect the left and right images.

— **Left Input:** The orange input is used to connect either the left eye image or the stack image.
— **Right Input:** The green input is used to connect the right eye image. This input is available only when the Stack Mode menu is set to Separate.

Outputs

Unlike most nodes in Fusion, New Eye has two outputs for the left and right eye.

— **Left Output:** This outputs the left eye image with a new disparity channel, or a Stacked Mode image with a new disparity channel.
— **Right Output:** This outputs the right eye image with a new disparity channel. This output is visible only if Stack Mode is set to Separate.
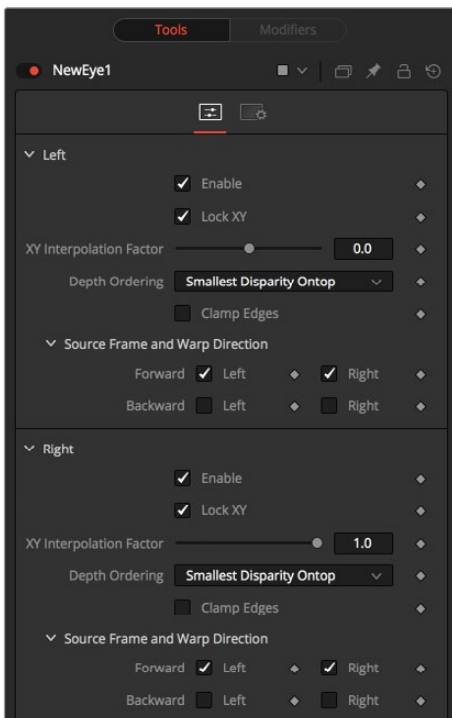
Basic Node Setup

The New Eye node in the example below constructs a new image by interpolating between the two stereo images using the embedded disparity channels.
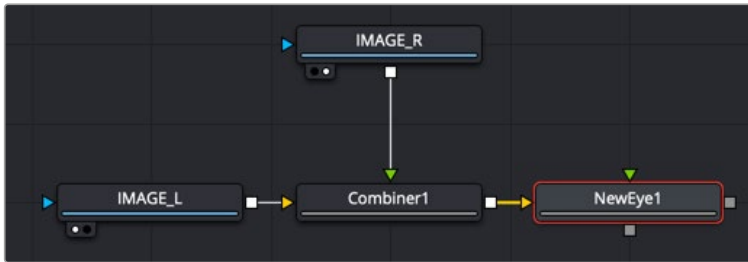
A New Eye node creates a new stereo image using embedded disparity

Inspector

The New Eye Controls tab

Controls Tab

The Controls tab is divided into identical parameters for the left eye and right eye. The parameters are used to select which eye to recreate and the methods used for the interpolation.

Enable

The Enable checkbox allows you to activate the left or right eye independently. The New Eye will replace enabled eye with an interpolated eye. For example, if the left eye is your "master" eye and you are recreating the right eye, you would disable the left eye and enable the right eye.

Lock XY

Locks the X and Y interpolation parameters. When they are unlocked, you can provide separate interpolation factors for using the X and Y disparity. For example, if you are working with the right eye and you have the X Interpolation slider set to 1.0 and the Y Interpolation slider set to -1.0, you will be effectively interpolating the left eye onto the right eye but vertically aligned to the left eye.

XY Interpolation Factor

Interpolation determines where the interpolated frame is positioned, relative to the two source frames: A slider position of -1.0 outputs the frame Left and a slider position of 1.0 outputs the frame Right. A slider position of 0.0 outputs a result that is halfway between Left and Right.

Depth Ordering

The Depth Ordering is used to determine which parts of the image should be rendered on top. When warping images, there is often overlap. When the image overlaps itself, there are two options for which values should be drawn on top.

— **Largest Disparity On Top:** The larger disparity values will be drawn on top in the overlapping image sections.

— **Smallest Disparity On Top:** The smaller disparity values will be drawn on top in the overlapping image sections.

Clamp Edges

Under certain circumstances, this option can remove the transparent gaps that may appear on the edges of interpolated frames. Clamp Edges will cause a stretching artifact near the edges of the frame that is especially visible with objects moving through it or when the camera is moving.

Because of these artifacts, it is a good idea to use Clamp Edges only to correct small gaps around the edges of an interpolated frame.

Softness

Helps to reduce the stretchy artifacts that might be introduced by Clamp Edges.

If you have more than one of the Source Frame and Warp Direction checkboxes turned on, this can lead to doubling up of the stretching effect near the edges. In this case, you'll want to keep the softness rather small at around 0.01. If you have only one checkbox enabled, you can use a larger softness at around 0.03.

Source Frame and Warp Direction

The output of this node is generated by combining up to four different warps. You can choose to use either the color values from the left or right frame in combination with the Forward (left > right) Disparity or the Backward (right > left) Disparity. Sometimes you will want to replace an existing eye. For example, if you want to regenerate the right eye, you would only use left eye warps.

It's good to experiment with various options to see which gives the best effect. Using both the left and right eyes can help fill in gaps on the left/right side of images. Using both the Forward/Backward Disparity can give a doubling-up effect in places where the disparities disagree with each other.

— **Left Forward:** Takes the Left frame and uses the Forward Disparity to interpolate the new frame.

— **Right Forward:** Takes the Right frame and uses the Forward Disparity to interpolate the new frame.

— **Left Backward:** Takes the Left frame and uses the Back Disparity to interpolate the new frame.

— **Right Backward:** Takes the Right frame and uses the Back Disparity to interpolate the new frame.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Stereo nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Splitter [Spl]

The Splitter node

> **NOTE:** The Splitter node is available only in Fusion Studio and DaVinci Resolve Studio.

Splitter Node Introduction

The Splitter takes a stacked input image—for example, created with the Combiner—and provides two output images: a left eye and a right eye.

Inputs

The two inputs on the Splitter node are used to connect the left and right images.

— **Left Input:** The orange input is used to connect a stacked stereo image.

Outputs

Unlike most nodes in Fusion, the Splitter node has two outputs for the left and right eye.

— **Left Output:** This outputs the left eye image.
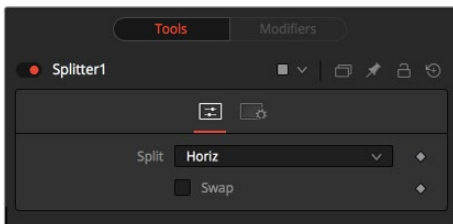— **Right Output:** This outputs the right eye image.

Basic Node Setup

Below, a stacked stereo image is connected to the input on a Splitter where a left and right eye is output.

A Splitter node creates a left and right image from a stacked stereo image

Inspector

The Splitter Controls tab

Controls Tab

The Controls tab is used to define the type of stacked image connected to the node's input.

Split

The Split menu contains three options for determining the orientation of the stacked input image.

— **None:** No operation takes place. The output image on both outputs is identical to the input image.
— **Horiz:** The node expects a horizontally stacked image. This will result in two output images, each being half the width of the input image.
— **Vert:** The node expects a vertically stacked image. This will result in two output images, each being half the height of the input image.

Swap Eyes

Allows you to easily swap the left and right eye outputs.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Stereo nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Stereo Align [SA]

The StereoAlign node

> **NOTE:** The Stereo Align node is available only in Fusion Studio and DaVinci Resolve Studio.

Stereo Align Node Introduction

This extremely versatile node for fixing Stereo issues can be used for performing any of the following actions or combinations thereof:

— Vertical alignment of one eye to the other
— Changing the convergence
— Changing the eye separation

By combining these operations in one node, you can execute them using only a single image resampling. In essence, this node can be thought of as applying scales and translation to the disparities and then using the modified disparities to interpolate between the views.

> **NOTE:** Changing the eye separation can cause holes to appear, and it may not be possible to fill them since the information needed may not be in either image. Even if the information is there, the disparity may have mismatched the holes. You may need to fill the holes manually. This node modifies only the RGBA channels.

> **TIP:** Stereo Align does not interpolate the aux channels but instead destroys them. In particular, the disparity channels are consumed/destroyed. Add another Disparity node after the StereoAlign if you want to generate Disparity for the realigned footage.

Inputs

The two inputs on the Stereo Align node are used to connect the left and right images.

— **Left Input:** The orange input is used to connect either the left eye image or the stack image.
— **Right Input:** The green input is used to connect the right eye image. This input is available only when the Stack Mode menu is set to Separate.

Outputs

Unlike most nodes in Fusion, Stereo Align has two outputs for the left and right eye.

— **Left Output:** This outputs the left eye image with a new disparity channel, or a Stacked Mode image with a new disparity channel.
— **Right Output:** This outputs the right eye image with a new disparity channel. This output is visible only if Stack Mode is set to Separate.
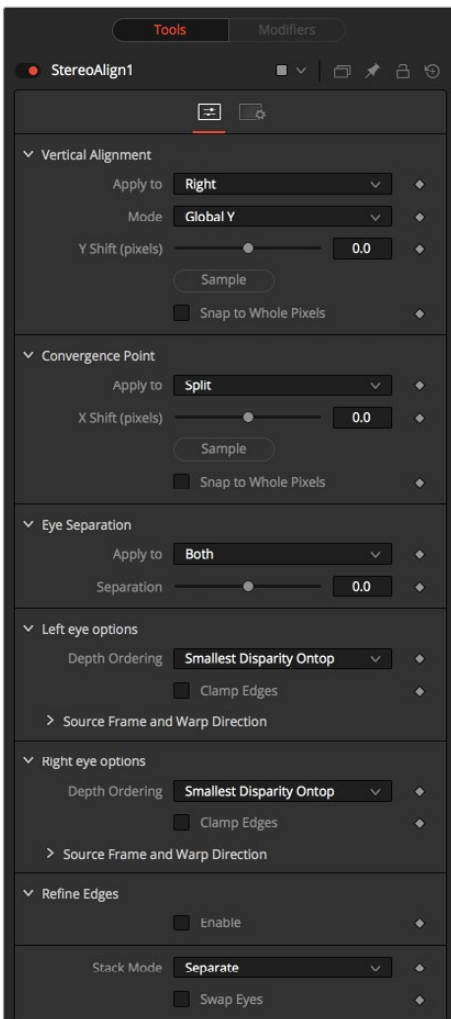
Basic Node Setup

Below, the Stereo Align receives the left and right eye images with disparity. Once the adjustments are made, another Disparity node is added after it to generate disparity for the realigned footage.

A Stereo Align node destroys the disparity channel, so another Disparity node is placed after it

Inspector

The Stereo Align Controls tab

Controls Tab

Vertical Alignment

This option determines how the vertical alignment is split between two eyes. Usually, the left eye is declared inviolate, and the right eye is aligned to it to avoid resampling artifacts.

When doing per pixel vertical alignment, it may be helpful to roughly pre-align the images by a global Y-shift before disparity computation because the disparity generation algorithm can have problems resolving small objects that move large distances.

Also, be aware that you must be careful about lens distortion because even if two cameras are perfectly vertically aligned, they will still have vertical disparities due to lens distortion. As a best practice, remove the lens distortion before computing the disparity. When a vertical alignment of the right eye is done, you have essentially removed the Y-component of the lens distortion in the right eye, and it will look wrong later when you try to distort it again.

Apply to

— **Right:** Only the right eye is adjusted.

— **Left:** Only the left eye is adjusted.

— **Both:** The vertical alignment is split evenly between the left and right eyes.

Mode

— **Global:** The eyes are simply translated up or down by the Y-shift to match up.

— **Per Pixel:** The eyes are warped pixel-by-pixel using the disparity to vertically align.

Keep in mind that this can introduce sampling artifacts and edge artifacts.

Y-shift

Y-shift is available only when the Mode menu is set to Global. You can either adjust the Y-shift manually to get a match or drag the Sample button into the viewer, which picks from the disparity channel of the left eye. Also remember, if you use this node to modify disparity, you can't use the Sample button while viewing the node's output.

Snap to Whole Pixels

You can snap the global shift to whole pixels by enabling this option. When enabled, there is no resampling of the image, but rather a simple shift is done so there will be no softening or image degradation.

Convergence Point

The Convergence Point section is used as a global X-translation of L/R images.

Apply to

This menu determines which eyes are affected by convergence. You can choose to apply the convergence to the left eye, right eye, or split between the two. In most cases, this will be set to Split. If you set the eyes to Split, then the convergence will be shared 50-50 between both eyes. Sharing the convergence between both eyes means you get half the shift in each eye, which in turn means smaller holes and artifacts that need to be fixed later. The tradeoff is that you've resampled both eyes rather than keeping one eye as a pure reference master.

X-shift

You can either use the slider to adjust the X-shift manually to get a match or use the Sample button to pick from the disparity channels for easy point-to-feature alignment.

Snap

You can snap the global shift to whole pixels using this option. In this mode, there is no resampling of the image, but rather a simple shift is done so there will be no softening or image degradation.

Eye Separation

Eye Separation changes the distance between the left/right eyes, causing objects in the left/right eyes to converge/diverge further depending on their distance from the camera.

This has the same effect as the Eye Separation option in the Camera 3D node.

Separation

This is a scale factor for eye separation.

— When set to 0.0, this leaves the eyes unchanged.
— Setting it to 0.1 increases the shifts of all objects in the scene by a factor of 10% in each eye.
— Setting it to 0.1 will scale the shifts of all objects 10% smaller.

Unlike the Split option for vertical alignment, which splits the alignment effect 50-50 between both eyes, the Both option will apply 100-100 eye separation to both eyes. If you are changing eye separation, it can be a good idea to enable per-pixel vertical alignment, or the results of interpolating from both frames can double up.

Left/Right Eye Options

The left and right eye options contain depth ordering and warp direction controls independently for the left and right eye.

Depth Ordering

The Depth Ordering is used to determine which parts of the image should be rendered on top. When warping images, there is often overlap. When the image overlaps itself, there are two options for which values should be drawn on top.

— **Largest Disparity On Top:** The larger disparity values will be drawn on top in the overlapping image sections.
— **Smallest Disparity On Top:** The smaller disparity values will be drawn on top in the overlapping image sections.

Clamp Edges

Under certain circumstances, this option can remove the transparent gaps that may appear on the edges of interpolated frames. Clamp Edges will cause a stretching artifact near the edges of the frame that is especially visible with objects moving through it or when the camera is moving.

Because of these artifacts, it is a good idea to use Clamp Edges only to correct small gaps around the edges of an interpolated frame.

Edge Softness

Helps to reduce the stretchy artifacts that might be introduced by Clamp Edges.

If you have more than one of the Source Frame and Warp Direction checkboxes turned on, this can lead to doubling up of the stretching effect near the edges. In this case, you'll want to keep the softness rather small at around 0.01. If you have only one checkbox enabled, you can use a larger softness at around 0.03.

Source Frame and Warp Direction

The output of this node is generated by combining up to four different warps. You can choose to use either the color values from the left or right frame in combination with the Forward (left > right)

Disparity or the Backward (right > left) Disparity. Sometimes you will want to replace an existing eye. For example, if you want to regenerate the right eye, you would use only left eye warps.

It's good to experiment with various options to see which gives the best effect. Using both the left and right eyes can help fill in gaps on the left/right side of images. Using both the Forward/Backward Disparity can give a doubling-up effect in places where the disparities disagree with each other.

— **Left Forward:** Takes the Left frame and uses the Forward Disparity to interpolate the new frame.

— **Right Forward:** Takes the Right frame and uses the Forward Disparity to interpolate the new frame.

— **Left Backward:** Takes the Left frame and uses the Back Disparity to interpolate the new frame.

— **Right Backward:** Takes the Right frame and uses the Back Disparity to interpolate the new frame.

Stack Mode

In Stack Mode, L and R outputs will output the same image.

If High Quality is off, the interpolations are done using nearest-neighbor sampling, leading to a more "noisy" result. To ensure High Quality is enabled, right-click under the viewers, near the transport controls, and choose High Quality from the pop-up menu.
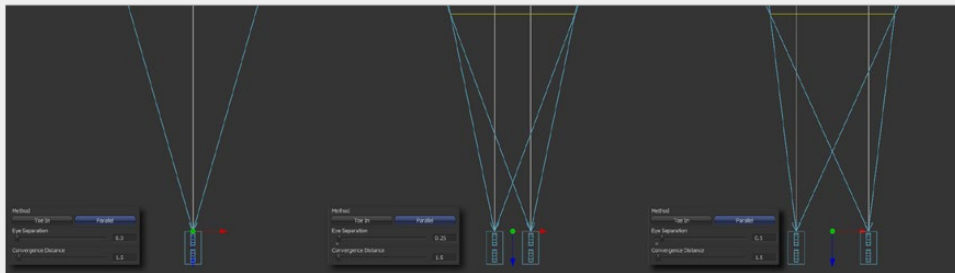
Swap Eyes

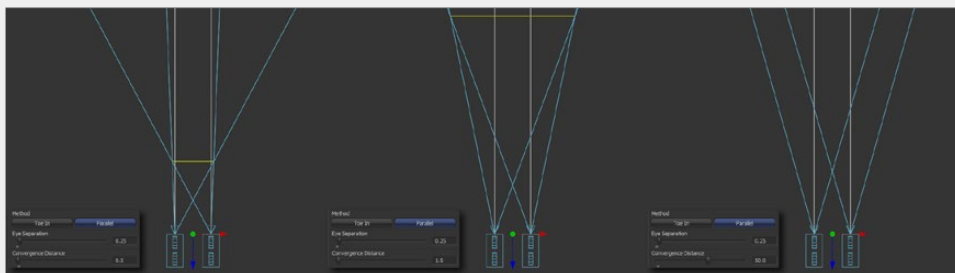Allows you to easily swap the left and right eye outputs.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Stereo nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

EXAMPLE

Different settings for Eye Separation...

...and example settings for Convergence

Z To Disparity [Z2D]

The Z To Disparity node

> **NOTE:** The Z To Disparity node is available only in Fusion Studio and DaVinci Resolve Studio.

Z To Disparity Node Introduction

Z To Disparity takes a stereo camera and an image containing a Z channel and outputs the same image but with disparity channels in it. This is useful for constructing a Disparity map from CG renders, which will be more accurate than the Disparity map created from the Disparity node.

Inputs

The three inputs on the Z To Disparity node are used to connect the left and right images and a camera node.

— **Left Input:** The orange input is used to connect either the left eye image or the stack image.
— **Right Input:** The green input is used to connect the right eye image. This input is available only when the Stack Mode menu is set to Separate.
— **Stereo Camera:** The magenta input is used to connect a stereo perspective camera, which may be either a Camera 3D with eye separation, or a tracked L/R Camera 3D.
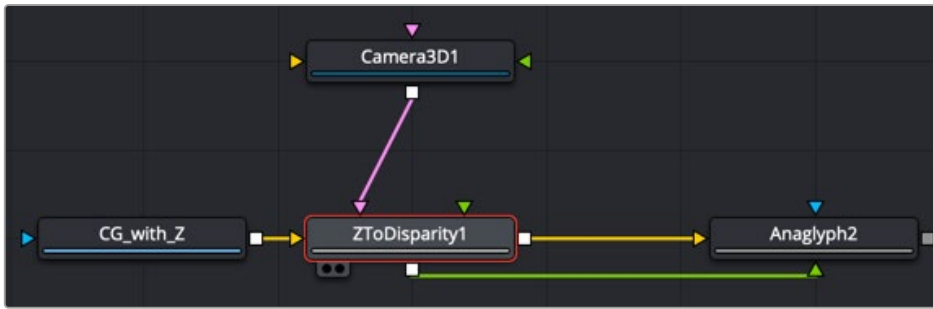
Outputs

Unlike most nodes in Fusion, Z To Disparity has two outputs for the left and right eye.

— **Left Output:** This outputs the left eye image containing a new disparity channel, or a Stacked Mode image with a new disparity channel.
— **Right Output:** This outputs the right eye image with a new disparity channel. This output is visible only if Stack Mode is set to Separate.
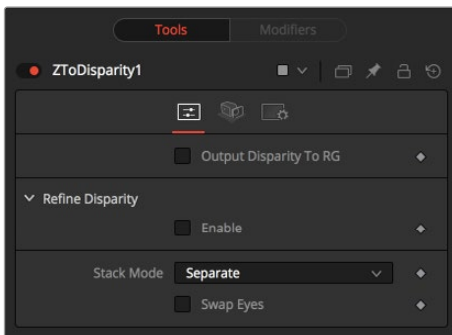
Basic Node Setup

Below, a stereo camera and an image containing a Z channel is connected to a Z To Disparity node. The same image is output with disparity channels.

A Z To Disparity node takes an image with a Z channel and creates a disparity channel

Inspector

The Z To Disparity Controls tab

Controls Tab

The Controls tab includes settings that refine the conversion algorithm.

Output Disparity To RGB

In addition to outputting disparity values into the disparity channel, activating this checkbox causes Z To Disparity to also output the disparity values into the color channels as {x, y, 0, 1}.

When activated, this option will automatically promote the RGBA color channels to float32. This option is useful for a quick look to see what the disparity channel looks like.

Refine Disparity

This refines the Disparity map based on the RGB channels.

Strength

Increasing this slider does two things. It smooths out the depth in constant color regions and moves edges in the Z channel to correlate with edges in the RGB channels. Increasing the refinement has the undesirable effect of causing texture in the color channel to show up in the Z channel. You will want to find a balance between the two.

Radius
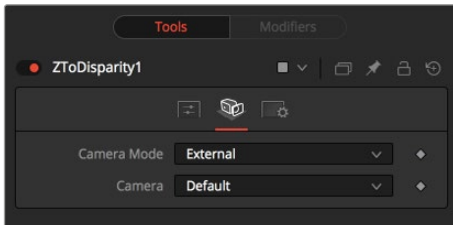
This is the pixel-radius of the smoothing algorithm.

Stack Mode

In Stack Mode, L and R outputs will output the same image.

If HiQ is off, the interpolations are done using nearest-neighbor sampling, leading to a more "noisy" result.

Swap Eyes

This allows you to easily swap the left and right eye outputs.

The Z To Disparity Camera tab

Camera Tab

The Camera tab includes settings for selecting a camera and setting its conversion point if necessary.
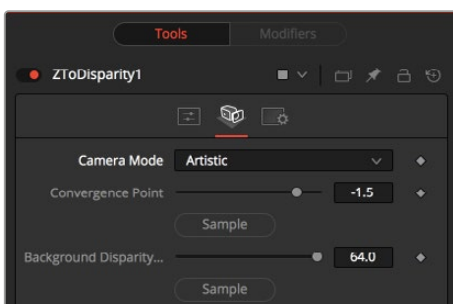
Camera Mode

If you need correct real-world disparity values because you are trying to match some effect to an existing scene, you should use the External setting to get precise disparity values back. When External is selected, a magenta camera input is available on the node to connect an existing stereo Camera 3D node, and use the Camera settings to determine the Disparity settings.

If you just want any disparity and do not particularly care about the exact details of how it is offset and scaled, or if there is no camera available, then the Artistic setting might be helpful.

Camera

If you connect a Merge 3D node that contains multiple cameras to the camera input, the Camera menu allows you to select the camera to use.

Artistic Camera mode

If you do not have a camera, you can adjust the artistic controls to produce a custom disparity channel whose values will not be physically correct but will be good enough for compositing hacks. There are two controls to adjust:

Convergence Point

This is the Z value of the convergence plane. This corresponds to the negative of the Convergence Distance control that appears in Camera 3D. At this distance, objects in the left and right eyes are at exactly the same position (i.e., have zero disparity).

Objects that are closer appear to pop out of the screen, and objects that are further appear behind the screen.

Background Disparity (Sample from Left Eye)

This is the disparity of objects in the distant background. You can think of this as the upper limit to disparity values for objects at infinity. This value should be for the left eye. The corresponding value in the right eye will be the same in magnitude but negative.
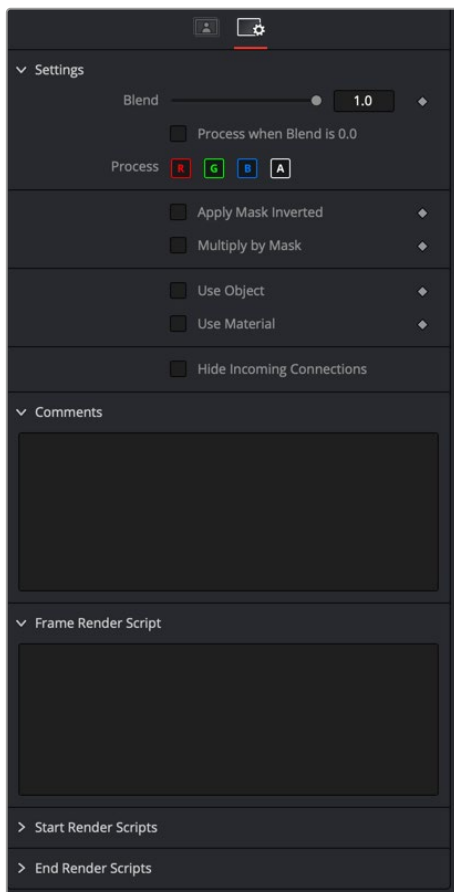
Settings Tab

The Settings tab in the Inspector is also duplicated in other Stereo nodes. These common controls are described in detail in the following "The Common Controls" section.

The Common Controls

The Common Settings tab can be found on virtually every tool found in Fusion. The following controls are specific settings for Stereo nodes.

Settings Tab

The Common Settings Stereo 3D Settings tab

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this will cause the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if the node is scripted to trigger a task, but the node's value is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the Red button on a Blur tool is deselected, the blur will first be applied to the image, and then the red channel from the original input will be copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this will generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option will cause the RGB values of the masked image to be multiplied by the mask channel's values. This will cause all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object ID and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels will be used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option is disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID will be used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the viewer. The image or sequence must have been rendered from a 3D software package with those channels included.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node will be displayed in the Inspector. Dragging a connected node from the node tree into the field will hide that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line will reappear.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Tracker Nodes

This chapter details the Tracker nodes available in Fusion.

The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Tracker [Tra]

The Tracker node

Tracker Node Introduction

The Tracker is used to detect and follow one or more pixel patterns across frames in moving video. The tracking data can then be used to control the position or values of other nodes in the composition (for example, the center of a Light Rays node). Additionally, trackers can be used to stabilize an image or to apply destabilization to one image based on the motion of another.

For more information, refer to Chapter 22, "Using the Tracker Node," in the Fusion Reference Manual or Chapter 82 in the DaVinci Resolve Reference Manual.

Inputs

The Tracker has three inputs:

— **Background:** The orange image input accepts the main 2D image to be tracked.
— **Foreground:** The optional green foreground accepts a 2D image to be merged on top of the background as a corner pin or match move.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the tracking to certain areas.

Basic Node Setup

Tracker nodes can be applied inline with other nodes or as a branch from the clip you want to track. When used inline, an image can be stabilized by connecting it to the orange background input. After the image is tracked, setting the Tracker's Operation menu to Match Move will apply a stabilization to the connected image.
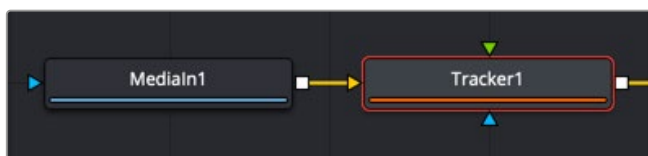

A Tracker node connected inline to an image for stabilization

When used as an offshoot from the node tree, an image can be tracked and then that tracking data is published for use on another node somewhere else in the node tree. The output of the Tracker does not need to connect to another node. The tracking data is published and can be used via the Connect To contextual menu.
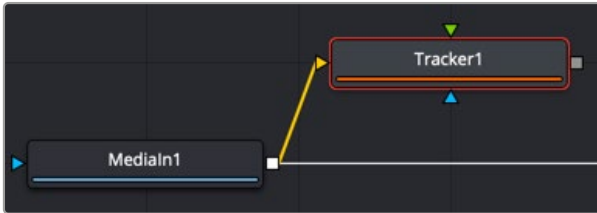
A Tracker node branched from the node tree

The Tracker can also work as a replacement for a Merge tool in match-moving setups. Below, the Tracker tracks the image connected to the orange background input and applies the tracking data to the image connected to the foreground input. The same foreground-over-background merge capabilities are available in the Tracker node.
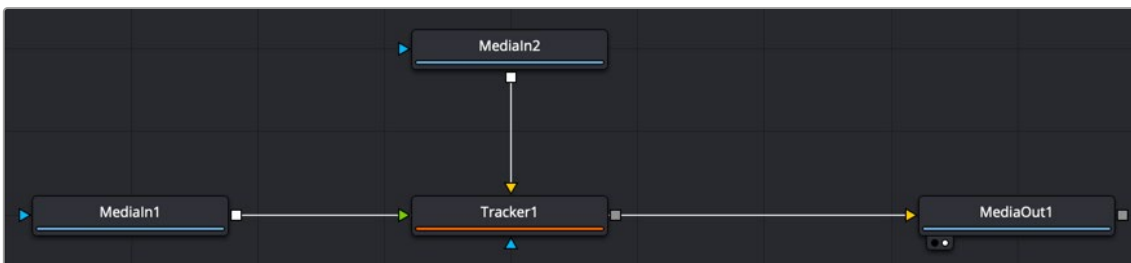
A Tracker node set up to apply a match move to the foreground input

Tracker Onscreen Controls

Each pattern in the Tracker has its own set of onscreen controls used to select the pixels in the image to be tracked. These controls are visible in the viewers whenever you select a tracker in the node tree.

Tracker onscreen controls
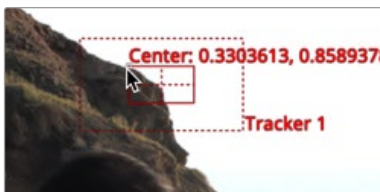
Pattern Rectangle

In the viewer, the tracker displays a solid-line red rectangle called the pattern rectangle. Every pixel within the rectangle makes up the pattern used for tracking. You can resize the pattern if necessary by dragging on the rectangle's borders.

A pattern rectangle identifies
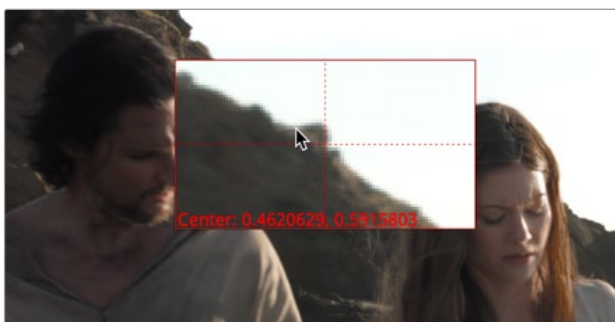the area to track

Search Rectangle

Whenever the mouse moves over the pattern rectangle, a second rectangle with a dashed outline appears. The dashed outline represents the search area, which determines how far away from the current pattern the Tracker looks in the next frame. The search area should always be larger than the pattern, and it should be large enough to encompass the largest frame-to-frame movement in the scene. Faster moving objects require larger search areas, and slower moving objects can get away with smaller search areas. The larger the search area, the longer it takes to track, so try not to make the search area larger than necessary. If the selected Tracker has a custom name, the name of that Tracker is displayed as a label at the bottom right of the search area rectangle.

The search rectangle is the
area searched from frame to
frame to locate the pattern.

Repositioning the Tracker

The pattern rectangle has a small handle in the upper-left corner. Dragging on the handle repositions the pattern. An enlarged view of the pattern is displayed under your mouse pointer to assist with the precise positioning of the pattern. This thumbnail disappears when the mouse button is released. You can adjust the magnification ratio in the Inspector's Options tab.

Dragging the handle magnifies the pattern
rectangle for precise placement.

Inspector

The Tracker node Trackers tab

Trackers Tab

The Trackers tab contains controls for creating, positioning, and initiating tracking operations. After tracking, offset controls are used to improve the alignment of the image following the track.

Track Buttons

There are four buttons to initiate tracking, and one button in the middle used to stop a track in progress. These buttons can track the current pattern forward or backward in time. Holding the pointer over each button displays a tooltip with the name of each button.

The buttons operate as follows:

— **Track Reverse:** Clicking this button causes all active trackers to begin tracking, starting at the end of the render range and moving backward through time until the beginning of the render range.

— **Track Reverse From Current Time:** Clicking this button causes all active trackers to begin tracking, starting at the current frame and moving backward through time until the beginning of the render range.

— **Stop Tracking:** Clicking this button or pressing ESC stops the tracking process immediately. This button is active only when tracking is in process.

— **Track Forward From Current Time:** Clicking this button causes all active trackers to begin tracking, starting at the current frame and moving forward through time until the end of the render range.

— **Track Forward:** Clicking this button causes all active trackers to begin tracking, starting at the first frame in the render range and moving forward through time until the end of the render range.

Tracking Behavior Controls

The following controls all affect how trackers adapt to changing patterns, how the resulting track path is defined, and how many keyframes are generated.

Frames Per Path Point

This slider determines how often the Tracker sets a keyframe on the path. The default is 1, which sets a keyframe on the tracked path at every frame.

Increasing the value causes the tracked path to be less accurate. This may be desirable if the track is returning fluctuating results, but under normal circumstances, leave this control at its default value.

> **TIP:** If the project is field rendered, a value of 1 sets a keyframe on every field. Since the Tracker is extremely accurate, this will result in a slight up-and-down jittering due to the position of the fields. For better results when tracking interlaced footage in Field mode, set the Frames Per Path Point slider to a value of 2, which results in one keyframe per frame of your footage.

Adaptive Mode

Fusion is capable of reacquiring the tracked pattern, as needed, to help with complex tracks. This menu determines the Adaptive tracking method.

— **None:** When set to None, the tracker searches for the original pattern in each frame.
— **Every Frame:** When set to Every Frame, the tracker reacquires the pattern every frame. This helps the Tracker compensate for gradual changes in profile and lighting over time.

— **Best Match:** When set to Best Match, the tracker compares the original selected pattern to the pattern acquired at each frame. If the variation between the two patterns exceeds the threshold amount defined by the Match Tolerance control, the tracker does not reacquire the pattern on that frame. This helps to avoid Tracker drift caused by transient artifacts that cross the pattern's path (such as a shadow).

Path Center

This menu determines how the Tracker behaves when repositioning a pattern. This menu is particularly useful when a pattern leaves the frame or changes so significantly that it can no longer be tracked.

— **Pattern Center:** When Pattern Center is selected in the menu, the tracked path continues from the center of the new path. This is appropriate when replacing an existing path entirely.

— **Track Center (append):** When Track Center (append) is selected in the menu, the path tracked by a new pattern will be appended to the existing path. The path created is automatically offset by the required amount. This setting is used to set a new tracking pattern when the original pattern moves out of the frame or gets obscured by other objects. This technique work bests if the new pattern is located close to the position of the original pattern to avoid any problems with parallax or lens distortion.

Tracker List

A Tracker node can track multiple patterns. Each tracker pattern created in the current Tracker node is managed in the Tracker List.

Tracker List

The Tracker List shows the names of all trackers created.

— Each tracker pattern appears in the list by name, next to a small checkbox. Clicking the name of the tracker pattern will select that tracker pattern.

— The controls below the list will change to affect that tracker pattern only. Click a selected tracker pattern once to rename the tracker pattern to something more descriptive.

— Clicking the checkbox changes the state of the tracker.

Tracker States

— **Enabled (black checkbox):** An enabled pattern will re-track each time the track is initiated. Its path data is available for use by other nodes, and the data is available for Stabilization and Corner Positioning.

— **Suspended (white circle):** A Suspended pattern does not re-track when the track is initiated. The data is locked to prevent additional changes. The data from the path is still available for other nodes, and the data is available for advanced Tracking modes like Stabilization and Corner Positioning.

— **Disabled (clear):** A Disabled pattern does not create a path when tracking is initialized, and its data is not available to other nodes or for advanced Tracking operations like Stabilization and Corner Positioning.
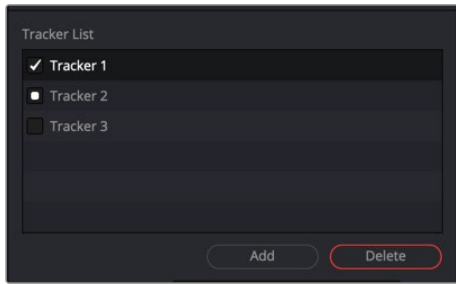
Three tracking patterns from top to bottom:
enabled, suspended, and disabled

Add/Delete Tracker

Use these buttons to add or delete trackers from your Tracker List.

Show

This menu selects what controls are displayed in the Tracker node controls. They do not affect the operation of the tracker; they only affect the lower half of the Inspector interface.

— **Selected Tracker Details:** When Selected Tracker Details is chosen, the controls displayed pertain only to the currently selected tracker. You will have access to the Pattern window and the Offset sliders.

— **All Trackers:** When All Trackers is selected, the pattern window for each of the added tracking patterns is displayed simultaneously below the Tracker List.

Left Pattern Display

The pattern display has two side-by-side image windows and a series of status bars. The window on the left shows the pattern initially selected, while the one on the right shows a real-time display of the current pattern as tracking progresses.

As the onscreen controls move while tracking, the display in the leftmost window updates to show the pattern. As the pattern moves, the vertical bars immediately to the right of the image indicate the clarity and contrast of the image channels.

The best channel or channels get selected for tracking based on clarity. These channels have a gray background in the vertical bar representing that channel. You can use the automatic tracking or override the selection and choose the channel by selecting the button beneath the channel to track.

Tracker pattern display

Under normal circumstances, the channel selected shows in the pattern display. If the selected channel is blue, then a grayscale representation of the blue channel for the pattern appears.
The image is represented in color only when you activate the Full Color button.

Override this behavior by selecting the Show Full Color button beneath the pattern display instead of the Show Selected Channel button.

> **TIP:** Because Fusion looks for the channel with the highest contrast automatically, you might end up tracking a noisy but high-contrast channel. Before tracking, it's always a good idea to zoom in to your footage and check the RGB channels individually.

Right Pattern Display

The pattern display on the right indicates the actual pattern acquired for tracking. This display is black until tracking the selected pattern for the first time. The pattern display becomes active during tracking, displaying the pattern that Fusion acquires from frame to frame.

As the tracking occurs, the pattern from each frame accumulates into a Flipbook, which can be played back in the pattern window after tracking by using the transport controls at the bottom of the window.

While the track is progressing, the vertical bar immediately to the right of the pattern shows how confident Fusion is that the current pattern matches the initially selected pattern. A green bar indicates a high degree of confidence that the current pattern matches the original, a yellow bar indicates less certainty, and a red bar indicates extreme uncertainty.

After tracking, the pattern display shows a small Flipbook of the track for that pattern to help identify problem frames for the track.

Tracker Sizes

In addition to onscreen controls, each tracker has a set of sizing parameters that let you adjust the pattern and search box.

— **Pattern Width and Height:** Use these controls to adjust the width and height of the selected tracker pattern manually. The size of the tracker pattern can also be adjusted in the viewer, which is the normal method, but small adjustments are often easier to accomplish with the precision of manual controls.

— **Search Width and Height:** The search area defines how far Fusion will look in the image from frame to frame to reacquire the pattern during tracking. As with the Pattern Width and Height, the search area can be adjusted in the viewer, but you may want to make small adjustments manually using these controls.

Tracked Center

This positional control indicates the position of the tracker's center. To remove a previously tracked path from a tracker pattern, right-click this parameter and select Remove Path from the contextual menu.

X and Y Offset

The Offset controls help to create a track for objects that may not provide very well defined or reliable patterns. The Offset controls permit the tracking of something close to the intended object instead. Use these Offsets to adjust the desired position of the path, while the tracker pattern rectangle is positioned over the actual tracking location.

The Offset can also be adjusted directly in the viewer by activating the Offsets button in the viewer toolbar.

The tracker offset icon in the upper left of the viewer is used to offset the tracking pattern from the intended object.
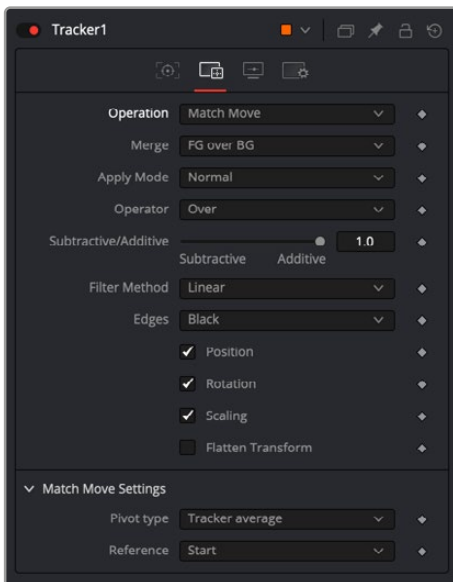
The Tracker Operation tab

Operation Tab

While the Trackers tab controls let you customize how the Tracker node analyzes motion to create motion paths, the Operation tab puts the analyzed motion data to use, performing image transforms of various kinds.

The Tracker node is capable of performing a wide variety of functions, from match moving an object into a moving scene, smoothing out a shaky camera movement, or replacing the content of a sign. Use the options and buttons in the Operation tab to select the function performed by the Tracker node.

Operation Menu

The Operation menu contains four functions performed by the Tracker. The remaining controls in this tab fine-tune the result of this selection.

— **None:** The Tracker performs no additional operation on the image beyond merely locating and tracking the chosen pattern. This is the default mode, used to create a path that will then drive another parameter on another node.

— **Match Move:** When only the orange background input is connected, this mode stabilizes the image. When a foreground image is connected to the green foreground input, the foreground image matches the position, rotation, and scaling based on the tracking patterns. Stabilizing and match move require a minimum of one tracking pattern to determine position, and two or more to determine scaling and rotation.

— **Corner Positioning:** The Corner Positioner mode tracks the four corners of a rectangular object and replaces the contents with a new image. This function requires a minimum of four tracking patterns. If there are not enough tracking patterns, new tracking patterns are added until the total equals four.

— **Perspective Positioning:** This mode is the inverse of the Corner Positioning mode. Rather than replacing the contents of the rectangle, the four trackers are mapped to the four corners of the image. This is generally used to remove perspective from an image. Like the Corner Positioning mode, this mode requires four tracking patterns, which automatically get added if there are fewer patterns.

Additional Layering Controls

When you choose any operation other than None, a series of additional controls appear.

Merge

The Merge control determines what is done (if anything) with the image provided to the green Foreground input of the Tracker. This menu appears when the operation is set to anything other than None.

— **BG Only:** The foreground input is ignored; only the background is affected. This is used primarily when stabilizing the background image.

— **FG Only:** The foreground input is transformed to match the movement in the background, and this transformed image is passed through the Tracker's output. This Merge technique is used when match moving one layer's motion to another layer's motion.

— **FG Over BG:** The foreground image is merged over the background image, using the Merge method described by the Apply Mode control that appears.

— **BG Over FG:** The background is merged over the foreground. This technique is often used when tracking a layer with an Alpha channel so that a more static background can be applied behind it.

Apply Mode and Operator Menus

This menu provides a variety of options that determine how the two layers should be combined. The options in this menu are identical to those found in the Merge node.

— **Apply Modes:** The Apply Mode setting determines the math used when blending or combining the foreground and background pixels.

  — **Normal:** The Default merge mode uses the foreground's alpha channel as a mask to determine which pixels are transparent and which are not. When this is active, another menu shows possible operations, including Over, In, Held Out, Atop, and XOr.

  — **Screen:** Screen merges the images based on a multiplication of their color values. The alpha channel is ignored, and layer order becomes irrelevant. The resulting color is always lighter. Screening with black leaves the color unchanged, whereas screening with white will always produce white. This effect creates a similar look to projecting several film frames onto the same surface. When this is active, another menu shows possible operations, including Over, In, Held Out, Atop, and XOr.

  — **Dissolve:** Dissolve mixes two image sequences together. It uses a calculated average of the two images to perform the mixture.

— **Multiply:** Multiplies the values of a color channel. This will give the appearance of darkening the image as the values are scaled from 0 to 1. White has a value of 1, so the result would be the same. Gray has a value of 0.5, so the result would be a darker image or, in other words, an image half as bright.

— **Overlay:** Overlay multiplies or screens the color values of the foreground image, depending on the color values of the background image. Patterns or colors overlay the existing pixels while preserving the highlights and shadows of the color values of the background image. The background image is not replaced but is mixed with the foreground image to reflect the original lightness or darkness of the background image.

— **Soft Light:** Soft Light darkens or lightens the foreground image, depending on the color values of the background image. The effect is similar to shining a diffused spotlight on the image.

— **Hard Light:** Hard Light multiplies or screens the color values of the foreground image, depending on the color values of the background image. The effect is similar to shining a harsh spotlight on the image.

— **Color Dodge:** Color Dodge uses the foreground's color values to brighten the background image. This is similar to the photographic practice of dodging by reducing the exposure of an area of a print.

— **Color Burn:** Color Burn uses the foreground's color values to darken the background image. This is similar to the photographic practice of burning by increasing the exposure of an area of a print.

— **Darken:** Darken looks at the color information in each channel and selects the background or foreground image's color value, whichever is darker, as the result color. Pixels lighter than the merged colors are replaced, and pixels darker than the merged color do not change.

— **Lighten:** Lighten looks at the color information in each channel and selects the background or foreground image's color values, whichever is lighter, as the result color value. Pixels darker than the merged color are replaced, and pixels lighter than the merged color do not change.

— **Difference:** Difference looks at the color information in each channel and subtracts the foreground color values from the background color values or the background from the foreground, depending on which has the greater brightness value. Merging with white inverts the color. Merging with black produces no change.

— **Exclusion:** Exclusion creates an effect similar to but lower in contrast than the Difference mode. Merging with white inverts the base color values. Merging with black produces no change.

— **Hue:** Hue creates a result color with the luminance and saturation of the background color values and the hue of the foreground color values.

— **Saturation:** Saturation creates a result color with the luminance and hue of the base color and the saturation of the blend color.

— **Color:** Color creates a result color with the luminance of the background color value and the hue and saturation of the foreground. This preserves the gray levels in the image and is useful for coloring monochrome images.

— **Luminosity:** Luminosity creates a color with the hue and saturation of the background color and the luminance of the foreground color. This mode creates an inverse effect from that of the Color mode.

— **Operator Modes:** This menu is used to select the Operation Mode of the merge. It determines how the foreground and background are combined to produce a result. This drop-down menu is visible only when the Merge node's Apply Mode is set to either Normal or Screen.

> **NOTE:** For an excellent description of the math underlying the Operation modes, read "Compositing Digital Images," Porter, T., and T. Duff, SIGGRAPH 84 proceedings, pages 253-259. Essentially, the math is as described below.

> **TIP:** Some modes not listed in the Operator drop-down menu (Under, In, Held In, Below) are easily obtained by swapping the foreground and background inputs and choosing a corresponding mode.

The formula used to combine pixels in the merge is always fg * x + bg * y. The different operations determine exactly what x and y are, as shown in the description for each mode.

The Operator modes are as follows:

— **Over:** The Over mode adds the foreground layer to the background layer by replacing the pixels in the background with the pixels from the Z wherever the foreground's alpha channel is greater than 1.

```
x = 1, y = 1-[foreground Alpha]
```

— **In:** The In mode multiplies the alpha channel of the background input against the pixels in the foreground. The color channels of the foreground input are ignored. Only pixels from the foreground are seen in the final output. This essentially clips the foreground using the mask from the background.

```
x = [background Alpha], y = 0
```

— **Held Out:** Held Out is essentially the opposite of the In operation. The pixels in the foreground image are multiplied against the inverted alpha channel of the background image. You can accomplish exactly the same result using the In operation and a Matte Control node to invert the matte channel of the background image.

```
x = 1-[background Alpha], y = 0
```

— **ATop:** ATop places the foreground over the background only where the background has a matte.

```
x = [background Alpha], y = 1-[foreground Alpha]
```

— **XOr:** XOr combines the foreground with the background wherever either the foreground or the background have a matte, but never where both have a matte.

```
x = 1-[background Alpha], y = 1-[foreground Alpha]
```

— **Subtractive/Additive:** This slider controls whether Fusion performs an Additive merge, a Subtractive merge, or a blend of both. This slider defaults to Additive merging for most operations, assuming the input images are premultiplied (which is usually the case). If you don't understand the difference between Additive and Subtractive merging, here's a quick explanation:

  — An Additive merge is necessary when the foreground image is premultiplied, meaning that the pixels in the color channels have been multiplied by the pixels in the alpha channel. The result is that transparent pixels are always black since any number multiplied by 0 always equals 0. This obscures the background (by multiplying with the inverse of the foreground alpha), and then simply adds the pixels from the foreground.

  — A Subtractive merge is necessary if the foreground image is not premultiplied. The compositing method is similar to an Additive merge, but the foreground image is first multiplied by its alpha to eliminate any background pixels outside the alpha area.

In most software applications, you will find the Additive/Subtractive option displayed as a simple checkbox. Fusion lets you blend between the Additive and Subtractive versions of the merge operation, which is occasionally useful for dealing with problem composites with edges that are calling attention to themselves as too bright or too dark.

> For example, using a Subtractive setting on a premultiplied image may result in darker edges. Using an Additive setting with a non-premultiplied image may result in lightening the edges. By blending between Additive and Subtractive, you can tweak the edge brightness to be just right for your situation.

Filter Method (Match Move)

Determines which filter to use to handle image transforms made using the Tracker node. This menu appears only when the Operation Mode is set to Match Move.

— **Box:** This is a simple interpolation resize of the image.

— **Linear:** This uses a simplistic filter, which produces relatively clean and fast results.

— **Quadratic:** This filter produces a nominal result. It offers a good compromise between speed and quality.

— **Cubic:** This produces better results with continuous-tone images but is slower than Bi-Cubic. If the images have fine detail in them, the results may be blurrier than desired.

— **Catmull-Rom:** This produces good results with continuous-tone images that are resized down. It produces sharp results with finely detailed images.

— **Gaussian:** This is very similar in speed and quality to Bi-Cubic.

— **Mitchell:** This is similar to Catmull-Rom but produces better results with finely detailed images. It is slower than Catmull-Rom.

— **Lanczos:** This is very similar to Mitchell and Catmull-Rom but is a little cleaner and also slower.

— **Sinc:** This is an advanced filter that produces very sharp, detailed results; however, it may produce visible "ringing" in some situations.

— **Bessel:** This is similar to the Sinc filter but may be slightly faster.

Edges

This menu selects how the revealed edges are handled when the image is moved to match position and scaling.

— **Black Edges:** Out-of-frame edges revealed by Stabilization are left black.
— **Wrap:** Portions of the image moved off frame to one side are used to fill edges that are revealed on the opposite side.
— **Duplicate:** The last valid pixel on an edge is repeated to the edge of the frame.
— **Mirror:** Image pixels are mirrored to fill to the edge of the frame.

Position, Rotation, and Scaling Checkboxes (Match Move)

The Position, Rotation, and Scaling checkboxes appear only when the mode is set to Match Move. They determine what components of motion that Stabilization will attempt to correct in the image. For example, if only the Position checkbox is selected, no attempt will be made to correct for Rotation and Scaling in the image.

Flatten Transformation (Match Move)

This checkbox appears only when the mode is set to Match Move. Like most transformations in Fusion, Stabilization is concatenated with other sequential transformations by default. Selecting this checkbox will flatten the transform, breaking any concatenation taking place and applying the transform immediately.

Mapping Type

The Mapping Type control appears only in the Corner Positioning mode. There are two options in the menu:

— **Bi_Linear:** The first method is Bi-Linear, where the foreground image is mapped into the background without any attempt to correct for perspective distortion. This is identical to how previous versions of Fusion operated.
— **Perspective:** The foreground image is mapped into the background taking perspective distortion into account. This is the preferred setting since it maps better to the real world than the older Bi-Linear setting.

Corner Selector (Corner or Perspective Positioning)

When the operation of the Tracker is set to either Corner or Perspective Positioning modes, four drop-down menus appear. These options choose which trackers map to each of the four corners of the rectangle. This is useful when a Tracker has more than four patterns selected, and you must choose which patterns the positioners use.

Rotate Clockwise and Counter-Clockwise Buttons (Corner or Perspective Positioning)

These controls appear only when the operation of the Tracker is set to either Corner or Perspective Positioning modes. They are used to rotate the foreground image by 90 degrees before it is applied to the background.

Stabilize Settings

The Tracker node automatically outputs several steady and unsteady position outputs to which other controls in the Node Editor can be connected. The Stable Position output provides X and Y coordinates to match or reverse motion in a sequence. These controls are available even when the operation is not set to Match Move, since the Stable Position output is always available for connection to other nodes.

Match Move Settings

These settings determine how tracking data is correlated with the reference pattern for making transforms.

Pivot Type

The Pivot type menu determines how the anchor point for rotation is selected.

— **Tracker Average:** Averages the location based on the tracking points.
— **Selected Tracker:** Provides a menu where one of the current trackers can be selected as the pivot point.
— **Manual:** Displays X and Y position number fields where you can manually position the pivot points.

Reference

The Reference mode determines the "snapshot frame" based on the frame where the pattern is first selected. All Stabilization is intended to return the image back to that reference.

— **Select Time:** Lets you select the current frame.
— **Start:** The Snapshot Frame is determined to be the first frame in the tracked path. All Stabilization is intended to return the image back to that reference.
— **Start and End:** The Start and End Reference mode is somewhat different from all other Reference modes. Where the others are intended to take a snapshot frame to which all stabilization returns, immobilizing the image, the Start and End mode is intended to smooth existing motion, without removing it. This mode averages the motion between the Start and End of the path, drawing a straight line between those points.

  When this mode is active, it reveals the Reference Intermediate Points control. Increasing the value of this control increases the number of points in the path used by the Reference, smoothing the motion from a straight line between Start and End without making it wholly linear.

— **End:** The Snapshot Frame is determined to be the last frame in the tracked path. All Stabilization is intended to return the image back to that reference.

> **TIP:** By default, the Tracker displays a single displacement path of the tracked data in the Spline Editor. To view X and Y paths of the tracked points in the Spline Editor, go to Preferences > Globals > Splines.

Display Options Tab

The Display Options tab lets you customize the look of onscreen controls in the viewer.
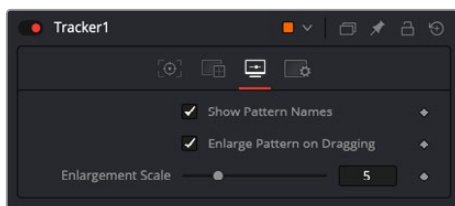
The Tracker Display Options tab

Show Pattern Names

This option defines whether the Tracker's pattern names will be displayed in the viewer. Switch it off to see the pattern rectangle instead.

Enlarged Pattern on Dragging

This option defines whether there is a magnified thumbnail view when positioning the pattern rectangle.

Enlargement Scale

The zoom factor that is used when positioning the pattern rectangle when the above option is activated.

> **TIP:** The outputs of a tracker (seen in the Connect to... menu) can also be used by scripts. They are:
>
> — SteadyPosition: Steady Position
>
> — UnsteadyPosition: Unsteady Position
>
> — SteadyAxis: Steady Axis
>
> — SteadySize: Steady Size
>
> — UnsteadySize: Unsteady Size
>
> — SteadyAngle: Steady Angle
>
> — UnsteadyAngle: Unsteady Angle
>
> — Position1: Tracker 1 Offset position
>
> — PerspectivePosition1: Tracker 1 Perspective Offset position
>
> — PositionX1: Tracker 1 Offset X position (3D Space)
>
> — PositionY1: Tracker 1 Offset Y position (3D Space)
>
> — PerspectivePositionX1: Tracker 1 Perspective Offset X position (3D Space)
>
> — PerspectivePositionY1: Tracker 1 Perspective Offset Y position (3D Space)
>
> — SteadyPosition1: Tracker 1 Steady Position
>
> — UnsteadyPosition1: Tracker 1 Unsteady Position (likewise for the 2nd, 3rd, and so on)

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Tracking nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Planar Tracker Node [PTra]

The Planar Tracker node

Planar Tracker Node Introduction

The Planar Tracker node is designed to solve a match-moving problem that commonly comes up during post-production. As an example, live-action footage can often contain a planar surface such as a license plate or a road sign that needs new numbers in the license plate or a new city's name on the road sign. Often, the problem is that the camera is moving in the shot, so the license plate or road sign is continuously changing perspective. You cannot just merge a new license plate over the existing one without accounting for the perspective distortions. A time-intensive way to solve this problem would be to use a Corner Pin node and manually keyframe the four corners. The Planar Tracker automates this keyframing process and tracks the perspective distortions of a planar surface over time. That tracking data is then applied with those same perspective distortions to a different foreground.

For more information on using the Planar Tracker, *see Chapter 83, "Planar Tracking,"* in the DaVinci Resolve Reference Manual, or Chapter 21 in the Fusion Reference Manual.

> **TIP:** Part of using a Planar Tracker is also knowing when to give up and fall back to using Fusion's Tracker node or to manual keyframing. Some shots are simply not trackable, or the resulting track suffers from too much jitter or drift. The Planar Tracker is a useful time-saving node in the artist's toolbox, but while it may track most shots, it is not a 100% solution.

What the Planar Tracker Saves

While the Planar Tracker does save the resulting final track in the composition on disk, it does not save temporary tracking information such as the individual point trackers (compared with the Camera Tracker, which does save the individual point trackers). Some consequences of this include:

— The point trackers no longer appear in the viewer when a comp containing a Planar Tracker node is saved and reloaded.

— Tracking may not be resumed after a comp containing a Planar Tracker node has been saved and reloaded. In particular, this also applies to auto saves. For this reason, it is good to complete all planar tracking within one session.

— The size of composition files is kept reasonable (in some situations, a Planar Tracker can produce hundreds of megabytes of temporary tracking data).

— Saving and loading of compositions is faster and more interactive.

Inputs

— **The Planar Tracker has four inputs:**

— **Background:** The orange background image input accepts a 2D image with the planar surface to be tracked.

— **Corner Pin 1:** The green corner pin 1 input accepts a 2D image to be pinned on top of the background. There may be multiple corner pin inputs, named Corner Pin 1, Corner Pin 2,...etc.

— **Occlusion Mask:** The white occlusion mask input is used to mask out regions that do not need to be tracked. Regions where this mask is white will not be tracked. For example, a person moving in front of and occluding bits of the pattern may be confusing the tracker, and a quickly-created rough rotomask around the person can be used to tell the tracker to ignore the masked-out bits.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the output of the Planar Tracker to certain areas.

Basic Node Setup

A basic Planar Tracker setup consists of just two nodes: a MediaIn connects to the background input and the Planar Tracker can be used as a separate branch from the rest of the node tree. Once the tracking is completed, a Planar Transform node should be generated to use the planar tracking data.

A Planar Tracker can be isolated on
its own branch of a node tree.

A Typical Planar Tracker Workflow

The following steps outline the workflow with the Planar Tracker:

1. **Remove lens distortion:** The more lens distortion in the footage, the more the resulting track will slide and wobble.

2. **Connect footage:** Connect a Loader or MediaIn node that contains a planar surface to the orange background input and view the Planar Tracker node in a viewer.

3. **Select a reference frame:** Move to a frame where the planar surface to be tracked is not occluded and click the Set button to set this as a reference frame.

4. **Choose the pattern:** In the viewer, make sure the onscreen controls are visible, and draw a polygon around the planar surface you want to track. This is called the "pattern." In most cases, this will probably be a rectangle, but an arbitrary closed polygon can be used. The pixels enclosed by this region will serve as the pattern that will be searched for on other frames. Note that it is important that the pattern is drawn on the reference frame. Do not confuse the pattern with the region to corner pin (which always has four corners and is separately specified in Corner Pin mode).

5 Adjust render range: In the Keyframes Editor, adjust the render range to match the range of frames where the planar surface is visible.

6 Adjust track options: Frequently changed options include Tracker, Motion Type, and Track Channel.

7 Mask out occluders: If moving objects partially cover up the planar surface, you may wish to connect an occlusion mask to the Planar Tracker. When using the Hybrid tracker, providing a mask to deal with occluding objects is strongly recommended, while with the Point tracker it is recommended to try tracking without a mask.

8 Track: Click the Go button to return to the reference frame. Press the Track To End button and wait for the track to complete. Click the Go button to return to the reference frame again. Press the Track To Start button and wait for the track to complete. Note that the tracks in the viewer are not selectable or deletable as they are in a Camera Tracker.

9 Check track quality: Visually inspect the track to see how accurate it is. Does it stick to the surface? Switching to Steady mode can help here.

10 Use the track: At this point, in most cases you will create a Planar Transform node from the Inspector and use it to apply the tracked perspective distortion onto masked images. If the image you are applying the tracking data to is full frame unmasked clip, you can use the Steady, Corner Pin, and Stabilize operation modes in in the Planar Tracker.

Inspector
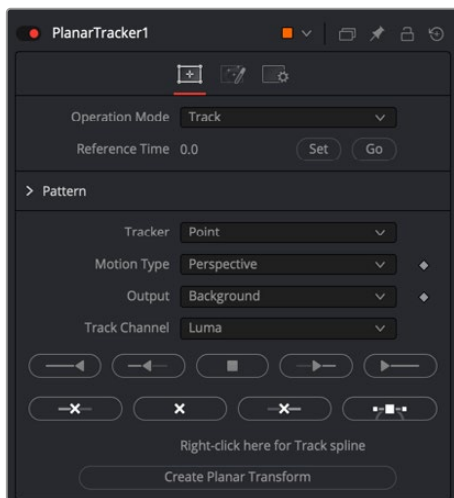
The Planar Tracker Controls tab

Controls Tab

The Controls tab contains controls for determining how the Planar Tracker will be used, setting the reference frame and initiating the track.

Operation Mode

The Operation Mode menu selects the purpose of the Planar Tracker node. The Planar Tracker has four modes of operation:

— **Track:** Used to isolate a planar surface and track its movement over time. Then, you can create a Planar Transform node that uses this data to match move another clip in various ways.

— **Steady:** After analyzing a planar surface, this mode removes all motion and distortions from the planar surface, usually in preparation for some kind of paint or roto task, prior to "unsteadying" the clip to restore the motion.

— **Corner Pin:** After analyzing a planar surface, this mode computes and applies a matching perspective distortion to a foreground image you connect to the foreground input of the Planar Tracker node, and merges it on top of the tracked footage.

— **Stabilize:** After analyzing a planar surface, this mode allows smoothing of a clip's translation, rotation, and scale over time. This is good for getting unwanted vibrations out of a clip while retaining the overall camera motion that was intended.

The last three modes (Steady, Corner Pin, and Stabilize) use the tracking data produced in Track mode.

> **NOTE:** None of the operations can be combined together. For example, both Corner Pin and Stabilize cannot be done at the same time, nor can a track be done while in corner pinning mode.

Reference Time

The Reference Time determines the frame where the pattern is outlined. It is also the time from which tracking begins. The reference frame cannot be changed once it has been set without destroying all pre-existing tracking information, so scrub through the footage to be tracked and choose carefully. The reference frame must be chosen carefully to give the best possible quality track.

You choose a reference frame by moving the playhead to an appropriate frame and then clicking the Set button to choose that frame.

Pattern Polygon

You specify which region of the image you want to track by drawing a polygon on the reference frame. Typically, when you first add a Planar Tracker node, you are immediately ready to start drawing a polygon in the viewer, so it's best to do this right away. When choosing where to draw a polygon, make sure the region selected belongs to a physically planar surface in the shot. In a pinch, a region that is only approximately planar can be used, but the less planar the surface, the poorer the quality of the resulting track.

As a rule of thumb, the more pixels in the pattern, the better the quality of the track. In particular, this means the reference frame pattern should be:

— As large as possible.

— As much in frame as possible.

— As unoccluded as possible by any moving foreground objects.

— At its maximal size (e.g., when tracking an approaching road sign, it is good to pick a later frame where it is 400 x 200 pixels big rather than 80 x 40 pixels).

— Relatively undistorted (e.g., when the camera orbits around a flat stop sign, it is better to pick a frame where the sign is face-on parallel to the camera rather than a frame where it is at a highly oblique angle).

If the pattern contains too few pixels or not enough trackable features, this can cause problems with the resulting track, such as jitter, wobble, and slippage. Sometimes dropping down to a simpler motion type can help in this situation.

After you've drawn a pattern, a set of Pattern parameters lets you transform and invert the resulting polygon, if necessary.

Track Mode

Track mode is unlike the other three options in the Operation menu in that is the only option that initiates the planar tracking. The other modes use the tracking data generated by the Track mode.

Tracker

There are two available trackers to pick from:

— **Point:** Tracks points from frame to frame. Internally, this tracker does not actually track points-per-se but rather small patterns like Fusion's Tracker node. The point tracker possesses the ability to automatically create its internal occlusion mask to detect and reject outlier tracks that do not belong to the dominant motion. Tracks are colored green or red in the viewer, depending on whether the point tracker thinks they belong to the dominant motion or they have been rejected. The user can optionally supply an external occlusion mask to further guide the Point tracker.

— **Hybrid Point/Area:** Uses an Area tracker to track all the pixels in the pattern. Unlike the Point tracker, the Area tracker does not possess the ability to automatically reject parts of the pattern that do not belong to the dominant motion, so you must manually provide it with an occlusion mask. Note that for performance reasons, the Hybrid tracker internally first runs the Point tracker, which is why the point tracks can still be seen in the viewer.

  There is no best tracker. They each have their advantages and disadvantages:

  — **Artist Effort (occlusion masks):** The Point tracker will automatically create its internal occlusion mask. However, with the Hybrid tracker, you need to spend more time manually creating occlusion masks.

  — **Accuracy:** The Hybrid tracker is more accurate and less prone to wobble, jitter, and drift since it tracks all the pixels in the pattern rather than a few salient feature points.

  — **Speed:** The Hybrid tracker is slower than the Point tracker.

In general, it is recommended to first quickly track the shot with the Point tracker and examine the results. If the results are not good enough, then try the Hybrid tracker.

Motion Type

Determines how the Planar Tracker internally models the distortion of the planar surface being tracked. The five distortion models are:

— Translation.
— Translation, Rotation (rigid motions).
— Translation, Rotation, Scale (takes squares to squares, scale is uniform in x and y).
— Affine includes translation, rotation, scale, skew (maps squares to parallelograms).
— Perspective (maps squares to generic quadrilaterals).

Each successive model is more general and includes all previous models as a special case.

When in doubt, choose Perspective for the initial track attempt. If the footage being tracked has perspective distortions in it, and the Planar Tracker is forced to work with a simpler motion type, this can end up causing the track to slide and wobble.

Sometimes with troublesome shots, it can help to drop down to a simpler motion model—for example, when many track points are clustered on one side of the tracked region or when tracking a small region where there are not many trackable pixels.

Output

Controls what is output from the Planar Tracker node while in the Track operation mode.

— **Background:** Outputs the input image unchanged.
— **Background - Preprocessed:** The Planar Tracker does various types of preprocessing on the input image (e.g., converting it to luma) before tracking. It can be useful to see this when deciding which track channel to choose.
— **Mask:** Outputs the pattern as a black and white mask.
— **Mask Over Background:** Outputs the pattern mask merged over the background.

Track Channel

Determines which image channel in the background image is tracked. It is good to pick a channel with high contrast, lots of trackable features, and low noise. Allowed values are red, green, blue, and luminance.

Tracking Controls

These controls are used to control the Tracker. Note that while tracking, only track to a new frame if the current frame is already tracked or it is the reference frame.

— **Track to start:** Tracks from the current frame backward in time to the start (as determined by the current render range).
— **Step tracker to previous frame:** Tracks from the current frame to the previous frame.
— **Stop tracking:** Stops any ongoing tracking operations.
— **Step tracker to next frame:** Tracks from the current frame to the next frame.
— **Track to end:** Tracks from the current frame forward in time to the end (as determined by the current render range).
— **Trim to start:** Removes all tracking data before the current frame.
— **Delete:** Deletes all tracking data at all times. Use this to destroy all current results and start tracking from scratch.
— **Trim to end:** Removes all tracking data after the current frame. This can be useful, for example, to trim the end of a track that has become inaccurate when the pattern starts to move off frame.

Show Splines

This button to the right of the "Trim to end" button opens the Spline Editor and shows the splines associated with the Planar Tracker node. This can be useful for manually deleting points from the Track and Stable Track splines.

Right-Click Here for Track Spline

While tracking, a spline containing 4 x 4 matrices at each keypoint is created. This is known as the "Track spline" or just "Track" for short. These matrices completely describe the distortions of the tracked pattern.

Create Planar Transform

After tracking footage, this button can be pressed to create a Planar Transform node on the Node Editor. The current tracking data is embedded in the Planar Transform node so that it can replicate the planar distortions tracked by the Planar Tracker node. Unless you are compositing a full frame foreground that matches the same dimensions as the raster, it is best to create a Planar Transform and use it to apply motion to the foreground.

Steady Mode

In Steady mode, the Planar Tracker transforms the background plate to keep the pattern as motionless as possible. Any leftover motion is because the Planar Tracker failed to follow the pattern accurately or because the pattern did not belong to a physically planar surface.

Steady mode is not very useful for actual stabilization, but is useful for checking the quality of a track. If the track is good, during playback the pattern should not move at all while the rest of the background plate distorts around it. It can be helpful to zoom in on parts of the pattern and place the mouse cursor over a feature and see how far that feature drifts away from the mouse cursor over time.

Steady Time

This is the time where the pattern's position is snapshotted and frozen in place. It is most common to set this to the reference frame.

Invert Steady Transform

This causes the Planar Tracker node to reverse the effects of the steady transform. This means two Planar Tracker nodes connected back to back with the second set to invert should give back the original image. If you place an effects node in between the two, then the effect will be locked in place. This should only be used to accomplish effects that cannot be done through corner pinning, since it involves two resamplings, causing softening of the background image.

Clipping Mode

Determines what happens to the parts of the background image that get moved off frame by the steady transform:

— **Domain:** The off frame parts are kept.
— **Frame:** The off frames parts are thrown away.

Domain mode is useful when Steady mode is being used to "lock" an effect to the pattern. As an example, consider painting on the license plate of a moving car. One way to do this is to use a Planar Tracker node to steady the license plate, then a Paint node to paint on the license plate, and then a second Planar Tracker to undo the steady transform. If the Clipping mode is set to Domain, the off frame parts generated by the first Planar Tracker are preserved so that the second Planar Tracker can, in turn, map them back into the frame.

Corner Pin Mode

In Corner Pin mode, one or more textures can be attached to a previously tracked planar surface and undergo the same perspective distortions as the surface.

The corner pin workflow with Planar Tracker is:

1 **Track:** select a planar surface in the shot that you wish to attach a texture to or replace the texture on. Track the shot (see the tracking workflow in the Track section).

2 **Switch the Operation Mode to Corner Pin:** When Corner Pin mode is entered from Track mode, the pattern polygon is hidden and a corner pin control is shown in the viewer.

3 **Connect in the texture:** In the Node Editor, connect the output of the MediaIn node containing the texture to the Corner Pin 1 input on the Planar Tracker node.

4 **Adjust corner pin:** Drag the corners of the corner pin in the viewer until the texture is positioned correctly. Sometimes the Show Grid option is useful when positioning the texture. Additionally, if it helps to position it more accurately, scrub to other times and make adjustments to the corner pin.

5 **Review: P**lay back the footage and make sure the texture "sticks" to the planar surface.

Merge Mode

Controls how the foreground (the corner pinned texture) is merged over the background (the tracked footage). If there are multiple corner pins, this option is shared by all of them. There are four options to pick from:

— BG only

— FG only

— FG over BG

— BG over FG

Number of Corner Pins

Use the + and - buttons to increase or decrease the number of corner pins. Each time an additional corner pin is created, a corresponding input appears on the node in the Node Editor.

Corner Pin 1 Input Group

Each corner pin has a group of related inputs:

— **Enable:** Controls the visibility of the corner pin in the viewer.

— **Show Grid:** Shows a grid over the corner pin. This can be useful when positioning the corners.

— **Merge Options:** Controls merging of corner pin texture over the background. (See the documentation for the Merge node.)

— **Reference Time Positions:** The positions of the four corners at the reference time. If the track was not perfect, these positions can be animated to make adjustments on top of the track.

Stabilize Mode

Stabilize mode is used to smooth out shakiness in the camera by applying a transform that partially counteracts the camera shake. This stabilizing transform (contained in the Stable Track spline) is computed by comparing neighboring frames.

> **NOTE:** Stabilize mode only smooths out motions, while Steady mode tries to completely "lock off" all motion.

Be aware that the Planar Tracker stabilizes based on the motion of the pattern, so it is essential to choose the pattern carefully. If the motion of the pattern does not represent the motion of the camera, then there may be unexpected results. For example, if tracking the side of a moving truck and the camera is moving alongside it, the Planar Tracker smooths the combined motion of both the truck and the mounted camera. In some cases, this is not the desired effect. It may be better to choose the pattern to be on some fixed object like the road or the side of a building, which would result in smoothing only the motion of the camera.

One unavoidable side effect of the stabilization process is that transparent edges appear along the edges of the image. These edges appear because the stabilizer does not have any information about what lies off frame, so it cannot fill in the missing bits. The Planar Tracker node offers the option to either crop or zoom away these edges. When filming, if the need for post-production stabilization is anticipated, it can sometimes be useful to film at a higher resolution (or lower zoom).

The Planar Tracker Stabilization Workflow

1   **Track:** Select a roughly planar region that represents the motion that you want to stabilize. Track the shot (see the tracking workflow in the Track section).

2   **Switch the Operation Mode to Stabilize:** Until a stabilization is computed, the Planar Tracker will just output the input footage.

3   **Adjust stabilization options:** Frequently changed options are Parameters to Smooth and Smoothing Radius.

4   **Compute stabilization:** Press the Compute Stabilization button and wait for the stabilization computations to finish. Play back the output of the Planar Tracker node to see the effects of the stabilization. Notice that transparent edges have been introduced around the edges of the image by the stabilization transform.

5   **Refine:** Adjust the stabilization options and recompute the stabilization as many times as desired.

6   **Handle transparent edges (optional):** Set the Frame Mode to either Zoom or Crop as desired and then click the Auto Zoom or Auto Crop button. Play back the footage to observe the effects. If there is too much zoom or the image has been cropped too small, try reducing the amount of smoothing.

Parameters to Smooth

Specify which of the following parameters to smooth:

— X Translation
— Y Translation
— Rotation
— Scale

Smoothing Window

When stabilizing a particular frame, this determines how the contributions of neighboring frames are weighted. Available choices are Box and Gaussian.

Smoothing Radius (Frames)

Determines the number of frames whose transforms are averaged together to compute the stabilization. A larger Smoothing Radius results in more smoothing but introduces more transparent edges.

Compute Stabilization

Clicking this button runs the stabilizer, overwriting the results of any previous stabilization. As soon as the stabilization is finished, the output of the Planar Tracker node will be immediately updated with the stabilization applied.

> **NOTE:** The stabilizer uses the Track spline (created by the tracker) to produce the Stable Track spline. Both of these splines' keyframes contain 4 x 4 matrices, and the keyframes are editable in the Spline Editor.

Clipping Mode

Determines what happens to the parts of the background image that get moved off frame by the stabilization:

— **Domain:** The off frame parts are kept.
— **Frame:** The off frames parts are thrown away.

Frame Mode

This controls how transparent edges are handled. The available options include:

— **Full:** Do nothing. Leaves the transparent edges as is.
— **Crop:** Crops away the transparent edges. When this option is selected, the size of Planar Tracker's output image is smaller than the input image. No image resamplings occur. In Crop mode, use the Auto Crop button or manually adjust the crop window by changing the X Offset, Y Offset, and Scale sliders.
  — **Auto Crop Button:** When this button is clicked, the Planar Tracker will examine all the frames and pick the largest possible crop window that removes all the transparent edges. The computed crop window will always be centered in frame and pixel aligned. When clicked, Auto Crop updates the X/Y Offset and Scale sliders.
— **Zoom:** Scales the image bigger until the transparent edges are off frame. Choosing this option causes an image resampling to occur. The downside of this approach is that it reduces the quality (slightly softens) of the output image. In Zoom mode, use the Auto Zoom button or manually adjust the zoom window by changing the X Offset, Y Offset, and Scale sliders.
  — **Auto Zoom:** When this button is clicked, the Planar Tracker will examine all the frames and pick the smallest possible zoom factor that removes all the transparent edges. The computed zoom window will always be centered in frame. When clicked, Auto Zoom updates the X/Y Offset and Scale sliders.

Right-Click Here for Stable Track Spline

Right-clicking over this label provides access to a spline whose keyframes contain 4 x 4 matrices which in turn represent the stabilization transforms. This is mostly here for completeness and for advanced users.

Options Tab

These controls affect the look of onscreen controls in the viewer.
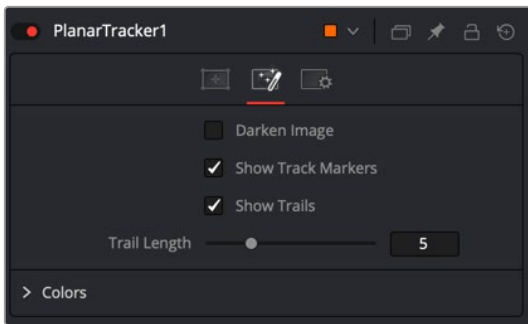
The Planar Tracker Options tab

Darken Image

Darkens the image while in Track mode in order to better see the controls and tracks in the viewer. The Shift+D keyboard shortcut toggles this.

Show Track Markers

Toggles the display of the dots marking the location of trackers at the current time.

Show Trails

Toggles the display of the trails following the location of trackers.

Trail Length

Allows changing the length of tracker trails. If the pattern is moving very slowly, increasing the length can sometimes make the trails easier to follow in the viewer. If the pattern is moving very fast, the tracks can look like spaghetti in the viewer. Decreasing the length can help.

Inlier/Outlier Colors

When tracking, the tracker analyzes the frame and detects which of the multitudinous tracks belong to the dominant motion and which ones represent anomalous, unexplainable motion. By default, tracks belonging to the dominant motion are colored green (and are called inliers) and those that do not belong are colored red (and are called outliers). Only the inlier tracks are used when computing the final resulting track.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Tracking nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Planar Transform Node [PXF]


The Planar Transform node

The Planar Transform node applies perspective distortions generated by a Planar Tracker node onto any input mask or masked image. The Planar Transform node can be used to reduce the amount of time spent on rotoscoping objects. The workflow here centers around the notion that the Planar Tracker node can be used to track objects that are only roughly planar. After an object is tracked, a Planar Transform node can then be used to warp a rotospline, making it approximately follow the object over time. Fine-level cleanup work on the rotospline then must be done.

Depending on how well the Planar Tracker followed the object, this can result in substantial time savings in the amount of tedious rotoscoping. The key to using this technique is recognizing situations where the Planar Tracker performs well on an object that needs to be rotoscoped.

A rough outline of the workflow involved is:

1   Track: Using a Planar Tracker node, select a pattern that represents the object to be rotoscoped. Track the shot (see the tracking workflow in the Track section for the Planar Tracker node).

2   Create a Planar Transform node: Press the Create Planar Transform button on the Planar Tracker node to do this. The newly created Planar Transform node can be freely cut and pasted into another composition as desired.

3   Rotoscope the object: Move to any frame that was tracked by the Planar Tracker. When unsure if a frame was tracked, look in the Spline Editor for a tracking keyframe on the Planar Transform node. Connect a Polygon node into the Planar Transform node. While viewing the Planar Transform node, rotoscope the object.

4   Refine: Scrub the timeline to see how well the polygon follows the object. Adjust the polyline on frames where it is off. It is possible to add new points to further refine the polygon.

Inputs

The Planar Transform has only two inputs:

— **Image Input:** The orange image input accepts a 2D image on which the transform will be applied.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the output of the Planar Transform to certain areas.

Basic Node Setup

The example below uses a Planar Transform node between a masked MediaIn2 node and the foreground input to a Merge node. The background MediaIn1 node connects to the Planar Tracker, which was used to generate the Planar Transform. Once the Planar Transform is created, the Planar Tracker is no longer needed in the node tree.

A Planar Transform creating a match move

Inspector
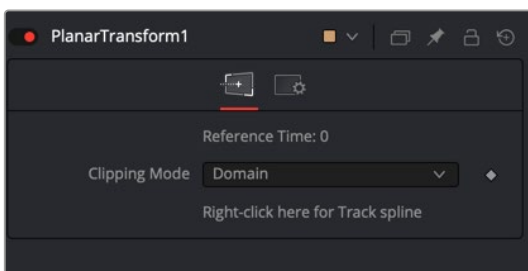
The Planar Transform Controls tab

Controls Tab

The Planar Transform node has very few controls, and they are all located in the Controls tab. It's designed to apply the analyzed Planar Tracking data as a match move,

Reference Time

This is the reference time that the pattern was taken from in the Planar Tracker node used to produce the Planar Transform.

Right-Click Here for Track Spline

The Track spline contains information about the perspective distortions stored in 4 x 4 matrices. When a Planar Transform node is exported from a Planar Tracker node, the Track spline produced by the Planar Tracker is shared by connecting it with the Planar Transform node. A consequence of this sharing of the Track spline is that if the track is changed in the Planar Tracker node, the Planar Transform will be automatically updated. Note that this spline can be examined in the Spline Editor, which is useful for seeing the extent of tracked frames.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Tracking nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Camera Tracker [CTra]

The Camera Tracker node

Camera Tracker Introduction

Camera tracking is match moving, and a vital link between 2D and 3D, allowing compositors to integrate 3D renders into live-action scenes. The Camera Tracker node is used to calculate the path of a live-action camera and generate a virtual camera in 3D space. This virtual camera's motion is intended to be identical to the motion of the actual camera that shot the scene. Using the calculated position and movement of the virtual camera provides the flexibility to add 3D elements to a live-action scene. Also, the Camera Tracker creates a point cloud in 3D space that can be used to align objects and other 3D models to the live-action scene.

For more information about how to use the Camera Tracker, *see Chapter 86, "3D Camera Tracking,"* in the DaVinci Resolve Reference Manual, or Chapter 24 in the Fusion Reference Manual.
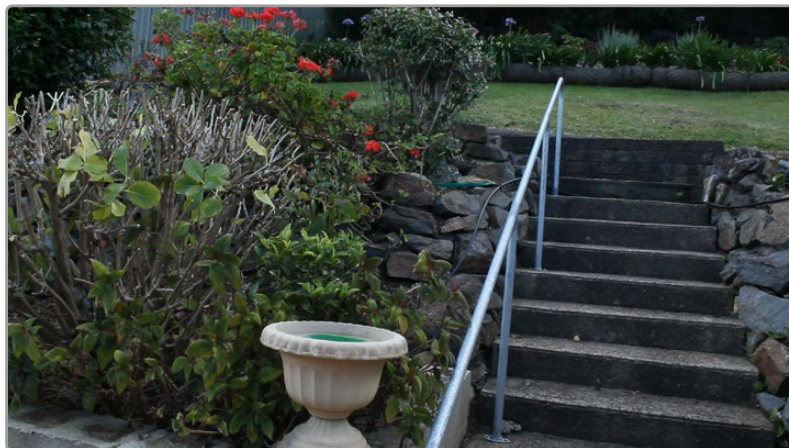
The original video clip

3D objects composited on to the video clip that use the camera tracker to remain aligned with the objects in the frame as the image moves

Inputs

The Camera Tracker has two inputs:

— **Background:** The orange image input accepts a 2D image you want tracked.

— **Occlusion Mask:** The white occlusion mask input is used to mask out regions that do not need to be tracked. Regions where this mask is white will not be tracked. For example, a person moving in front of and occluding bits of the scene may be confusing to the tracker, and a quickly-created rough mask around the person can be used to tell the tracker to ignore the masked-out bits.

Basic Node Setup

The Camera Tracker background input is used to connect the image you want tracked. Polygon masks can be connected into the occlusion mask input to identify areas the tracker should ignore.
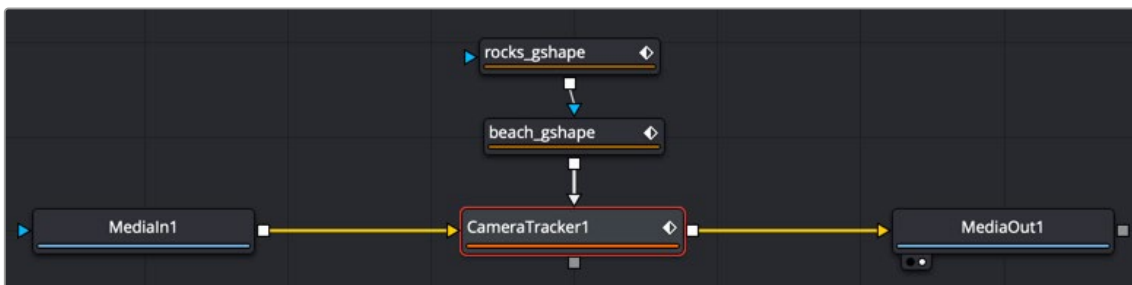
The Camera Tracker with occlusion masks

Inspector
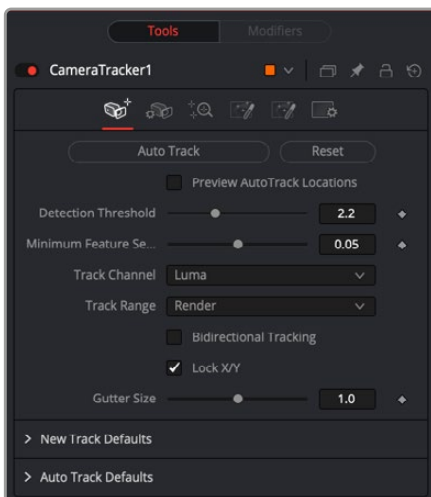
The Camera Tracker tab

Track Tab

The Track tab contains the controls you need to set up an initial analysis of the scene.

Auto Track

Automatically detects trackable features and tracks them through the source footage. Tracks will be automatically terminated when the track error becomes too high, and new tracks are created as

needed. The values of the Detection Threshold and Minimum Feature Separation sliders can be used to control the number and distribution of auto tracks.

Reset

Deletes all the data internal to the Camera Tracker node, including the tracking data and the solve data (camera motion path and point cloud). To delete only the solve data, use the Delete button on the Solve tab.

Preview AutoTrack Locations

Turning this checkbox on will show where the auto tracks will be distributed within the shot. This is helpful for determining if the Detection Threshold and Minimum Feature Separation need to be adjusted to get an even spread of trackers.

Detection Threshold

Determines the sensitivity to detect features. Automatically generated tracks will be assigned to the shot and the Detection Threshold will force them to be either in locations of high contrast or low contrast.

Minimum Feature Separation

Determines the spacing between the automatically generated tracking points. Decreasing this slider causes more auto tracks to be generated. Keep in mind that a large number of tracking points will also result in a lengthier solve.

Track Channel

Used to nominate a color channel to track: red, green, blue, or luminance. When nominating a channel, choose one that has a high level of contrast and detail.

Track Range

Used to determine which frames are tracked:

— **Global:** The global range, which is the full duration of the Timeline.
— **Render:** The render duration set on the Timeline.
— **Valid:** The valid range is the duration of the source media.
— **Custom:** A user determined range. When this is selected, a separate range slider appears to set the start and end of the track range.

Bidirectional Tracking

Enabling this will force the tracker to track backward after the initial forward tracking. When tracking backward, new tracks are not started but rather existing tracks are extended backward in time. It is recommended to leave this option on, as long tracks help give better solved cameras and point clouds.

Gutter Size

Trackers can become unstable when they get close to the edge of the image and either drift or jitter or completely lose their pattern. The Camera Tracker will automatically terminate any tracks that enter the gutter region. Gutter size is given as a percentage of pattern size. By default, it's 100% of pattern size, so a 0.04 pattern means a 0.04 gutter.

New Track Defaults

There are three methods in which the Camera Tracker node can analyze the scene, and each has its own strengths when dealing with certain types of camera movement.

— **Tracker:** Internally, all the Trackers use the Optical Flow Tracker to follow features over time and then further refine the tracks with the trusted Fusion Tracker or Planar Tracker. The Planar Tracker method allows the pattern to warp over time by various types of transforms to find the best fit.

These transforms are:

— Translation

— Translation and Rotation Translation, Rotation, and Scale Affine

— Perspective

It is recommended to use the default TRS setting when using the Planar Tracker. The Affine and Perspective settings need large patterns in order to track accurately.

— **Close Tracks When Track Error Exceeds:** Tracks will be automatically terminated when the tracking error gets too high. When tracking a feature, a snapshot of the pixels around a feature are taken at the reference time of the track. This is called a pattern, and that same pattern of pixels is searched for at future times. The difference between the current time pattern and the reference time pattern is called the track error. Setting this option higher produces longer but increasingly less accurate tracks.

— **Solve Weight:** By default, each track is weighted evenly in the solve process. Increasing a track's weight means it has a stronger effect on the solved camera path. This is an advanced option that should be rarely changed.

Auto Track Defaults

Set a custom prefix name and/or color for the automatically generated tracks. This custom color will be visible when Track Colors in the Options tab is set to User Assigned.

Camera Tab
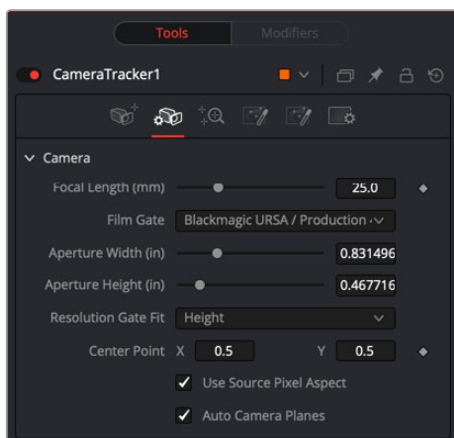
The Camera Tracker Camera tab

The controls of the Camera tab let you specify the physical aspects of the live-action camera, which will be used as a starting point when searching for solve parameters that match the real-world camera. The more accurate the information provided in this section, the more accurate the camera solve.

The Camera tab includes controls relating to the lens and gate aspects of the camera being solved for.

Focal Length

Specify the known constant focal length used to shoot the scene or provide a guess if the Refine Focal Length option is activated in the Solve tab.

Film Gate

Choose a film gate preset from the drop-down menu or manually enter the film back size in the Aperture Width and Aperture Height inputs. Note that these values are in inches.

Aperture Width

In the event that the camera used to shoot the scene is not in the preset drop-down menu, manually enter the aperture width (inches).

Aperture Height

In the event that the camera used to shoot the scene is not in the preset drop-down menu, manually enter the aperture height (inches).

Resolution Gate Fit

This defines how the image fits the sensor size. Often, film sensors are sized to cover a number of formats, and only a portion of the sensor area is recorded into an image.

> **EXAMPLE:** A 16:9 image is saved out of a full aperture-sized sensor.
>
> Typically, fit to Width or Height is the best setting. The other fit modes are Inside, Outside, or Stretched.

Center Point

This is where the camera lens is aligned to the camera. The default is (0.5, 0.5), which is the middle of the sensor.

Use Source Pixel Aspect

This will use the squeeze aspect of the pixels that is loaded in the image. HD is square pixels, but NTSC has a pixel aspect ratio of 0.9:1, and Anamorphic CinemaScope is 2:1 aspect. Disabling this option exposes Pixel X and Y number fields where you can customize the source pixel aspect.

Auto Camera Planes

When enabled, the camera's image plane and far plane are automatically moved to enclose the point cloud whenever a solve completes. Sometimes, though, the solver can atypically fling points off really deep into the scene, consequently pushing the image plane very far out. This makes the resulting scene unwieldy to work with in the 3D views. In these cases, disable this option to override this default behavior (or delete the offending tracks).

Solve Tab
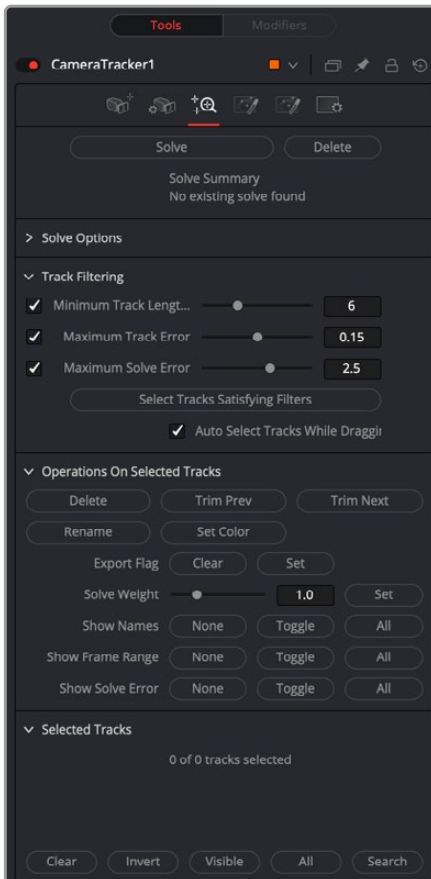
The Camera Tracker Solve tab

SolveTab

The Solve tab is where the tracking data is used to reconstruct the camera's motion path along with the point cloud. It is also where cleanup of bad or false tracks is done, and other operations on the tracks can be performed, such as defining which marks are exported in the Point Cloud 3D. The markers can also have their weight set to affect the solve calculations.

For example, a good camera solve may have already been generated, but there are not enough locators in the point cloud in an area where an object needs to be placed, so adding more tracks and setting their Solve Weight to zero will not affect the solved camera but will give more points in the point cloud.

Solve

Pressing Solve will launch the solver, which uses the tracking information and the camera specifications to generate a virtual camera path and point cloud, approximating the motion of the physical camera in the live-action footage. The console will automatically open, displaying the progress of the solver.

Delete

Delete will remove any solved information, such as the camera and the point cloud, but will keep all the tracking data.

Average Solve Error

Once the camera has been solved, a summary of the solve calculation is displayed at the top of the Inspector. Chief among those details is the Average Solve Error. This number is a good indicator of whether the camera solve was successful. It can be thought of as the difference (measured in pixels) between tracks in the 2D image and the reconstructed 3D locators reprojected back onto the image through the reconstructed camera. Ultimately, in trying to achieve a low solve error, any value less than 1.0 pixels will generally result in a good track. A value between 0.6 and 0.8 pixels is considered excellent.

Clean Tracks by Filter

Clicking this button selects tracks based on the Track Filtering options. If the Auto Delete Tracks By Filter checkbox is activated, the selected tracks will be deleted as well.

Clean Foreground Tracks

Clicking this button makes a selection of the tracks on fast-moving objects that would otherwise cause a high solve error. The selection is determined by the Foreground Threshold slider.

Foreground Threshold

This slider sets the detection threshold for finding the tracks on fast-moving objects. The higher the value, the more forgiving.

Auto Delete Tracks by Filter

With this checkbox enabled, tracks that are selected by the Clean Tracks By Filter button will be deleted. Enable the checkbox, and then press Clean Tracks By Filter. Any track that meets the filtering options is then selected and deleted.

Auto Delete Foreground Tracks

With this checkbox enabled, tracks that are selected by the Clean Foreground Tracks button will be deleted. Enable the checkbox, and then press Clean Foreground Tracks. Any track that meets the foreground threshold criteria is deleted.

Accept Solve Error

This slider sets an acceptable maximum threshold level for the solve error. If the solve error is greater than this value, the Camera Tracker will sweep the focal length setting in an attempt to bring the solve error under the Accept Solve Error value. If the solver cannot find a solution, the Camera Tracker will display a message in the console that the solver failed. If a solution cannot be found, ideally you should try to input the correct focal length or alternatively manually clean some noisy tracks then re-solve.

Auto Select Seed Frames

With this enabled, the Camera Tracker nominates two frames that will be used as a reference for initiating the solve. These two frames are initially solved for and a camera is reconstructed, and then gradually more frames are added in, and the solution is "grown" outward from the seed frames. The choice of seed frames strongly affects the entire solve and can easily cause the solve to fail. Seed frames can be found automatically or defined manually.

Disabling this will allow the user to select their own two frames. Manual choice of seed frames is an option for advanced users. When choosing seed frames, it is important to satisfy two conflicting desires: the seed frames should have lots of tracks in common yet be far apart in perspective (i.e., the baseline distance between the two associated cameras is long).

Refine Focal Length

Enabling this will allow the solver to adjust the focal length of the lens to match the tracking points. You can prevent the focal length being adjusted by setting the Focal Length parameter in the Camera Tab.

Enable Lens Parameter

When enabled, lens distortion parameters are exposed to help in correcting lens distortion when solving.

— **Refine Center Point:** Normally disabled, camera lenses are normally centered in the middle of the film gate but this may differ on some cameras For example, a cine camera may be set up for Academy 1.85, which has a sound stripe on the left, and shooting super35, the lens is offset to the right.

— **Refine Lens Parameters:** This will refine the lens distortion or curvature of the lens. There tends to be larger distortion on wide angle cameras

> **NOTE:** When solving for the camera's motion path, a simulated lens is internally created to model lens distortion in the source footage. This simulated lens model is much simpler than real-world lenses but captures the lens distortion characteristics important for getting an accurate camera solve. Two types of distortion are modeled by Camera Tracker:
>
> **Radial Distortion:** The strength of this type of distortion varies depending on the distance from the center of the lens. Examples of this include pincushion, barrel, and mustache distortion. Larger values correspond to larger lens curvatures. Modeling radial distortion is especially important for wide angle lenses and fisheye lenses (which will have a lot of distortion because they capture 180 degrees of an environment and then optically squeeze it onto a flat rectangular sensor).
>
> **Tangential Distortion:** This kind of distortion is produced when the camera's imaging sensor and physical lens are not parallel to each other. It tends to produce skew distortions in the footage similar to distortions that can be produced by dragging the corners of a corner pin within Fusion. This kind of distortion occurs in very cheap consumer cameras and is practically non-existent in film cameras, DSLRs, and pretty much any kind of camera used in film or broadcast. It is recommended that it be left disabled.

Enable Lens Parameters

When disabled, the Camera Tracker does not do any lens curvature simulations. This is the default setting and should remain disabled if there is a very low distortion lens or the lens distortion has already been removed from the source clip. Activating the Enable Lens Parameters checkbox determines which lens parameters will be modeled and solved for. Parameters that are not enabled will be left at their default values. The following options are available:

— **Radial Quadratic:** Model only Quadratic radial lens curvature, which is either barrel or pincushion distortion. This is the most common type of distortion. Selecting this option causes the low and high order distortion values to be solved for.

— **Radial Quartic:** Model only Quartic radial lens curvature, which combines barrel and pincushion distortion. This causes the low and high order distortion values to be solved for.

— **Radial & Tangential:** Model and solve for both radial and tangential distortion. Tangential relates to misaligned elements in a lens.

— **Division Quadratic:** Provides a more accurate simulation of Quadratic radial lens curvature. This causes the low and high order distortion values to be solved for.

— **Division Quartic:** Provides a more accurate simulation of Quartic radial lens curvature. This causes the low and high order distortion values to be solved for.

  — **Lower Order Radial Distortion:** This slider is available for all simulations. It determines the quadratic lens curvature.

  — **Higher Order Radial Distortion:** This slider is available only for Quartic simulations. Determines the quartic lens curvature.

  — **Tangential Distortion X/Y:** These sliders are available only for Tangential simulations. Determines skew distortion.

Track Filtering

The Camera Tracker can produce a large number of automatically generated tracks. Rather than spending a lot of time individually examining the quality of each track, it is useful to have some less time-intensive ways to filter out large swaths of potentially bad tracks. The following input sliders are useful for selecting large amounts of tracks based on certain quality metrics, and then a number of different possible operations can be made on them. For example, weaker tracks can selected and deleted, yielding a stronger set of tracks to solve from. Each filter can be individually enabled or disabled.

Minimum Track Length (Number of Markers)

Selects tracks that have a duration shorter than the slider's value. Short tracks usually don't get a chance to move very far and thus provide less perspective information to the solver than a longer track, yet both short and long tracks are weighted evenly in the solve process, making long tracks more valuable to the solver. Locators corresponding to shorter tracks are also less accurately positioned in 3D space than those corresponding to longer tracks. If the shot has a lot of long tracks, it can be helpful to delete the short tracks. For typical shots, using a value in the range of 5 to 10 is suggested. If there are not a lot of long tracks (e.g., the camera is quickly rotating, causing tracks to start and move off frame quickly), using a value closer to 3 is recommended.

Maximum Track Error

Selects tracks that have an average track error greater than the slider's value. When tracking, tracks are automatically terminated when their track error exceeds some threshold. This auto termination controls the maximum track error, while this slider controls the average track error. For example, tracks following the foliage in a tree tend to be inaccurate and sometimes may be detected by their high average error.

Maximum Solve Error

Selects tracks that have a solve error greater than the slider's value. One of the easiest ways to increase the accuracy of a camera solve is to select the 20% of the tracks with the highest solve error and simply delete them (although this can sometimes make things worse).

Select Tracks Satisfying Filters

Selects the tracks within the scene that meet the above Track Filtering values. Note that when this button is pressed, the tracks that satisfy the filter values are displayed in the Selected Tracks area of the Solve tab and are colored in the viewer. This button is useful when Auto Select Tracks While Dragging Sliders is turned off or if the selection, for example, was accidentally lost by mis-clicking in the viewer.

Auto Select Tracks While Dragging Sliders

When this is ticked on, dragging the above sliders (Minimum Track Length, Maximum Track Error, Maximum Solve Error) will cause the corresponding tracks to be interactively selected in the viewer.

Operations on Selected Tracks

Tracks selected directly in the viewer with the mouse or selected via track filtering can have the following operations applied:

Delete	Will remove the tracks from the set. When there are bad tracks, the simplest and easiest option is to simply delete them.
Trim Previous	Will cut the tracked frames from the current frame to the start of the track. Sometimes it can be more useful to trim a track than deleting it. For example, high quality long tracks that become inaccurate when the feature they are tracking starts to become occluded or when the tracked feature moves too close to the edge of the image.
Trim Next	Will cut the tracked frames from the current frame to the end of the track.
Rename	Will replace the current auto generated name with a new name.
Set Color	Will allow for user assigned color of the tracking points.
Export Flag	This controls whether the locators corresponding to the selected tracks will be exported in the point cloud. By default all locators flagged as exportable.
Solve Weight	By default, all the tracks are used and equally weighted when solving for the camera's motion path. The most common use of this option is to set a track's weight to zero so it does not influence the camera's motion path but is still has a reconstructed 3D locator. Setting a tracks' weight to values other than 1.0 or 0.0 should only be done by advanced users. Onscreen display of track names and values are controlled by these functions: — **None:** Will clear/hide the selected tracks. — **Toggle:** Will swap the selected tracks and unselect sets. — **All:** Will select all tracks. — **Show Names:** Will display the track name, by default these are a number. — **Show Frame Range:** Will display the start and end frame of a track. — **Show Solve Error:** Will display the amount of solve error each selected track has.

Selected Tracks

This area displays the properties of a track point or group of points. It has options to:

— **Clear:** Deselects all tracks and clears this area.

— **Invert:** Deselects the current selected tracks and selects the other tracks.

— **Visible:** Selects all the trackers at the current frame.

— **All:** Selects all trackers on all frames.

— **Search:** Selects tracks whose names contain a substring.

> **TIP:** Also select tracks directly in the 2D viewer using the mouse or in the 3D viewer by selecting their corresponding locators in the point cloud.

Export Tab

The Export tab lets you turn the tracked and solved data this node has generated into a form that can be used for compositing.
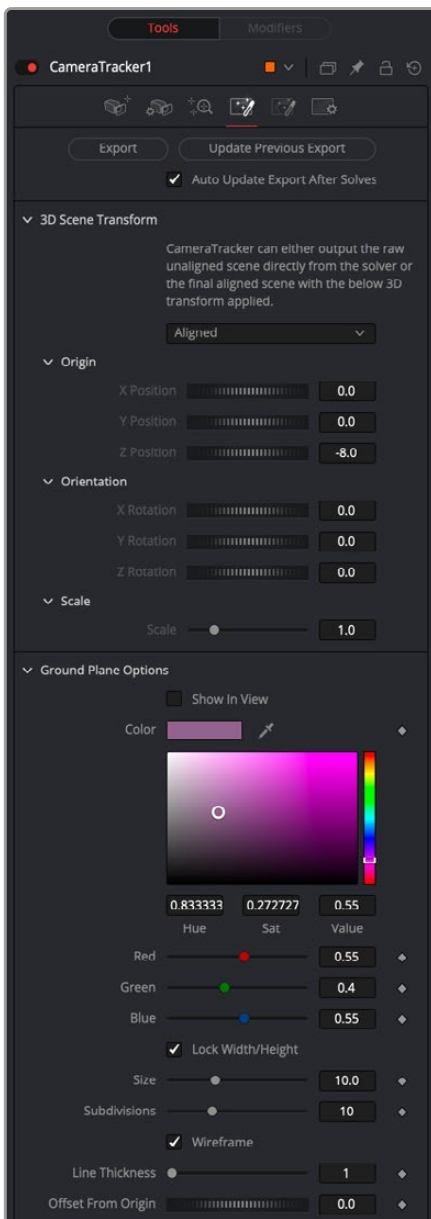
The Camera Tracker Export tab

Export

The Export button will create a basic setup that can be used for 3D match moving:

— A Camera 3D with animated translation and rotation that matches the motion of the live-action camera and an attached image plane.

— A Point Cloud 3D containing the reconstructed 3D positions of the tracks.

— A Shape 3D set to generate a ground plane.

— A Merge 3D merging together the camera, point cloud, and ground plane. When the Merge 3D is viewed through the camera in a 3D viewer, the 3D locators should follow the tracked footage.

— A Renderer 3D set to match the input footage.

The export of individual nodes can be enabled/disabled in the Export Options tab.

Update Previous Export

When this button is clicked, the previously exported nodes are updated with any new data generated. These previously exported nodes are remembered in the Previous Export section at the bottom of this section. Here's an example of how this is handy:

1 Solve the camera and export.

2 Construct a complex Node Editor based around the exported nodes for use in set extension.

3 The camera is not as accurate as preferred or perhaps the solver is rerun to add additional tracks to generate a denser point cloud. Rather than re-exporting the Camera 3D and Point Cloud 3D nodes and connecting them back in, just press the Update Previous Export button to "overwrite" the existing nodes in place.

Automatically Update Previous Export After Solves

This will cause the already exported nodes (Camera 3D, Point Cloud 3D, Lens Distort, Renderer 3D, and the ground plane) to auto update on each solve.

3D Scene Transform

Although the camera is solved, it has no idea where the ground plane or center of the scene is located. By default, the solver will always place the camera in Fusion's 3D virtual environment so that on the first frame it is located at the origin (0, 0, 0) and is looking down the -Z axis. You have the choice to export this raw scene without giving the Camera Tracker any more information, or you can set the ground plane and origin to simplify your job when you begin working in the 3D scene. The 3D Scene Transform controls provide a mechanism to correctly orient the physical ground plane in the footage with the virtual ground plane in the 3D viewer. Adjusting the 3D Scene Transform does not modify the camera solve but simply repositions the 3D scene to best represent the position of the live-action camera.

> **NOTE:** If you export the scene and then make changes in the 3D Scene Transform, it is important to manually click Update Previous Export to see the results in the exported nodes.

Aligned/Unaligned

The Aligned/Unaligned menu locks or unlocks the origin and ground plane settings. When set to Unaligned, you can select the ground plane and origin either manually or by selecting locators in the viewer. When in unaligned mode, a 3D Transform control in the 3D viewer can be manually manipulated to adjust the origin.

Once alignment of the ground plane and origin has been completed, the section is locked by switching the menu to Aligned.

Set from Selection

When set to unaligned, buttons labeled Set from Selection are displayed under the Origin, Orientation, and Scale sections. Clicking these buttons takes the selecting locators in the viewer and aligns the ground plane or origin based on the selection.

For instance, to set the ground plane, do the following:

1 After solving, set the 3D Scene Transform menu to Unaligned.

2 Find a frame where the ground plane is at its largest and clearest point.

3 In the viewer, drag a selection rectangle around all the ground plane locators.

4 Hold Shift and drag again to add to the selection.

5 In the Orientation section, make sure the Selection Is menu correctly matches the orientation of the selected locators.

6 Click the Set from Selection button located under the Orientation parameters.

7 Set the 3D Scene Transform menu back to Aligned.

To get the best result when setting the ground plane, try to select as many points as possible belonging to the ground and having a wide separation.

> **TIP:** When selecting points for the ground plane, it is helpful to have the Camera Tracker node viewed in side-by-side 2D and 3D views. It may be easier to select tracks belonging to the ground by selecting tracks from multiple frames in the 2D viewer rather than trying to box select locators in the 3D viewer.

Setting the origin can help you place 3D objects in the scene with more precision. To set the origin, you can follow similar steps, but only one locator is required for the origin to be set. When selecting a locator for the origin, select one that has a very low solve error.

Ground Plane Options

These controls let you adjust the ground plane for the scene, which is a crucial step in making sure the composite looks correct.

Color	Will set the color of the ground plane.
Size	Controls how big the ground plane can be set.
Subdivision Level	Shows how many polygons are in the ground plane.

Wireframe	Sets whether the ground plane is set as wireframe or solid surface when displayed in 3D.
Line Thickness	Adjusts how wide the lines will draw in the view.
Offset	By default, the center of the ground plane is placed at the origin (0, 0, 0). This can be used to shift the ground plane up and down along the y-axis.

Export Options

Provides a checkbox list of what will be exported as nodes when the Export button is pressed. These options are Camera, Point Cloud, Ground Plane, Renderer, Lens Distortion, and Enable Image Plane in the camera.

The Animation menu allows you to choose between animating the camera and animating the point cloud. Animating the camera leaves the point cloud in a locked position while the camera is keyframed to match the live-action shot. Animating the point cloud does the opposite. The camera is locked in position while the entire point cloud is keyframed to match the live-action shot.

Previous Export

When the Update Previous Export button is clicked, the previously exported nodes listed here are updated with any new data generated (this includes the camera path and attributes, the point cloud, and the renderer).

Options Tab

The Options tab lets you customize the Camera Tracker's onscreen controls so you can work most effectively with the scene material you have.
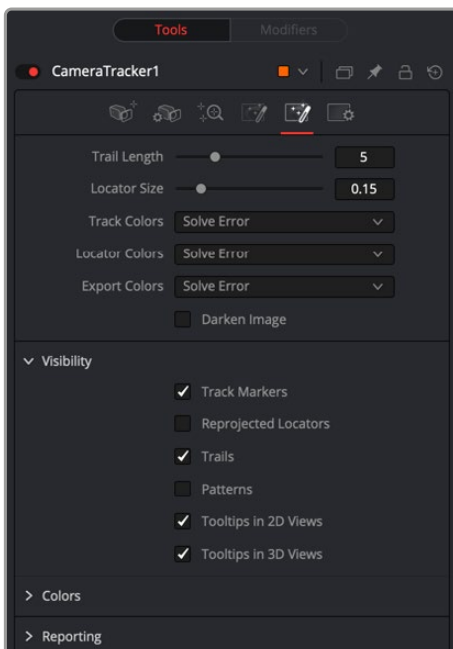
The Camera Tracker Options tab

Trail Length

Displays trail lines of the tracks overlaid on the viewer. The amount of frames forward and back from the current frame is set by length.

Location Size

In the 3D viewer, the point cloud locators can be sized by this control.

Track Colors, Locator Colors, and Export Colors each have options for setting their color to one of the following:

— User Assigned

— Solve Error

— Take From Image

— White

Track Colors	Onscreen tracks in the 2D view.
Locator Colors	Point Cloud locators in the 3D view.
Export Colors	Colors of the locators that get exported within the Point Cloud node.

Darken Image

Dims the brightness of the image in viewers to better see the overlaid tracks. This affects both the 2D and 3D viewers.

Visibility

Toggles which overlays will be displayed in the 2D and 3D viewers. The options are Tracker Markers, Trails, Tooltips in the 2D Viewer, Tooltips in the 3D viewer, Reprojected Locators, and Tracker Patterns.

Colors

Sets the color of the overlays.

— **Selection Color:** Controls the color of selected tracks/locators.

— **Preview New Tracks Color:** Controls the color of the points displayed in the viewer when the Preview AutoTrack Locations option is enabled.

— **Solve Error Gradient:** By default, tracks and locators are colored by a green-yellow-red gradient to indicate their solve error. This gradient is completely user adjustable.

Reporting

Outputs various parameters and information to the Console.

Understanding Camera Tracking

On large productions, camera tracking or 3D match moving is often handed over to experts who have experience with the process of tracking and solving difficult shots. There is rarely a shot where you can press a couple of buttons and have it work perfectly. It does take an understanding of the whole process and what is essential to get a good solved track.

The Camera Tracker must solve for hundreds of thousands of unknown variables, which is a complex task. For the process to work, it is essential to get good tracking data that exists in the shot for a long time. False or bad tracks will skew the result. This section explains how to clean up false tracks and other techniques to get a good solve.

Workflow

Getting a good solve is a repeated process.

> Track > Solve > Refine Filters > Solve > Cleanup tracks > Solve > Cleanup from point cloud > Solve > Repeat.

Initially, there are numerous tracks, and not all are good, so a process of filtering and cleaning up unwanted tracks to get to the best set is required. At the end of each cleanup stage, pressing Solve ideally gives you a progressively lower solve error. This needs to be below 1.0 for it to be good for use with full HD content, and even lower for higher resolutions. Refining the tracks often but not always results in a better solve.

False Tracks

False tracks are caused by a number of conditions, such as moving objects in a shot, or reflections and highlights from a car. There are other types of false tracks like parallax errors where two objects are at different depths, and the intersection gets tracked. These moiré effects can cause the track to creep. Recognizing these False tracks and eliminating them is the most important step in the solve process.

Track Lengths

Getting a good set of long tracks is essential; the longer the tracks are, the better the solve. The Bi-Directional tracking option in the Tracker tab is used to extend the beginning of tracks in time. The longer in time a track exists and the more tracks that overlap in time of a shot, the more consistent and accurate the solve.

Seed Frames

Two seed frames are used in the solve process. The algorithm chooses two frames that are as far apart in time yet share the same tracks. That is why longer tracks make a more significant difference in the selection of seed frames.

The two Seed frames are used as the reference frames, which should be from different angles of the same scene. The solve process will use these as a master starting point to fit the rest of the tracks in the sequence.

There is an option in the Solve tab to Auto Detect Seed Frames, which is the default setting and most often a good idea. However, auto detecting seed frames can make for a longer solve. When refining the Trackers and re-solving, disable the checkbox and use the Seed 1 and Seed 2 sliders to enter the previous solve's seed frames. These seed frames can be found in the Solve Summary at the top of the Inspector after the initial solve.

Refine Filters

After the first solve, all the Trackers will have extra data generated. These are solve errors and tracking errors.

Use the refine filters to reduce unwanted tracks, like setting minimum tracker length to eight frames. As the value for each filter is adjusted, the Solve dialog will indicate how many tracks are affected by the filter. Then Solve again.

Onscreen Culling

Under the Options tab, set the track to 20. This will display each track on footage with +-20 frame trail. When scrubbing/playing through the footage, false tracks can be seen and selected onscreen, and deleted by pressing the Delete key. This process takes an experienced eye to spot tracks that go bad. Then Solve again.

You can view the exported scene in a 3D perspective viewer. The point cloud will be visible. Move and pan around the point cloud, select and delete points that seem to have no inline with the image and the scene space. Then Solve again.

Repeat the process until the solve error is below 1.0 before exporting.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Tracking nodes. These common controls are described in detail in the following "The Common Controls" section.

The Common Controls

Nodes that handle tracking operations share several identical controls in the Inspector. This section describes controls that are common among Tracking nodes.

Inspector
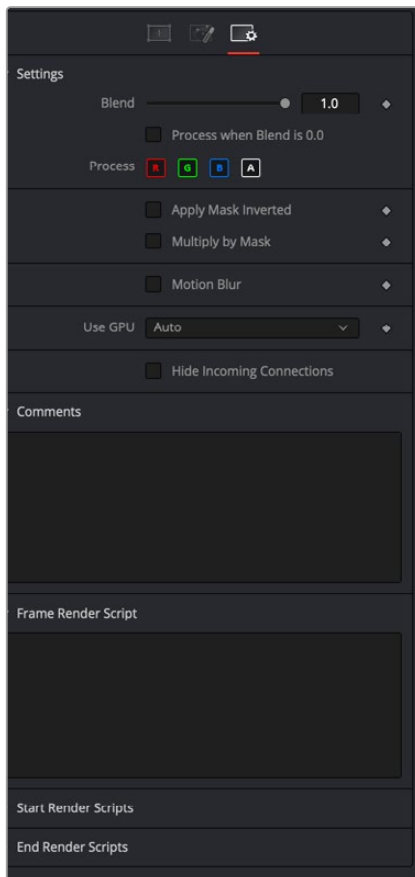
The Common Tracking Controls Settings tab

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Tracking category. The controls are consistent and work the same way for each tool.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this will cause the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if the node is scripted to trigger a task, but the node's value is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the Red button on a Blur tool is deselected, the blur will first be applied to the image, and then the red channel from the original input will be copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this will generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option will cause the RGB values of the masked image to be multiplied by the mask channel's values. This will cause all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object ID and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels will be used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option is disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID will be used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the viewer. The image or sequence must have been rendered from a 3D software package with those channels included.

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 will cause Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one full frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows for the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off GPU hardware-accelerated rendering. Enabled uses the GPU hardware for rendering the node. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node will be displayed in the Inspector. Dragging a connected node from the node tree into the field will hide that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line will reappear.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Transform Nodes

This chapter details the Transform nodes available in Fusion.

The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Camera Shake [CSh]

The Camera Shake node

Camera Shake Node Introduction

This node can simulate a variety of camera shake-style motions from organic to mechanical. It is not the same as the Shake Modifier, which generates random number values for parameters.

For more information on the Shake modifier, *see Chapter 122, "Modifiers,"* in the DaVinci Resolve Reference Manual, or Chapter 60 in the Fusion Reference Manual.

The Camera Shake node concatenates its result with adjacent transformation nodes for higher-quality processing.

Inputs

The two inputs on the Camera Shake node are used to connect a 2D image and an effect mask, which can be used to limit the camera shake area.

— **Input:** The orange input is used for the primary 2D image that shakes.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the camera shake area to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Camera Shake background input is used to connect the image you want to transform. Polygon masks can be connected into the occlusion mask input to identify areas the camera shake should ignore.
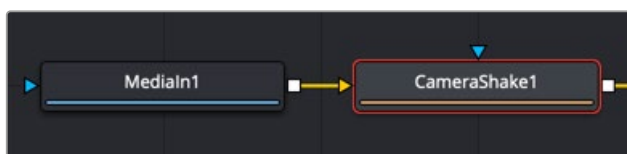
The Camera Shake node can be connected directly after a MediaIn node or any node providing a 2D output.

Inspector
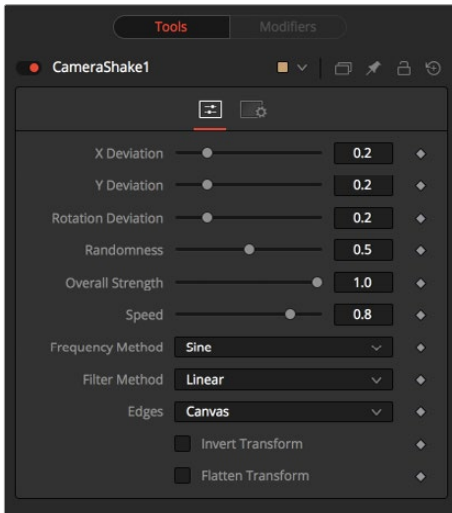
The Camera Shake Controls tab

Controls Tab

The Controls tab includes parameters for adjusting the offsets, strength, speed, and frequency of the simulated camera shake movement.

Deviation X and Y

These controls determine the amount of shake applied to the image along the horizontal (X) and vertical (Y) axes. Values between 0.0 and 1.0 are permitted. A value of 1.0 generates shake positions anywhere within the boundaries of the image.

Rotation Deviation

This determines the amount of shake that is applied to the rotational axis. Values between 0.0 and 1.0 are permitted.

Randomness

Higher values in this control cause the movement of the shake to be more irregular or random. Smaller values cause the movement to be more predictable.

Overall Strength

This adjusts the general amplitude of all the parameters and blends that affect in and out. A value of 1.0 applies the effect as described by the remainder of the controls.

Speed

Speed controls the frequency, or rate, of the shake.

Frequency Method

This selects the overall shape of the shake. Available frequencies are Sine, Rectified Sine, and Square Wave. A Square Wave generates a much more mechanical-looking motion than a Sine.

Filter Method

When rescaling a pixel, surrounding pixels are often used to give a more realistic result. There are various algorithms for combining these pixels, called filters. More complex filters can give better results but are usually slower to calculate.

The best filter for the job often depends on the amount of scaling and on the contents of the image itself.

— **Box:** This is a simple interpolation resize of the image.
— **Linear:** This uses a simplistic filter, which produces relatively clean and fast results.
— **Quadratic:** This filter produces a nominal result. It offers a good compromise between speed and quality.
— **Cubic:** This produces better results with continuous-tone images. If the images have fine detail in them, the results may be blurrier than desired.
— **Catmull-Rom:** This produces good results with continuous-tone images that are resized down. This produces sharp results with finely detailed images.
— **Gaussian:** This is very similar in speed and quality to Bi-Cubic.
— **Mitchell:** This is similar to Catmull-Rom but produces better results with finely detailed images. It is slower than Catmull-Rom.
— **Lanczos:** This is very similar to Mitchell and Catmull-Rom but is a little cleaner and also slower.
— **Sinc:** This is an advanced filter that produces very sharp, detailed results; however, it may produce visible "ringing" in some situations.
— **Bessel:** This is similar to the Sinc filter but may be slightly faster.

Window Method (Sinc and Bessel Only)

Some filters, such as Sinc and Bessel, require an infinite number of pixels to calculate exactly. To speed up this operation, a windowing function is used to approximate the filter and limit the number of pixels required. This control appears when a filter that requires windowing is selected.

— **Hanning:** This is a simple tapered window.
— **Hamming:** Hamming is a slightly tweaked version of Hanning that does not taper all the way down to zero.
— **Blackman:** A window with a more sharply tapered falloff.
— **Kaiser:** A more complex window with results between Hamming and Blackman.

Most of these filters are useful only when making an image larger. When shrinking images, it is common to use the Bi-Linear filter, however, the Catmull-Rom filter will apply some sharpening to the results and may be useful for preserving detail when scaling down an image.

EXAMPLE

Resize filters. From left to right: Nearest Neighbor, Box, Linear, Quadratic, Cubic, Catmull-Rom, Gaussian, Mitchell, Lanczos, Sinc, and Bessel.

Edges

This menu determines how the Edges of the image are treated.

— **Canvas:** This causes the edges that are revealed by the shake to be the canvas color—usually transparent or black.

— **Wrap:** This causes the edges to wrap around (the top is wrapped to the bottom, the left is wrapped to the right, and so on).

— **Duplicate:** This causes the Edges to be duplicated, causing a slight smearing effect at the edges.

— **Mirror:** Image pixels are mirrored to fill to the edge of the frame.

Invert Transform

Select this control to Invert any position, rotation, or scaling transformation. This option might be useful for exactly removing the motion produced in an upstream Camera Shake.

Flatten Transform

The Flatten Transform option prevents this node from concatenating its transformation with adjacent nodes. The node may still concatenate transforms from its input, but it will not concatenate its transformation with the node at its output.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Transform nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Crop [Crp]
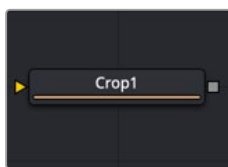
The Crop node

Crop Node Introduction

The Crop node can be used to cut out a portion of an image or to offset the image into a larger image area. However, unlike using a mask, this node actually changes the resolution of the image.

> **TIP:** You can crop an image in the viewer by activating the Allow Box Selection in the upper-left corner of the viewer while the Crop node is selected and viewed. Then, drag a crop rectangle around the area of interest to perform the operation.

> **NOTE:** Because this node changes the physical resolution of the image, animating the parameters is not advised.

Inputs

The single input on the Crop node is used to connect a 2D image for cropping.

— **Input:** The orange input is used for the primary 2D image you want to crop.

Basic Node Setup

Below, the Crop node is inserted between the MediaIn1 node and the background input of the Merge. Unlike using a mask tool, cropping the MediaIn1 changes the resolution of the clip. The cropped MediaIn1 node connected to the orange background input also sets the resolution of the Merge output.
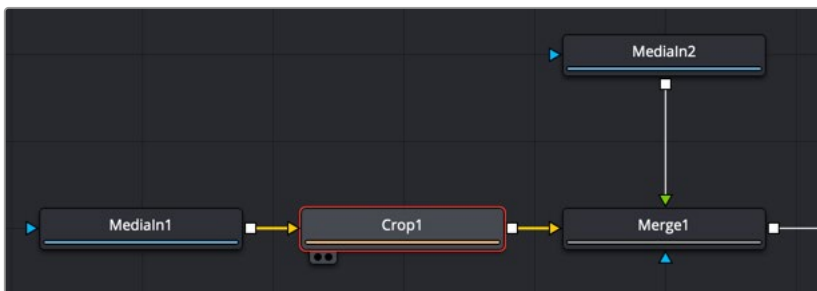
The Crop node can be used to cut out a portion of an image.

Inspector
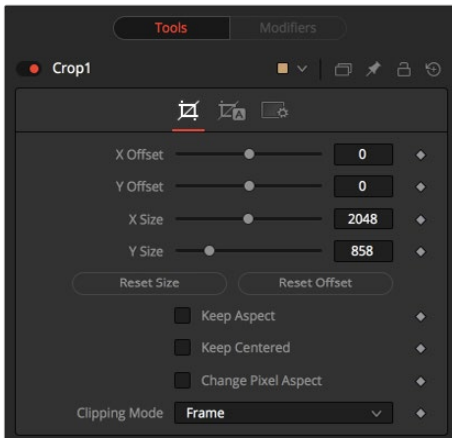
The Crop Controls tab

Controls Tab

The Controls tab provides XY Offset and XY Size methods for cropping the image.

Offset X and Y

These controls position the image off the screen by pushing it left/right or up/down. The cropped image disappears off the edges of the output image. The values of these controls are measured in pixels.

Size X and Y

Use these controls to set the vertical and horizontal resolution of the image output by the Crop node. The values of these controls are measured in pixels.

Keep Aspect

When toggled on, the Crop node maintains the aspect of the input image.

Keep Centered

When toggled on, the Crop node automatically adjusts the X and Y Offset controls to keep the image centered. The XY Offset sliders are automatically adjusted, and control over the cropping is done with the Size sliders or the Allow Box Selection button in the viewer.

Reset Size

This resets the image dimensions to the size of the input image.

Reset Offset

This resets the X and Y Offsets to their defaults.

Change Pixel Aspect

Enable this checkbox to reveal a Pixel Aspect control that can be used to change the image's pixel aspect.

Clipping Mode

This option sets the mode used to handle the edges of the image when performing domain of definition (DoD) rendering. This is profoundly important for nodes like Blur, which may require samples from portions of the image outside the current domain.

— **Frame:** The default option is Frame, which automatically sets the node's domain of definition to use the full frame of the image, effectively ignoring the current domain of definition. If the upstream DoD is smaller than the frame, the remaining area in the frame will be treated as black/transparent.

— **Domain:** Setting this option to Domain will respect the upstream DoD when applying the node's effect. This can have adverse clipping effects in situations where the node employs a large filter.

— **None:** Setting this option to None does not perform any source image clipping at all. This means that any data required to process the node's effect that would normally be outside the upstream DoD is treated as black/transparent.

Auto Crop Tab

Auto Crop tab analyzes the selected channel and crops the image based on that channel's boundaries. The adjustments from auto crop are seen in the Crop tab parameters.
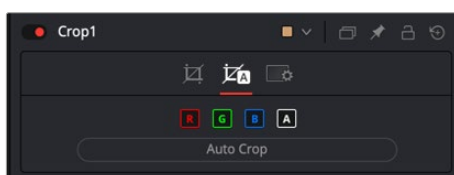
The Auto Crop tab

RGBA Color Channels

Select which channels are examined for an Auto Crop. This is useful for auto cropping images with non-solid backgrounds in a specific color channel, like a blue color gradient. Toggling the channel off causes Auto Crop to ignore it when evaluating the image.

Auto Crop

This evaluates the image and attempts to determine the background color. It then crops each side of the image to the first pixel that is not that color.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Transform nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

DVE [DVE]
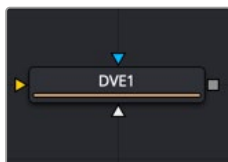
The DVE node

DVE Node Introduction

The DVE (Digital Video Effects) node is a 3D-image transformation similar to nodes found in old, tape-based online editing suites. The node encompasses image rotations, perspective changes, and Z moves. The axis can be defined for all transformations.

Inputs

The three inputs on the DVE node are used to connect a 2D image, DVE mask, and an effect mask, which can be used to limit the DVE area.

— **Input:** The orange input is used for the primary 2D image that is transformed by the DVE.

— **DVE Mask:** The white DVE mask input is used to mask the image prior to the DVE transform being applied. This has the effect of modifying both the image and the mask.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input causes the DVE to modify only the image within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

In the example below, the DVE node is inserted between the MediaIn2 node and the foreground input of the Merge. The MediaIn1 node is manipulated in the DVE node and composited over the top of the MediaIn1 node.
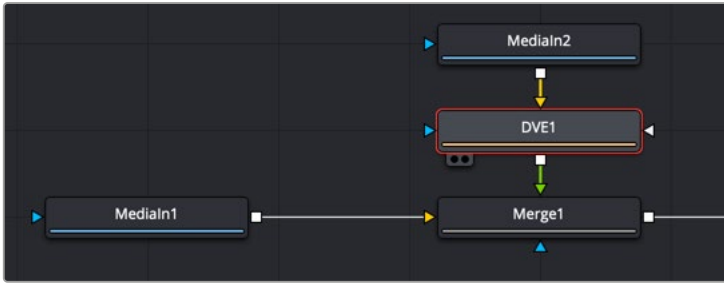
The DVE node modifying the foreground input of a Merge node

Inspector
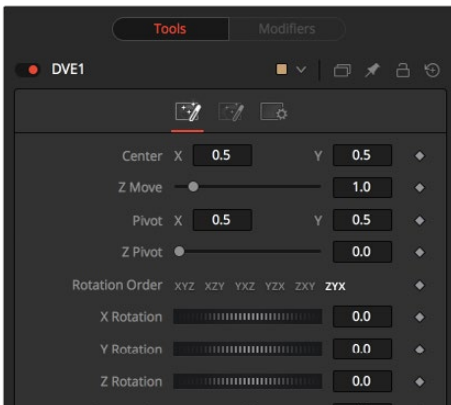
The DVE Controls tab

Controls Tab

The Controls tab includes all the transform parameters for the DVE.

Pivot X, Y, and Z

Positions the axis of rotation and scaling. The default is 0.5, 0.5 for X and Y, which is in the center of the image, and 0 for Z, which is at the center of Z space.

Rotation Order

Use these buttons to determine in what order rotations are applied to the image.

XYZ Rotation

These controls are used to rotate the image around the pivot along the X-, Y- and Z-axis.

Center X and Y

This positions the center of the DVE image onscreen. The default is 0.5, 0.5, which positions the DVE in the center of the image.

Z Move

This zooms the image in and out along the Z-axis. Visually, when this control is animated, the effect is similar to watching an object approach from a distance.

Perspective

This adds additional perspective to an image rotated along the X- or Y-axis, similar to changing the Field of View and zoom of a camera.

Masking Tab

The DVE node allows pre-masking of its input image. This offers the ability to create transformations from the masked area of the image while leaving the remainder of the image unaffected.
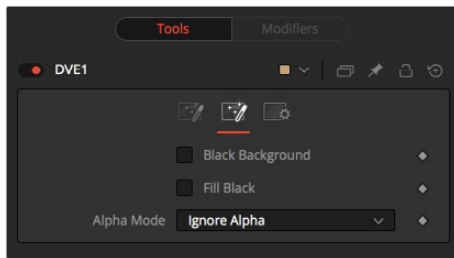
The DVE Masking tab

Unlike regular effect masks, the masking process occurs before the transformation. All the usual mask types can be applied to the DVE mask.

Black Background

Toggle this on to erase the area outside the mask from the transformed image.

Fill Black

Toggle this on to erase the area within the mask (before transformation) from the DVE's input, effectively cutting the masked area out of the image. Enabling both Black Background and Fill Black will show only the masked, transformed area.

Alpha Mode

This determines how the DVE will handle the alpha channel of the image when merging the transformed image areas over the untransformed image.

— **Ignore Alpha:** This causes the input image's alpha channel to be ignored, so all masked areas will be opaque.

— **Subtractive/Additive:** These cause the internal merge of the pre-masked DVE image over the input image to be either Subtractive or Additive.

   — An Additive setting is necessary when the foreground DVE image is premultiplied, meaning that the pixels in the color channels have been multiplied by the pixels in the alpha channel. The result is that transparent pixels are always black, since any number multiplied by 0 always equals 0. This obscures the background (by multiplying with the inverse of the foreground alpha), and then simply adds the pixels from the foreground.

   — A Subtractive setting is necessary if the foreground DVE image is not premultiplied. The compositing method is similar to an Additive merge, but the foreground DVE image is first multiplied by its own alpha, to eliminate any background pixels outside the alpha area.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Transform nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Letterbox [Lbx]

The Letterbox node

Letterbox Node Introduction

Use the Letterbox node to adapt existing images to the frame size and aspect ratios of any other format. The most common use of this node is to convert film resolution images to HD-sized frames for viewing on an external television monitor. Horizontal or vertical black edges are automatically added where necessary to compensate for aspect ratio differences. This node actually changes the resolution of the image.

> **NOTE:** Because this node changes the physical resolution of the image, animating the controls is not recommended.

Inputs

The single input on the Letterbox node is used to connect a 2D image for letterbox/cropping.

— **Input:** The orange input is used for the primary 2D image you want to letterbox/crop.

Basic Node Setup

The Letterbox node is used in the example below to change the resolution of the Merge node's output. Depending on how the resolution is modified, side pillars, or a horizontal letterbox mask, is applied to "fill in" the frame area, which the Merge node output does not cover.
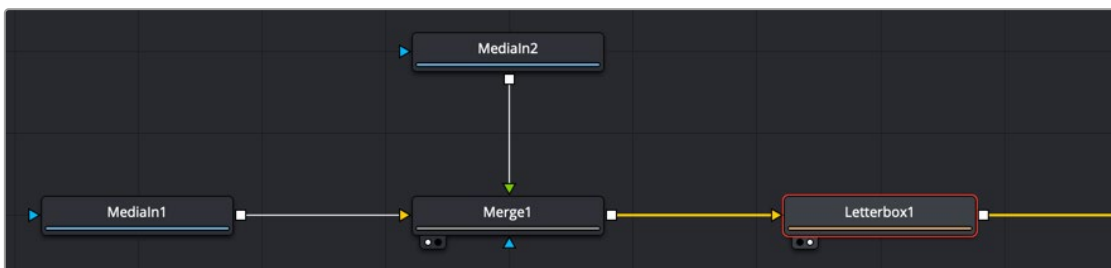
The Letterbox node converting the Merge output resolution and adding letterbox masking where needed.

Inspector

The Letterbox Controls tab

Controls Tab

The Controls tab includes parameters for adjusting the resolution and pixel aspect of the image. It also has the option of letterboxing or pan-and-scan formatting.

Width and Height

The values of these controls determine the size of the output image as measured in pixels.

> **TIP:** You can use the formatting contextual menu to quickly select a resolution from a list. Place the pointer over the Width or Height controls, and then right-click to display the contextual menu. The bottom of the menu displays a Select Frame Format submenu with available frame formats. Select any one of the choices from the menu to set the Height, Width, and Aspect controls automatically.

Auto Resolution

Activating this checkbox automatically sets the Width and Height sliders to the Frame Format settings found in the Preferences window for Fusion Studio or to the resolution of the DaVinci Resolve Timeline.

Pixel Aspect X and Y

These controls determine the pixel aspect ratio of the output image.

Center X and Y

This Center control repositions the image window when used in conjunction with Pan-and-Scan mode. It has no effect on the image when the node is set to Letterbox mode.

Mode

This control is used to determine the Letterbox node's mode of operation.

— **Letterbox/Envelope:** This corrects the aspect of the input image and resizes it to match the specified width.

— **Pan-and-Scan:** This corrects the aspect of the input image and resizes it to match the specified height. If the resized input image is wider than the specified width, the Center control can be used to animate the visible portion of the resized input.

Filter Method

When rescaling a pixel, surrounding pixels are often used to give a more realistic result. There are various algorithms for combining these pixels, called filters. More complex filters can give better results but are usually slower to calculate. The best filter for the job often depends on the amount of scaling and on the contents of the image itself.

— **Box:** This is a simple interpolation resize of the image.
— **Linear:** This uses a simplistic filter, which produces relatively clean and fast results.
— **Quadratic:** This filter produces a nominal result. It offers a good compromise between speed and quality.
— **Cubic:** This produces better results with continuous-tone images. If the images have fine detail in them, the results may be blurrier than desired.
— **Catmull-Rom:** This produces good results with continuous-tone images that are resized down. This produces sharp results with finely detailed images.
— **Gaussian:** This is very similar in speed and quality to Bi-Cubic.
— **Mitchell:** This is similar to Catmull-Rom but produces better results with finely detailed images. It is slower than Catmull-Rom.
— **Lanczos:** This is very similar to Mitchell and Catmull-Rom but is a little cleaner and also slower.
— **Sinc:** This is an advanced filter that produces very sharp, detailed results; however, it may produce visible "ringing" in some situations.
— **Bessel:** This is similar to the Sinc filter but may be slightly faster.

Window Method (Sinc and Bessel Only)

Some filters, such as Sinc and Bessel, require an infinite number of pixels to calculate exactly. To speed up this operation, a windowing function is used to approximate the filter and limit the number of pixels required. This control appears when a filter that requires windowing is selected.

— **Hanning:** This is a simple tapered window.
— **Hamming:** Hamming is a slightly tweaked version of Hanning that does not taper all the way down to zero.
— **Blackman:** A window with a more sharply tapered falloff.
— **Kaiser:** A more complex window with results between Hamming and Blackman.

Most of these filters are useful only when making an image larger. When shrinking images, it is common to use the Bi-Linear filter; however, the Catmull-Rom filter will apply some sharpening to the results and may be useful for preserving detail when scaling down an image.

EXAMPLE

Different resize filters. From left to right: Nearest Neighbor, Box, Linear, Quadratic, Cubic, Catmull-Rom, Gaussian, Mitchell, Lanczos, Sinc, and Bessel.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Transform nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Resize [Rsz]
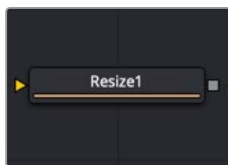
The Resize node

Resize Node Introduction

Use the Resize node to increase or decrease the resolution of an input image. This is useful for converting images from one format to another (for example, from film to video resolution). This node actually changes the resolution of the image.

> **NOTE:** Because this node changes the physical resolution of the image, animating the controls is not advised.

Inputs

The single input on the Resize node is used to connect a 2D image for resizing.

— **Input:** The orange input is used for the primary 2D image you want to resize.

Basic Node Setup

Below, the Resize node is inserted between the MediaIn1 node and the background input of the Merge. Unlike using a Transform tool, resizing the MediaIn1 changes the resolution of the clip. The resized MediaIn1 node connected to the orange background input also sets the resolution of the Merge output.
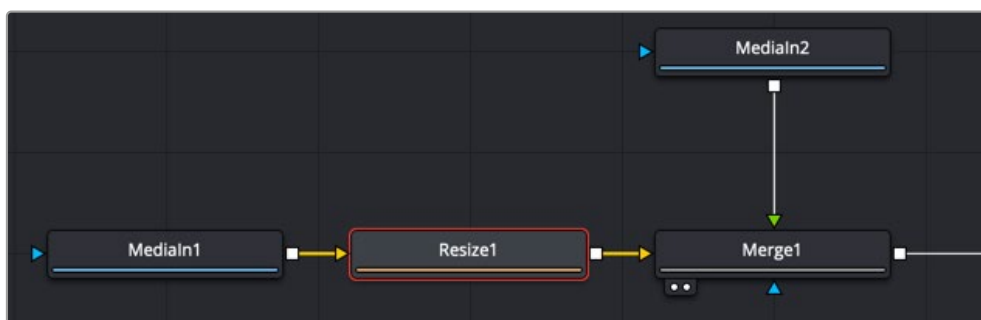
The Resize node can be used to scale an image and change its resolution.

Inspector
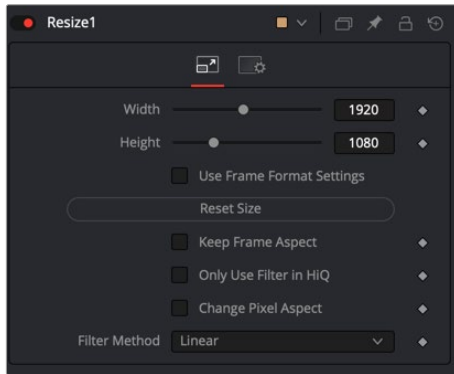
The Resize Controls tab

Controls Tab

The Controls tab includes parameters for changing the resolution of the image. It uses pixel values in the Width and Height controls.

Width

This controls the new resolution for the image along the X-axis.

Height

This controls the new resolution for the image along the Y-axis.

> **TIP:** You can use the formatting contextual menu to quickly select a resolution from a list. Place the mouse pointer over the Width or Height controls, and then right-click to display the contextual menu. The bottom of the menu displays a Select Frame Format submenu with available frame formats. Select any one of the choices from the menu to set the Height and Width controls automatically.
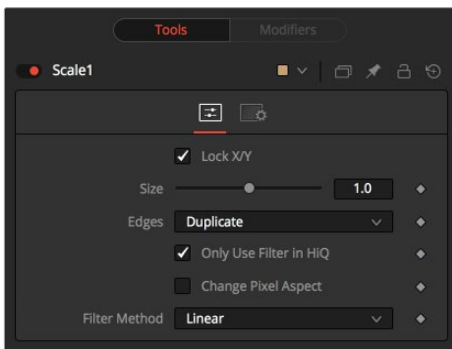
Auto Resolution

Activating this checkbox automatically sets the Width and Height sliders to the Frame Format settings found in the Preferences window for Fusion Studio or the resolution in the DaVinci Resolve Timeline.

Reset Size

Resets the image dimensions to the original size of the image.

Keep Frame Aspect

When toggled on, the Resize node maintains the aspect of the original image, preserving the original ratio between width and height.

Only Use Filter in HiQ

The Resize node will normally use the fast Nearest Neighbor filter for any non-HiQ renders, where speed is more important than full accuracy. Disable this checkbox to force Resize to always use the selected filter for all renders.

Change Pixel Aspect

Enable this checkbox to reveal a Pixel Aspect control that can be used to change the image's pixel aspect.

Filter Method

When rescaling a pixel, surrounding pixels are often used to give a more realistic result. There are various algorithms for combining these pixels, called filters. More complex filters can give better results but are usually slower to calculate. The best filter for the job often depends on the amount of scaling and on the contents of the image itself.

— **Box:** This is a simple interpolation resize of the image.
— **Linear:** This uses a simplistic filter, which produces relatively clean and fast results.
— **Quadratic:** This filter produces a nominal result. It offers a good compromise between speed and quality.
— **Cubic:** This produces better results with continuous-tone images. If the images have fine detail in them, the results may be blurrier than desired.
— **Catmull-Rom:** This produces good results with continuous-tone images that are resized down. This produces sharp results with finely detailed images.
— **Gaussian:** This is very similar in speed and quality to Bi-Cubic.
— **Mitchell:** This is similar to Catmull-Rom but produces better results with finely detailed images. It is slower than Catmull-Rom.
— **Lanczos:** This is very similar to Mitchell and Catmull-Rom but is a little cleaner and also slower.
— **Sinc:** This is an advanced filter that produces very sharp, detailed results; however, it may produce visible "ringing" in some situations.
— **Bessel:** This is similar to the Sinc filter but may be slightly faster.

Window Method (Sinc and Bessel Only)

Some filters, such as Sinc and Bessel, require an infinite number of pixels to calculate exactly. To speed up this operation, a windowing function is used to approximate the filter and limit the number of pixels required. This control appears when a filter that requires windowing is selected.

— **Hanning:** This is a simple tapered window.
— **Hamming:** Hamming is a slightly tweaked version of Hanning that does not taper all the way down to zero.
— **Blackman:** A window with a more sharply tapered falloff.
— **Kaiser:** A more complex window with results between Hamming and Blackman.

Most of these filters are useful only when making an image larger. When shrinking images, it is common to use the Bi-Linear filter; however, the Catmull-Rom filter will apply some sharpening to the results and may be useful for preserving detail when scaling down an image.

Different resize filters. From left to right: Nearest Neighbor, Box, Linear, Quadratic, Cubic, Catmull-Rom, Gaussian, Mitchell, Lanczos, Sinc, and Bessel.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Transform nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Scale [Scl]

The Scale node

Scale Node Introduction

The Scale node is almost identical to the Resize node, except that Resize uses exact dimensions, whereas the Scale node uses relative dimensions to describe the change to the source image's resolution. This node actually changes the resolution of the image.

> **NOTE:** Because this node changes the physical resolution of the image, animating the controls is not advised.

Inputs

The single input on the Scale node is used to connect a 2D image for scaling.

— **Input:** The orange input is used for the primary 2D image you want to scale.

Basic Node Setup

Below, the Scale node is inserted between the MediaIn1 node and the background input of the Merge. Unlike using a Transform tool, scaling the MediaIn1 changes the resolution of the clip. The resized MediaIn1 node connected to the orange background input also sets the resolution of the Merge output.
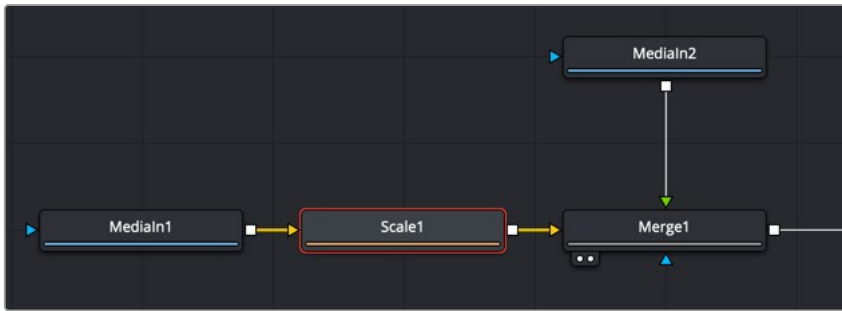
The Scale node can be used to scale an image and change its resolution.

Inspector

The Scale Controls tab

Controls Tab

The Controls tab includes parameters for changing the resolution of the image. It uses a multiplier of size to set the new resolution. An Edges menu allows you to determine how the edges of the frame are handled if the scaling decreases.

Lock X/Y

When selected, only a Size control is shown, and changes to the image's scale are applied to both axes equally. If the checkbox is cleared, individual Size controls appear for both X and Y Size.

Size

The Size control is used to set the scale used to adjust the resolution of the source image. A value of 1.0 would have no affect on the image, while 2.0 would scale the image to twice its current resolution. A value of 0.5 would halve the image's resolution.

Only Use Filter in HiQ

The Scale node will normally use the fast Nearest Neighbor filter for any non-HiQ renders, where speed is more important than full accuracy. Disable this checkbox to force Scale to always use the selected filter for all renders.

Change Pixel Aspect

Enable this checkbox to reveal a Pixel Aspect control that can be used to change the image's pixel aspect.

Filter Method

When rescaling a pixel, surrounding pixels are often used to give a more realistic result. There are various algorithms for combining these pixels, called filters. More complex filters can give better results but are usually slower to calculate. The best filter for the job often depends on the amount of scaling and on the contents of the image itself.

— **Box:** This is a simple interpolation resize of the image.
— **Linear:** This uses a simplistic filter, which produces relatively clean and fast results.
— **Quadratic:** This filter produces a nominal result. It offers a good compromise between speed and quality.
— **Cubic:** This produces better results with continuous-tone images. If the images have fine detail in them, the results may be blurrier than desired.
— **Catmull-Rom:** This produces good results with continuous-tone images that are resized down. This produces sharp results with finely detailed images.
— **Gaussian:** This is very similar in speed and quality to Bi-Cubic.
— **Mitchell:** This is similar to Catmull-Rom but produces better results with finely detailed images. It is slower than Catmull-Rom.
— **Lanczos:** This is very similar to Mitchell and Catmull-Rom but is a little cleaner and also slower.
— **Sinc:** This is an advanced filter that produces very sharp, detailed results; however, it may produce visible "ringing" in some situations.
— **Bessel:** This is similar to the Sinc filter but may be slightly faster.

Window Method (Sinc and Bessel Only)

Some filters, such as Sinc and Bessel, require an infinite number of pixels to calculate exactly. To speed up this operation, a windowing function is used to approximate the filter and limit the number of pixels required. This control appears when a filter that requires windowing is selected.

— **Hanning:** This is a simple tapered window.
— **Hamming:** Hamming is a slightly tweaked version of Hanning that does not taper all the way down to zero.
— **Blackman:** A window with a more sharply tapered falloff.
— **Kaiser:** A more complex window with results between Hamming and Blackman.

Most of these filters are useful only when making an image larger. When shrinking images, it is common to use the Bi-Linear filter; however, the Catmull-Rom filter will apply some sharpening to the results and may be useful for preserving detail when scaling down an image.

> **NOTE:** Because this node changes the physical resolution of the image, animating the controls is not advised.
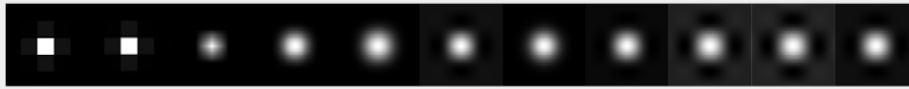
Different resize filters. From left to right: Nearest Neighbor, Box, Linear, Quadratic, Cubic, Catmull-Rom, Gaussian, Mitchell, Lanczos, Sinc, and Bessel.

Transform [XF]
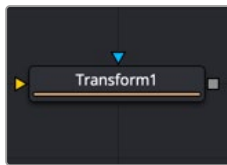
The Transform node

Transform Node Introduction

The Transform node can be used for simple 2D transformations of the image, such as moving, rotating, and scaling. The image's aspect can also be modified using the Transform node.

The Transform node concatenates its result with adjacent Transformation nodes. The Transform node does not change the image's resolution.

Inputs

The two inputs on the Transform node are used to connect a 2D image and an effect mask, which can be used to limit the transformed area.

— **Input:** The orange input is used for the primary 2D image that gets transformed.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the transform area to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Transform node in the example below is inserted between the MediaIn2 node and the foreground input of the Merge. Unlike using a Scale or Resize tool, transforming the MediaIn2 does not change the resolution of the clip. For that reason, it is the tool most often used to scale, move, and rotate a clip.
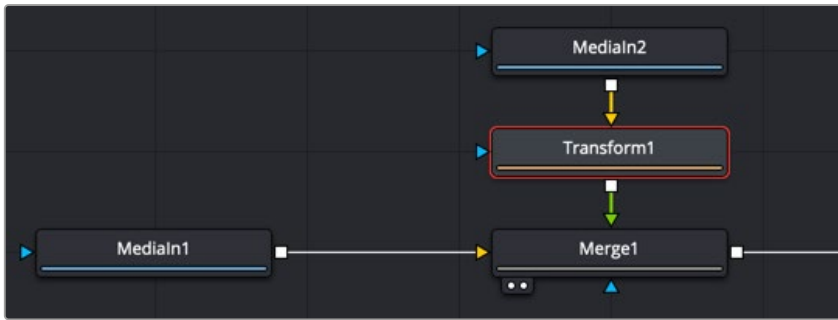
The Transform node can be used to scale an image without changing its resolution.

Inspector
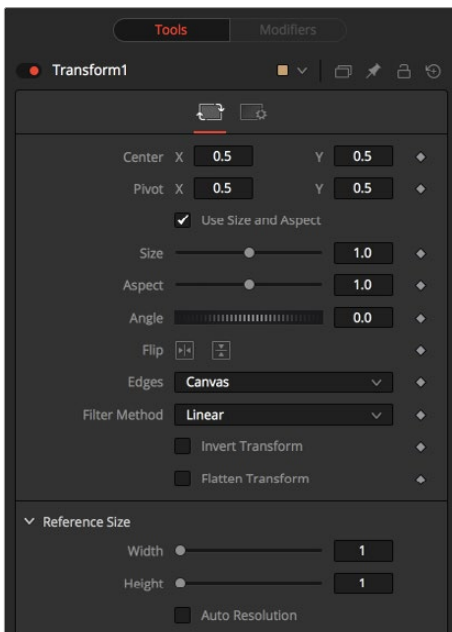
The Transform Controls tab

Controls Tab

The Controls tab presents multiple ways to transform, flip (vertical), flop (horizontal), scale, and rotate an image. It also includes reference size controls that can reinterpret the coordinates used for width and height from relative values of 0-1 into pixel values based on the image's resolution.

Center X and Y

This sets the position of the image on the screen. The default is 0.5, 0.5, which places the image in the center of the screen. The value shown is always the actual position multiplied by the reference size. See below for a description of the reference size.

Pivot X and Y

This positions the axis of rotation and scaling. The default is 0.5, 0.5, which is the center of the image.

Use Size and Aspect

This checkbox determines whether the Transform node provides independent Size controls for the X and Y scale or if Size and Aspect controls are used instead.

Size

This modifies the scale of the image. Values range from 0 to 5, but any value greater than zero can be entered into the edit box. If the Use Size and Aspect checkbox is selected, this control will scale the image equally along both axes. If the Use Size and Aspect option is off, independent control is provided for X and Y.

Aspect

This control changes the aspect ratio of an image. Setting the value above 1.0 stretches the image along the X-axis. Values between 0.0 and 1.0 stretch the image along the Y-axis. This control is available only when the Use Size and Aspect checkbox is enabled.

Angle

This control rotates the image around the axis. Increasing the Angle rotates the image in a counterclockwise direction. Decreasing the Angle rotates the image in a clockwise direction.

Flip Horizontally and Vertically

Toggle this control on to flip the image along the X- or Y-axis.

Edges

This menu determines how the edges of the image are treated when the edge of the raster is exposed.

— **Canvas:** This causes the edges of the image that are revealed to show the current Canvas Color. This defaults to black with no Alpha and can be set using the Set Canvas Color node.

— **Wrap:** This wraps the edges of the image around the borders of the image. This is useful for seamless images to be panned, creating an endless moving background image.

— **Duplicate:** This causes the edges of the image to be duplicated as best as possible, continuing the image beyond its original size.

— **Mirror:** Image pixels are mirrored to fill to the edge of the frame.

Filter Method

When rescaling a pixel, surrounding pixels are often used to give a more realistic result. There are various algorithms for combining these pixels, called filters. More complex filters can give better results but are usually slower to calculate. The best filter for the job often depends on the amount of scaling and on the contents of the image itself.

— **Box:** This is a simple interpolation resize of the image.

— **Linear:** This uses a simplistic filter, which produces relatively clean and fast results.

— **Quadratic:** This filter produces a nominal result. It offers a good compromise between speed and quality.

— **Cubic:** This produces better results with continuous-tone images. If the images have fine detail in them, the results may be blurrier than desired.

— **Catmull-Rom:** This produces good results with continuous-tone images that are resized down. This produces sharp results with finely detailed images.

— **Gaussian:** This is very similar in speed and quality to Bi-Cubic.

— **Mitchell:** This is similar to Catmull-Rom but produces better results with finely detailed images. It is slower than Catmull-Rom.

— **Lanczos:** This is very similar to Mitchell and Catmull-Rom but is a little cleaner and also slower.

— **Sinc:** This is an advanced filter that produces very sharp, detailed results; however, it may produce visible "ringing" in some situations.

— **Bessel:** This is similar to the Sinc filter but may be slightly faster.

Window Method (Sinc and Bessel Only)

Some filters, such as Sinc and Bessel, require an infinite number of pixels to calculate exactly. To speed up this operation, a windowing function is used to approximate the filter and limit the number of pixels required. This control appears when a filter that requires windowing is selected.

— **Hanning:** This is a simple tapered window.

— **Hamming:** Hamming is a slightly tweaked version of Hanning that does not taper all the way down to zero.

— **Blackman:** A window with a more sharply tapered falloff.

— **Kaiser:** A more complex window with results between Hamming and Blackman.

Most of these filters are useful only when making an image larger. When shrinking images, it is common to use the Bi-Linear filter; however, the Catmull-Rom filter will apply some sharpening to the results and may be useful for preserving detail when scaling down an image.

EXAMPLE

Different resize filters. From left to right: Nearest Neighbor, Box, Linear, Quadratic, Cubic, Catmull-Rom, Gaussian, Mitchell, Lanczos, Sinc, and Bessel.

Invert Transform

Select this control to invert any position, rotation, or scaling transformation. This option is useful when connecting the Transform to the position of a tracker for the purpose of reintroducing motion back into a stabilized image.

Flatten Transform

The Flatten Transform option prevents this node from concatenating its transformation with adjacent nodes. The node may still concatenate transforms from its input, but it will not concatenate its transformation with the node at its output.

Reference Size

The controls under the Reference Size menu do not directly affect the image. Instead they allow you to control how Fusion represents the position of the Transform node's center.

Normally, coordinates are represented as values between 0 and 1, where 1 is a distance equal to the full width or height of the image. This allows for resolution independence, because you can change the size of the image without having to change the value of the center.

One disadvantage to this approach is that it complicates making pixel-accurate adjustments to an image. To demonstrate, imagine an image that is 100 x 100 pixels in size. To move the center of the image to the right by 5 pixels, we would change the X value of the transform center from 0.5, 0.5 to 0.55, 0.5. We know the change must be 0.05 because 5/100 = 0.05.

The Reference Size controls allow you to specify the dimensions of the image. This changes the way the control values are displayed, so that the Center shows the actual pixel positions in the X and Y number fields of the Center control. Extending our example, if you set the Width and Height to 100 each, the Center would now be shown as 50, 50, and we would move it 5 pixels toward the right by entering 55, 50.

Internally, the Transform node still stores this value as a number between 0 to 1, and if you were to query the Center controls value via scripting, or publish the Center control for use by other nodes, then you would retrieve the original normalized value. The change is visible only in the value shown for Transform Center in the node control.

Reference Width and Height Sliders

Set these to the width and height of the image to change the way that Fusion displays the values of the Transform node's Center control.

Auto Resolution

Enable this checkbox to use the current frame format settings in Fusion Studio or the timeline resolution in DaVinci Resolve to set the Reference Width and Reference Height values.

The Common Controls

Nodes that handle Transform operations share several identical controls in the Inspector. This section describes controls that are common among Transform nodes.

Inspector
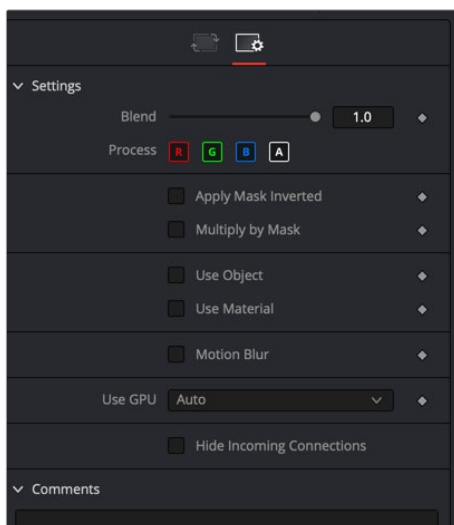
The Transform Common controls

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Transform category. The Settings controls are even found on third-party Transform-type plugin tools. The controls are consistent and work the same way for each tool.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this will cause the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if the node is scripted to trigger a task, but the node's value is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the Red button on a Blur tool is deselected, the blur will first be applied to the image, and then the red channel from the original input will be copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this will generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option will cause the RGB values of the masked image to be multiplied by the mask channel's values. This will cause all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object ID and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels will be used, if present. The specific Material ID or Object ID affected is chosen using the next set of controls.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option is disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID will be used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the viewer. The image or sequence must have been rendered from a 3D software package with those channels included.

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 will cause Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:** Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one full frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows for the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off GPU hardware-accelerated rendering. Enabled uses the GPU hardware for rendering the node. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node will be displayed in the Inspector. Dragging a connected node from the node tree into the field will hide that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line will reappear.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

USD Nodes

This chapter details the Universal Scene Descriptor (USD) nodes available in Fusion.

> The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.
>
> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

uCamera [uCa]
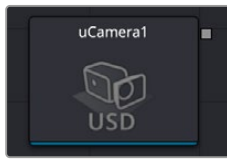
The USD Camera Node

uCamera Node Introduction

The uCamera node generates a virtual camera for viewing the USD scene in its 3D environment. It closely emulates the settings used in real cameras to make matching live-action or 3D-rendered elements as seamless as possible. Adding any cameras to a 3D composite allows you to frame the elements in a composite how you want and animate the camera during a scene to create moving camera shots.

Inputs

— None.

Basic Node Setup

The output of a uCamera node should be connected to a uMerge node to build it into a USD 3D scene with other assets and nodes.
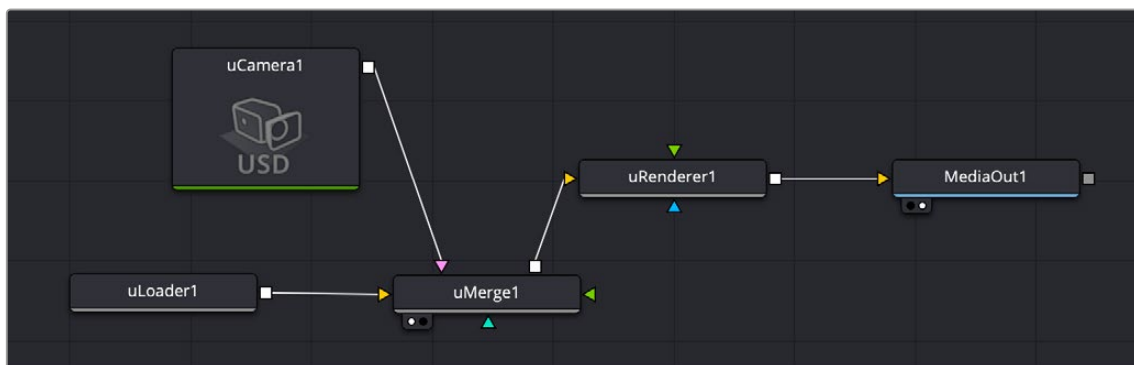
The uCamera node is connected to a uMerge node to give standard camera controls to a USD scene.

Displaying a uCamera node directly in the viewer shows only an empty scene; there is nothing for the camera to see. To view the scene through the camera, view the uMerge node where the uCamera is connected, or any node downstream of that uMerge. Then right-click on the viewer and select uCamera > [Camera name] from the contextual menu. Right-clicking on the axis label found in the lower corner of each USD viewer also displays the Camera submenu.

The aspect of the viewer may be different from the aspect of the camera, so the camera view may not match the actual boundaries of the image rendered by the uRenderer node. Guides can be enabled to represent the portion of the view that the camera sees and assist you in framing the shot. Right-click on the viewer and select an option from the Guides > Frame Aspect submenu. The default option uses

the format enabled in the Composition > Frame Format preferences. To toggle the guides on or off, select Guides > Show Guides from the viewers' contextual menu, or use the Command-G (macOS) or Ctrl-G (Windows) keyboard shortcut when the viewer is active.

Inspector
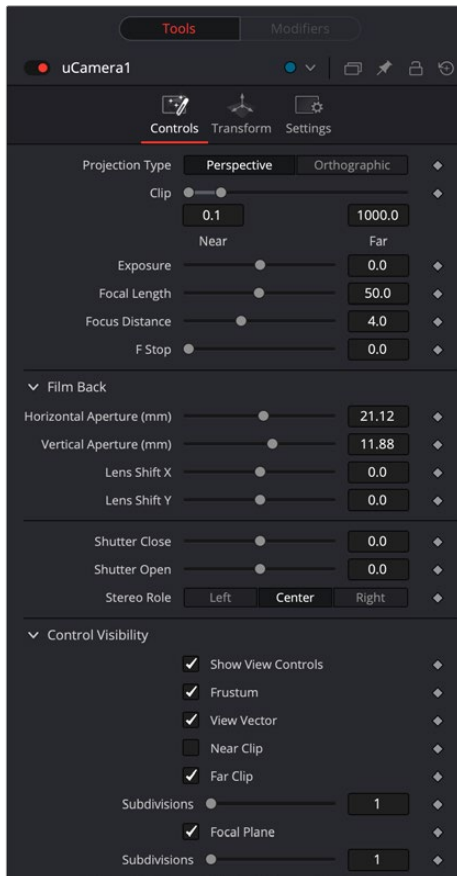
The uCamera Controls tab

Controls Tab

The Controls tab contains some of the most fundamental camera settings, including the camera's clipping plains, focal length, and film back properties.

Projection Type

The Projection Type menu is used to select between Perspective and Orthographic cameras. Generally, real-world cameras are perspective cameras. An orthographic camera uses parallel orthographic projection, a technique where the view plane is perpendicular to the viewing direction. This produces a parallel camera output that is undistorted by perspective.

Orthographic cameras present controls only for the near and far clipping planes and a control to set the viewing scale.

Near/Far Clip

The clipping planes are used to limit what geometry in a scene is rendered based on an object's distance from the camera's focal point. Clipping planes ensure objects that are extremely close to the camera, as well as objects that are too far away to be useful, are excluded from the final rendering.

The default perspective camera ignores this setting unless the Adaptive Near/Far Clip checkbox located under the Near/Far Clip control is disabled.

The clip values use units, so a far clipping plane of 20 means that any object more than 20 units from the camera is invisible to the camera. A near clipping plane of 0.1 means that any object closer than 0.1 units is also invisible.

> **NOTE:** A smaller range between the near and far clipping planes allows greater accuracy in all depth calculations. If a scene begins to render strange artifacts on distant objects, try increasing the distance for the Near Clip plane.

Exposure

When selected, the renderer automatically adjusts the camera's Near/Far Clipping plane to match the extents of the scene. This setting overrides the values of the Near and Far clip range controls described above. This option is not available for orthographic cameras.

Focal Length

In the real world, a lens' focal length is the distance from the center of the lens to the film plane. The shorter the focal length, the closer the focal plane is to the back of the lens. The focal length is measured in millimeters. The angle of view and focal length controls are directly related. Smaller focal lengths produce a wider angle of view, so changing one control automatically changes the other to match.

The relationship between focal length and angle of view is angle = 2 * arctan[aperture / 2 / focal_length]. Use the vertical aperture size to get the vertical angle of view and the horizontal aperture size to get the horizontal angle of view.

Focal Distance

Like a focal point on a real-world camera, this setting defines the distance from the camera to an object and is used to calculate depth of field.

F Stop

This is used to define the aperture size of the synthetic lens; it will affect exposure and is used to calculate depth of field.

Film Back

This section allows you to control the technical parameters of the non-lens part of the camera.

Horizontal/Vertical Aperture

The Horizontal Aperture Width and Vertical Aperture Height sliders control the dimensions of the camera's aperture or the portion of the camera that lets light in on a real-world camera. In video and film cameras, the aperture is the size of the opening that defines the area of each frame exposed, also known as sensor size.

Lens Shift X/Y

Provides controls to manually offset the camera lens width and height.

Shutter Close / Open

These define when the shutter of the camera will open and close; this will affect Exposure and Motion Blur.

Stereo Role

If the camera is stereo, this defines the center of role for the two cameras, left, center, or right.

Control Visibility

This section allows you to selectively activate the onscreen controls that are displayed along with the camera.

— **Show View Controls:** Displays or hides all camera onscreen controls in the viewers.
— **Frustum:** Displays the actual viewing cone of the camera.
— **View Vector:** Displays a white line inside the viewing cone, which can be used to determine the center of the frame.
— **Near Clip:** The Near Clipping plane. This plane can be subdivided for better visibility.
— **Far Clip:** The Far Clipping plane. This plane can be subdivided for better visibility.
— **Focal Plane:** The plane set by the Focus Distance. This plane can be subdivided for better visibility.
— **Convergence Distance:** The point of convergence when using Stereo mode. This plane can be subdivided for better visibility.

Common Controls

Transform and Settings Tab

The Transform and Settings tab in the Inspector are also duplicated in other USD nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

uImage Plane [uIm]
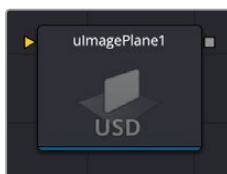
The USD Image Plane node

uImage Plane Node Introduction

The uImage Plane node produces 2D planar geometry in 3D space. The node is most commonly used to represent 2D images on "cards" in the 3D space. The aspect of the image plane is determined by the aspect of the image that is input.

Inputs

The input on this node is for the image that will be texture mapped to the plane.

— **Input:** The yellow input is used for the connection of a MediaIn image.

Basic Node Setup
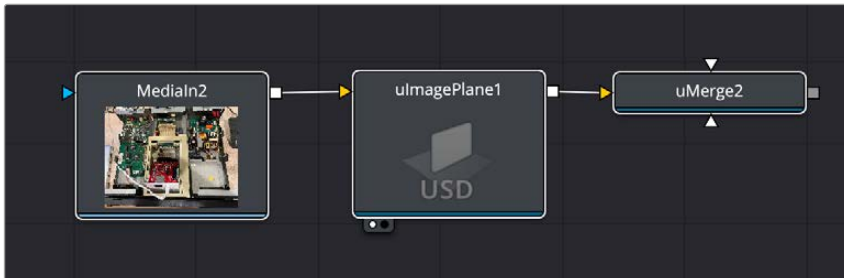
The uImage Plane Node accepts a 2D image and places it into a 3D USD scene.

Inspector
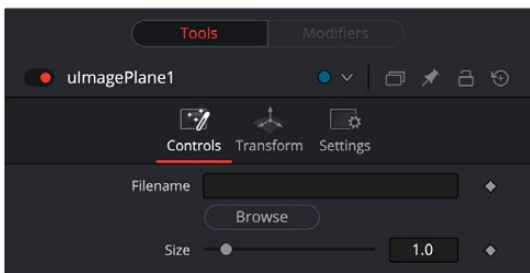
The uImage Plane controls

Controls Tab

Most of the Controls tab is taken up by common controls. The Image Plane-specific controls at the top of the Inspector allow minor adjustments.

File Name

The uImagePlane has an input so an image can be piped directly into the node. This node can also directly load an image by browsing, and it will show the name and path in the filename. Use the Browse button to open the file browser and select an image.

Size

Size will set the size of the image plane in the USD 3D scene.

Common Controls

Transform and Settings Tab

The Transform and Settings tab in the Inspector are also duplicated in other USD nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

uLoader [uLd]
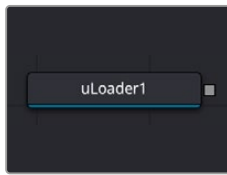
The USD Loader node

uLoader Node Introduction

The uLoader node is used for importing USD files into your Fusion Composition. It will load usd, usda, usdc, and usdz formats.

Inputs

— None.

Basic Node Setup

The uLoader node imports a USD scene and then merges it with a USD Camera for view control.

Inspector
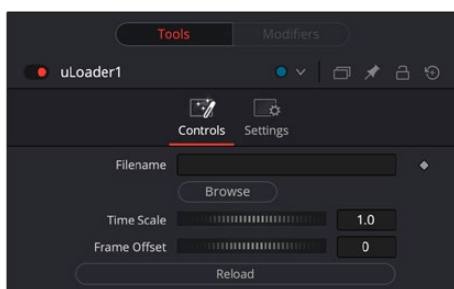
The USD Loader controls

Controls Tab

Filename

This control allows you to source a USD file to bring into the Fusion project. Use the Browse button to open the file browser and select a USD file (.usd, .usda, .usdc, or .usdz).

Time Scale

Animated .usd scenes can have their animation speed adjusted to be faster and slower using time scale.

Frame Offset

Will define when an animation will start, so timing can be moved along the timeline.

Reload

This button will reload the USD file set in the Filename field above. This lets you make changes to the USD file in another application and see the results in Fusion by reloading it.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other USD nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

uMerge [uMg]

The USD Merge node

uMerge Node Introduction

The uMerge node is the primary node in Fusion that you use to combine separate 3D elements into the same 3D environment.

For example, in a scene created with an image plane, a camera, and a light, the camera would not be able to see the image plane and the light would not affect the image plane until all three objects are introduced into the same environment using the uMerge node.

The uMerge provides the standard transformation controls found on most nodes in Fusion's USD suite. Unlike those nodes, changes made to the translation, rotation, or scale of the uMerge affect all the objects connected to the uMerge. This behavior forms the basis for all parenting in Fusion's 3D environment.

Inputs

The uMerge node displays three inputs initially, but as each input is connected a new input appears on the node, assuring there is always one free to add a new element into the scene.

— **SceneInput[#]:** These inputs are used to connect USD image planes, 3D cameras, lights, entire USD scenes, as well as other uMerge nodes. There is no limit to the number of inputs this node can accept. The node dynamically adds more inputs as needed, ensuring that there is always at least one input available for connection.

Basic Node Setup
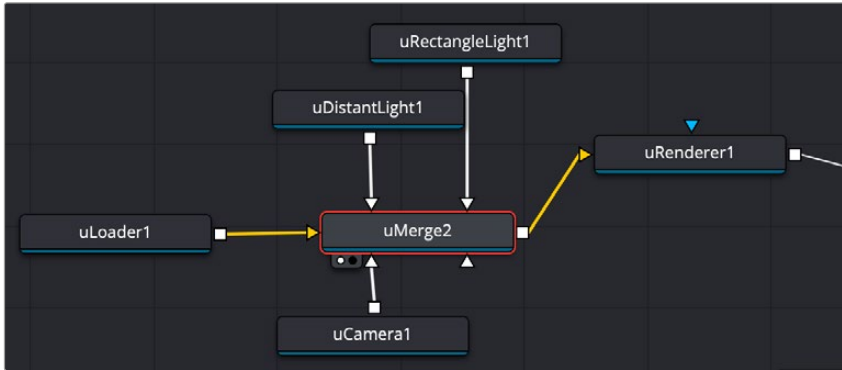
The uMerge node combines the 3D USD scene from the uLoader, two light sources, and the camera, then passes that view out to the uRenderer node to flatten into a 2D image.

Inspector

Common Controls

Transform and Settings Tab

The Transform and Settings tab in the Inspector is also duplicated in other USD nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

uRenderer [uRn]

The uRenderer node

uRenderer Node Introduction

The uRenderer node converts the USD 3D scene into a 2D image, using either a default perspective camera or one of the cameras found in the scene. Every USD scene in a composition terminates with at least one uRenderer node.

Inputs

The uRenderer node has two inputs. The main scene input takes in the uMerge or other USD nodes that need to be converted to 2D. The effect mask limits the uRenderer output.

— **SceneInput:** The orange scene input is a required input that accepts a USD scene that you want to convert to 2D.
— **EffectMask:** The blue effects mask input uses a 2D image to mask the output of the node.

Basic Node Setup

All USD scenes must end with a uRenderer node. The uRenderer node is used to convert a 3D scene into a 2D image. Below, the uRenderer node takes the output of a uMerge node and renders the 3D scene into a 2D image.
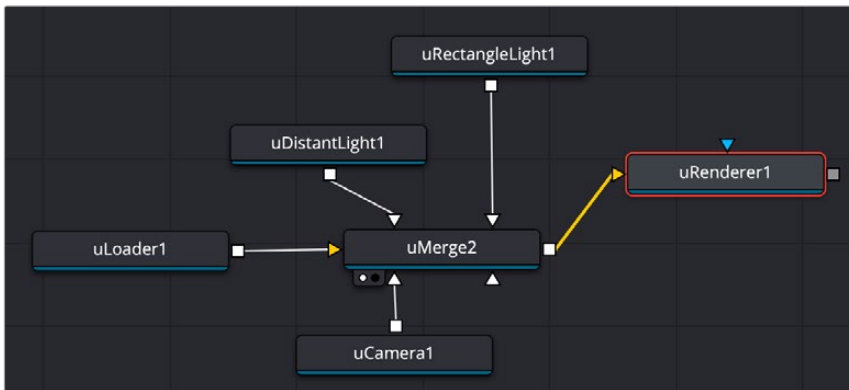
The uRenderer node takes the input from the uMerge node and flattens it into a 2D image for display.

Inspector
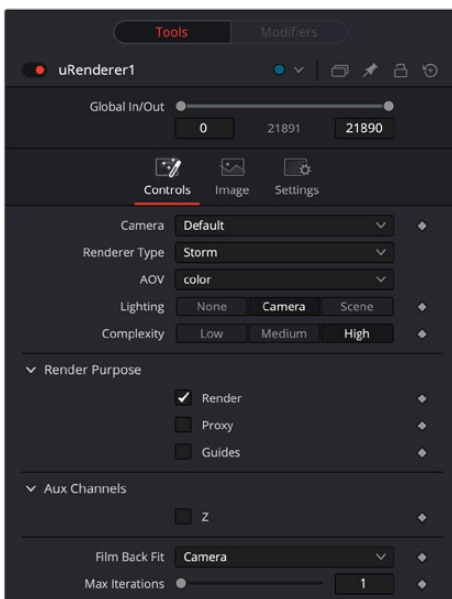
USD Renderer controls

Controls Tab

Camera

The Camera menu is used to select which camera from the scene is used when rendering. The Default setting uses the first camera found in the scene. If no camera is located, the default perspective view is used instead.

Renderer Type

This menu lists the available render engines. Storm is currently the only renderer available.

AOV

AOV or Arbitrary Output Variable, are additional passes that the render can output; these are also known as Deep Channels or Auxiliary channels. Color is one; others include Depth and Camera Depth.

— **Color:** This outputs the main full-color render of the USD scene. This will include textures and lighting

— **Depth:** This outputs a black-and-white depth map. This particular depth map is normalized for every frame, showing dynamic black to white values.

— **PrimID:** This outputs a numeric value for each different prim in the USD scene.

— **Camera Depth:** This outputs an accurate camera depth map, which calculates the camera's position relative to the USD scene. The image is rendered as a float 32bit image; at first glance this will appear black.

Lighting

Controls how the USD scene is to be lit.

— **None:** Disables all lighting.

— **Camera:** Enabled objects are lit by the default camera light

— **Scene:** Enabled objects are lit by any lights in the USD scene.

— **Enable Sky Dome:** When checked, the sky dome texture is displayed and rendered from the uRenderer.

Complexity

This defines how much detail is rendered: Low, Medium, or High detail.

Aux Channel Z

This option enables rendering of the Z channel, the depth channel that represents the distance of each pixel from the camera. When enabled, the camera depth map is rendered to the Z channel.

Film Back Fit

Defines how the rendered image fits the camera's Film Back. If the resolution of the images being rendered is not the same relative dimensions as the film back then these options set how to fit the image.

Max Iterations

This defines the amount of iterations of the rendering algorithm that will be processed.
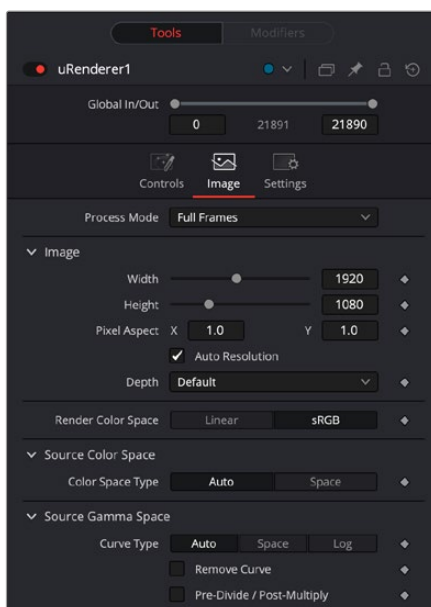
USD Renderer Image controls

Image Tab

Process Mode: Lets you choose whether the output of the node will be processed as Full Frames or via one of the specified interlaced methods.

— **Image:** Lets you control the output image size and aspect ratio.
  — **Width/Height:** Lets you set the pixel dimensions for the output of the node.
  — **Pixel Aspect XY:** Lets you set a custom pixel aspect ratio for non-square video formats.
  — **Auto Resolution:** Sets the resolution to the Timeline resolution. Uncheck to manually set the width/height above.
  — **Depth:** Lets you manually set the bit depth of the output from the uRenderer.

Render Color Space

Allows you to render the output to Linear or sRGB color spaces.

Source Color Space

The Color Space Type menu sets the color space of the media to help achieve a linear workflow. Unlike the Gamut tool, this doesn't perform any actual color space conversion but rather adds the source space data into the metadata, if that metadata doesn't already exist. The metadata can then be used downstream by a Gamut tool with the From Image option, or in a Saver if explicit output spaces are defined there.

— **Auto:** Passes along any metadata that might be in the incoming image.
— **Space:** Allows the user to set the color space from a variety of options.

Source Gamma Space

The Curve Type menu automatically determines or allows you to choose the Gamma setting for the image and allows the option to remove the Gamma curve to help achieve a linear workflow.

— **Auto:** Passes along any metadata that might be in the incoming image.
— **Space:** Allows you to choose a specific setting from a Gamma Space drop-down menu, while a visual graph lets you see a representation of the gamma setting you've selected.
— **Log:** Similar to the Log-Lin node, this option reveals specific log-encoded gamma profiles so that you can select the one that matches your content. A visual graph shows a representation of the log setting you've selected. When Cineon is selected from the Log Type menu, additional Lock RGB, Level, Soft Clip, Film Stock Gamma, Conversion Gamma, and Conversion table options are presented to finesse the gamma output.
— **Remove Curve:** Depending on the selected gamma space or on the gamma space found in Auto mode, the associated gamma curve is removed from the material, effectively converting it to output in a linear color space.
— **Pre-Divide/Post-Mujltiply:** Lets you convert "straight" Alpha channels into pre-multiplied Alpha channels, when necessary.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other USD nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

uShape [uSh]
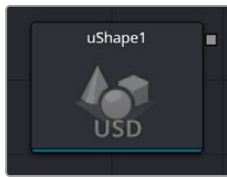
The USD Shape node

uShape Node Introduction

The uShape node is used to produce several basic primitive 3D shapes, including capsule, torus, planes, cubes, spheres, and cylinders.

Inputs

— None.

Basic Node Setup
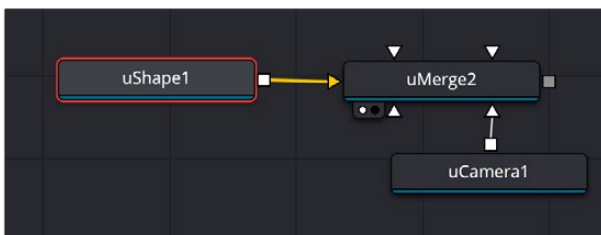
The uShape node generates a 3D shape that is merged with a uCamera node to adjust the view.

Inspector
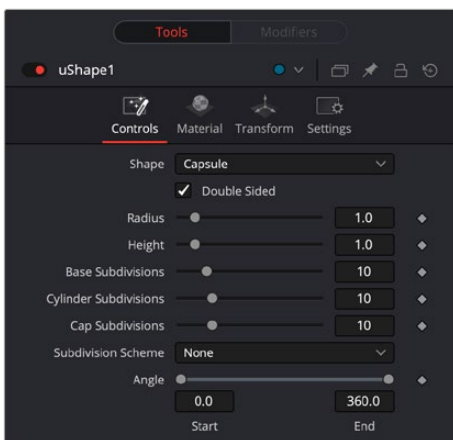
The USD Shape Controls

Controls Tab

Shape

This menu allows you to select the primitive geometry produced by the Shape 3D node. The remaining controls in the Inspector change to match the selected shape.

The Shape primitives are Capsule, Cone, Cube, Cylinder, Ico sphere, Plane, Sphere and Torus.

Double Sided

This will make the polygons have two faces and receive lighting on both sides.

Lock Width/Height/Depth

Only for Plane and Cube shapes. If this checkbox is selected, the width, height, and depth controls are locked together as a single size slider. Otherwise, individual controls over the size of the shape along each axis are provided.

Size Width/Height/Depth

Only for Plane and Cube shapes. Used to control the size of the shape.

Radius

When a Capsule, Sphere, Cylinder, Cone, or Torus is selected in the shape menu, this control sets the radius of the selected shape.

Height

When a Capsule, Cone, or Cylinder is selected in the shape menu, this control sets the height of the selected shape.

Top Radius

When a cone is selected in the Shape menu, this control is used to define a radius for the top of a cone, making it possible to create truncated cones.

Subdivision Level/Base/Height/Cap/Cylinder

The Subdivision controls are used to determine the tessellation of the mesh on all shapes. The higher the subdivision, the more vertices and polygons in each shape.

Subdivision Scheme

The Subdivision Scheme defines which algorithm is used to tessellate the polygons in the shapes. The methods are None, Bilinear, Loop and Catmull-Clark.

Angle

When the Capsule, Cone, Cylinder, Sphere, or Torus shape is selected in the Shape menu, this range control determines how much of the shape is drawn. A start angle of 180° and end angle of 360° would only draw half of the shape.

Latitude

When a Sphere or Torus is selected in the Shape menu, this range control is used to crop or slice the object by defining a latitudinal subsection of the object.

Cap Bottom/Top

When Cylinder or Cone is selected in the Shape menu, the Bottom Cap and Top Cap checkboxes are used to determine if the end caps of these shapes are created or if the shape is left open

Section

When the Torus is selected in the Shape menu, Section controls the thickness of the tube making up the torus.

Material Tab
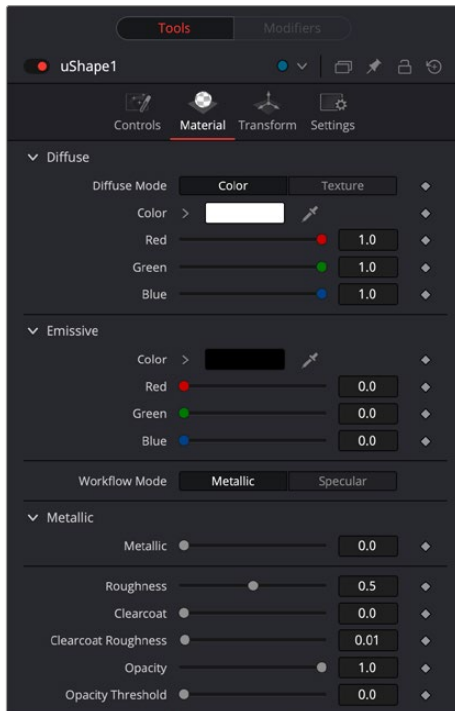
The USD Shape Material controls

Diffuse Mode

Diffuse describes the base surface characteristics without any additional effects like reflections or specular highlights.

— **Color:** Provides controls to modify the shape's diffuse color.
— **Texture:** This control allows you to use a source image as the diffuse surface color. Use the Browse button to open the file browser and select an image file.

Emissive

Emissive adds a global lighting effect to a shape, creating a layer of color over the existing texture. It can be adjusted to different levels of intensity and can be used to simulate the emission of light.

Workflow Mode

Lets you set the mode between Metallic and Specular.

— **Metallic:** When enabled, this control shows the Metallic slider. The Metallic slider adds a reflective quality to objects, creating the appearance of metal. It enhances the reflective properties of a shape.
— **Specular:** When enabled, this shows the Specular Color controls. Specular Color determines the color of light that reflects from a shiny surface. The more specular a material is, the glossier it appears.

Roughness

Affects the amount of light reflection and scattering on a surface, giving it a smoother or more textured appearance depending on the setting. It is a key element in creating realistic reflections.

Clearcoat

Affects the appearance of a surface by adding a glossy layer on top of it, mimicking the effect of a protective coating. It is commonly used in creating materials like car paint or polished metal.

Clearcoat Roughness

Adjusts the level of imperfection or smoothness of the glossy layer added on top of a surface. This can affect the realism and reflection of materials like car paint or polished metal.

Opacity

Reduces the material and object opacity, impacting the color and Alpha values of the shape.

Common Controls

Transform and Settings Tab

The Transform and Settings tab in the Inspector is also duplicated in other USD nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

uTransform [uXf]
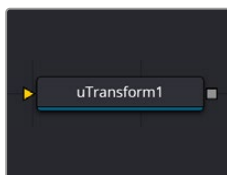
The USD Transform node

uTransform Node Introduction

The uTransform node is the same as a Null node in 3D; it will transform objects in a scene. Translation, rotation, and scale are the basic transforms supported, as well as Pivot location and target location for rotation.

Inputs

The uTransform node has a single required input for a USD scene or USD object.

— **Scene Input:** The orange scene input is connected to a USD scene or USD object to apply a second set of transformation controls.

Basic Node Setup

The uTransform node adds 3D position, rotation, and pivot control onto any existing transforms in the USD node prior to it. You can combine multiple uTransform nodes together to build parenting or hierarchical movement.
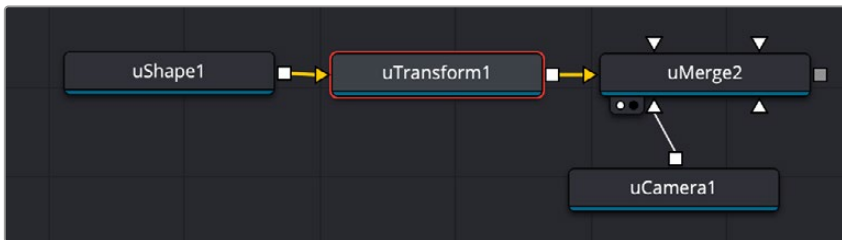
The uTransform node gives you another set of 3D transform controls that can be used to modify those in the Transform tab on the uShape node.

Inspector
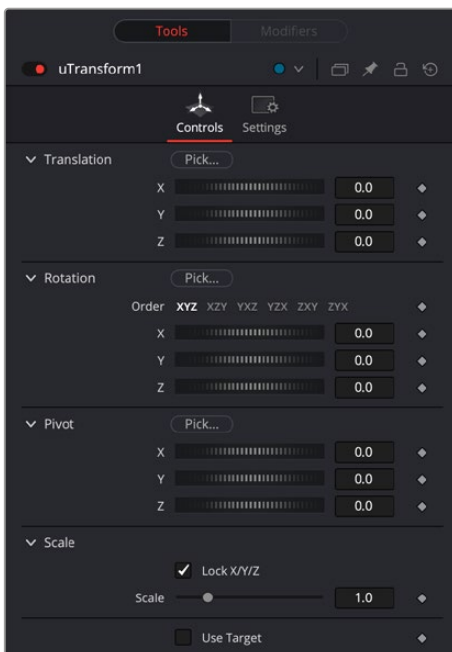
The USD Transform controls

Controls Tab

Translation

Transform controls are used to position the USD shape in 3D space.

Rotation Order

Use these buttons to select the order used to apply the rotation along each axis of the object.
For example, XYZ would apply the rotation to the X-axis first, followed by the Y-axis, and then the Z-axis.

Rotation

Use these controls to rotate the shape around its pivot point. If the Use Target checkbox is selected, then the rotation is relative to the position of the target; otherwise, the global axis is used.

Pivot

A pivot point is the point around which an object rotates. Normally, an object rotates around its own center, which is considered to be a pivot of 0,0,0. These controls can be used to offset the pivot from the center.

Scale

If the Lock X/Y/Z checkbox is checked, a single scale slider is shown. This adjusts the overall size of the object. If the Lock checkbox is unchecked, individual X, Y, and Z sliders are displayed to allow scaling in any dimension.

Scale

Selecting the Use Target checkbox enables a set of controls for positioning an XYZ target. When the target is enabled, the object always rotates to face the target. The rotation of the object becomes relative to the target.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other USD nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

USD Lights

USD light tools are a powerful addition to Fusion's USD toolset. They allow for greater control and flexibility when creating and manipulating lighting in your USD scenes. There are six lights available in the USD toolset, they are:

uCylinder Light [uCL]
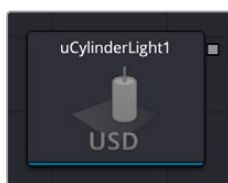
The USD Cylinder light

uCylinder Light Node Introduction

A uCylinder light is geometric light that has length and diameter, similar to fluorescent light. This light shows an onscreen control. Translation and rotation of the control place this in the scene.

Inspector
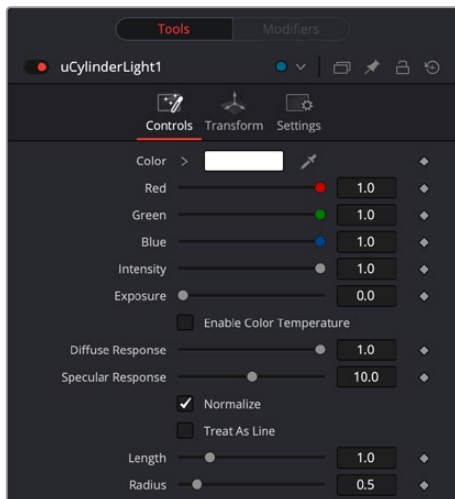
The USD Cylinder Light controls

Controls Tab

The Controls tab is used to set the color and brightness of the uCylinder light. The position, direction, and scale of the light source is controlled in the Transform tab.

Color

Use this standard Color control to set the color of the light.

Intensity

Use this slider to set the intensity of the light. A value of 0.2 indicates 20% percent light.

Exposure

This will change how much light will expose a scene; this is similar to Intensity.

Color Temperature / Enable

This sets the color temperature of a light source. Its default is 6500K, which is daylight temperature.

Diffuse Response

Controls the amount the light will contribute to the Diffuse color of a material.

Specular Response

Controls the amount the light will contribute to the Specular color of a material.

Normalize

This will normalize the light's contribution in the scene.

Treat As Line

This will make the light a simpler line light source.

Length

Defines the length of the light.

Radius

Defines the diameter of the light.

uDisk Light [uDi]
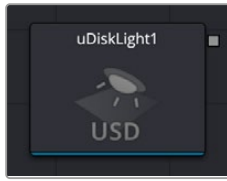
The USD Disk light

uDisk Light Node Introduction

A uDisk light is a round flat light, similar to an umbrella or softlight. The Controls tab is used to set the color and brightness of the uDisk light. The position, direction, and scale of the light source is controlled in the Transform tab.

Inspector
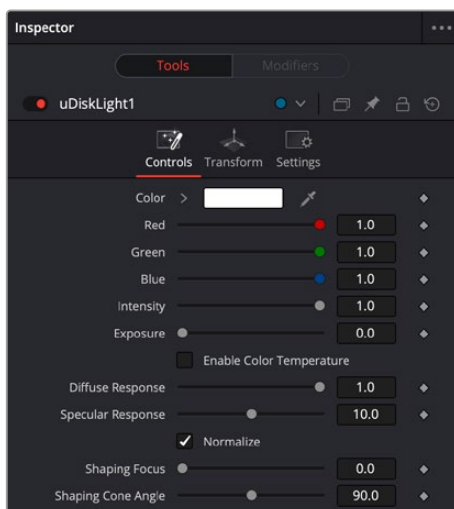
The USD Disk Light controls

Controls Tab

The Controls tab is used to set the color and brightness of the uDisk light. The position, direction, and scale of the light source is controlled in the Transform tab.

Color

Use this standard Color control to set the color of the light.

Intensity

Use this slider to set the intensity of the light. A value of 0.2 indicates 20% percent light.

Exposure

This will change how much light will expose a scene; this is similar to Intensity.

Color Temperature / Enable

This sets the color temperature of a light source. Its default is 6500K, which is daylight temperature.

Diffuse Response

Controls the amount the light will contribute to the Diffuse color of a material.

Specular Response

Controls the amount the light will contribute to the Specular color of a material.

Normalize

This will normalize the light's contribution in the scene.

Shaping Focus

This defines focus point of the lens effect of this light.

Shaping Cone Angle

Defines the spread of the light.

Radius

Defines the diameter of the light.

uDistant Light [uDL]
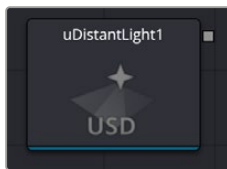
The USD Distant light

uDistant Light Node Introduction

A uDistant light is similar to a directional light and simulates distance source, similar to sunlight. This light shows an onscreen control. The rotation of the control is used to determine from where in the scene the light direction is pointing.

Inspector

The USD Distant Light controls

Controls Tab

The Controls tab is used to set the color and brightness of the uDistant light. The position, direction, and scale of the light source is controlled in the Transform tab.

Color

Use this standard Color control to set the color of the light.

Intensity

Use this slider to set the intensity of the light. A value of 0.2 indicates 20% percent light.

Exposure

This will change how much light will expose a scene; this is similar to Intensity.

Color Temperature / Enable

This sets the color temperature of a light source. Its default is 6500K, which is daylight temperature.

Diffuse Response

Controls the amount the light will contribute to the Diffuse color of a material.

Specular Response

Controls the amount the light will contribute to the Specular color of a material.

Normalize

This will normalize the light's contribution in the scene.

Angular Size

Defines the spread of the light.

uDome Light [uDo]
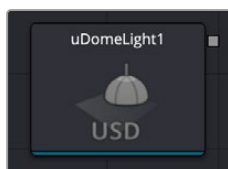
The USD Dome light

uDome Light Node Introduction

A uDome light uses SDR and HDR images and surrounds the entire scene in a sphere light mapped with the image, similar to environment light. This light shows an onscreen control. The rotation of the control is used to determine from where in the scene the light direction is pointing.

Inspector
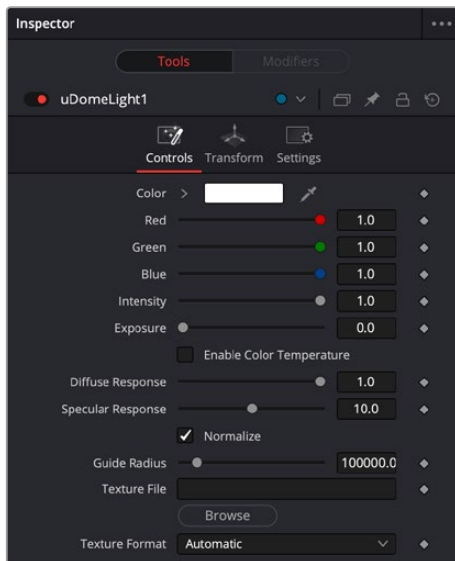
The USD Dome Light controls

Controls Tab

The Controls tab is used to set the color and brightness of the uDome light. The position, direction, and scale of the light source is controlled in the Transform tab.

Color

Use this standard Color control to set the color of the light.

Intensity

Use this slider to set the intensity of the light. A value of 0.2 indicates 20% percent light.

Exposure

This will change how much light will expose a scene; this is similar to Intensity.

Color Temperature / Enable

This sets the color temperature of a light source. Its default is 6500K, which is daylight temperature.

Diffuse Response

Controls the amount the light will contribute to the Diffuse color of a material.

Specular Response

Controls the amount the light will contribute to the Specular color of a material.

Normalize

This will normalize the light's contribution in the scene.

Guide Radius

This sets how large the dome is around the scene. The default is set to large and far away, similar to the outdoors. This radius can be set smaller to simulate a room size environment.

Texture File / Browse

This will load an image to map on the Domelight.

Texture Format

Sphere mapped images can be stored in different schemes; the common schemes are Lat-Long, MirrorBall, Angular, and Cube Mapped Vertical Cross.

uRectangle Light [uRL]

The USD Rectangle light

uRectangle Light Node Introduction

A uRectangle light is a flat light, similar to softlight or window . The Controls tab are used to set the color and brightness of the uRectangle light. The position, direction, and scale of the light source is controlled in the Transform tab.

Inspector
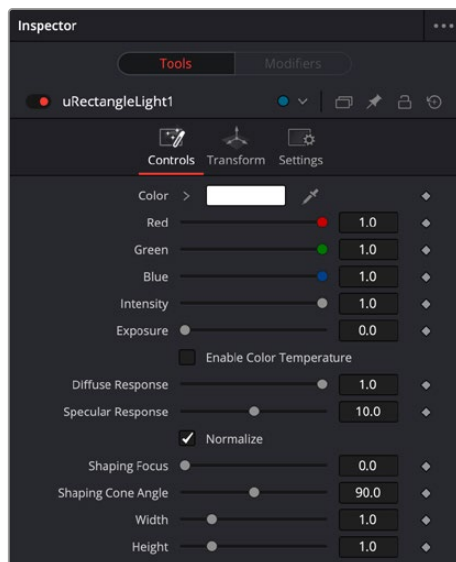
The USD Rectangle Light controls

Controls Tab

The Controls tab is used to set the color and brightness of the uDome light. The position, direction, and scale of the light source is controlled in the Transform tab.

Color

Use this standard Color control to set the color of the light.

Intensity

Use this slider to set the intensity of the light. A value of 0.2 indicates 20% percent light.

Exposure

This will change how much light will expose a scene; this is similar to Intensity.

Color Temperature / Enable

This sets the color temperature of a light source. Its default is 6500K, which is daylight temperature.

Diffuse Response

Controls the amount the light will contribute to the Diffuse color of a material.

Specular Response

Controls the amount the light will contribute to the Specular color of a material.

Normalize

This will normalize the light's contribution in the scene.

Shaping Focus

This defines focus point of the lens effect in this light.

Shaping Cone Angle

Defines the angle of spread of the light.

Width

Defines how wide the light appears.

Height

Defines how high the light appears.

uSphere Light [uSL]

The USD Sphere light

uSphere Light Node Introduction

A uSphere light is local light that is similar to point light but also has radius. This light shows an onscreen control. The rotation of the control is used to determine from where in the scene the light direction is pointing.

Inspector
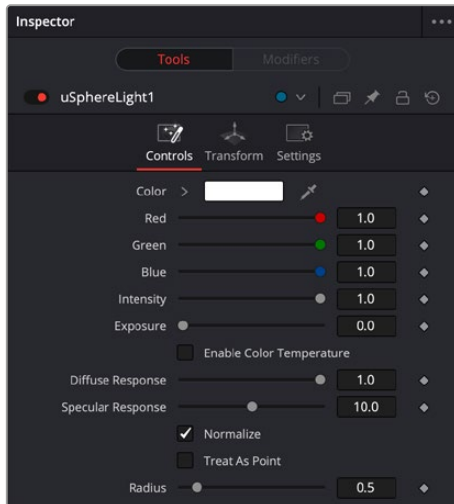
The USD Sphere Light controls

Controls Tab

The Controls tab is used to set the color and brightness of the uSphere light. The position, direction, and scale of the light source is controlled in the Transform tab.

Color

Use this standard Color control to set the color of the light.

Intensity

Use this slider to set the intensity of the light. A value of 0.2 indicates 20% percent light.

Exposure

This will change how much light will expose a scene; this is similar to Intensity.

Color Temperature / Enable

This sets the color temperature of a light source. Its default is 6500K, which is daylight temperature.

Diffuse Response

Controls the amount the light will contribute to the Diffuse color of a material.

Specular Response

Controls the amount the light will contribute to the Specular color of a material.

Normalize

This will normalize the light's contribution in the scene.

Treat as Point

This will make the light a simple classic point source.

Radius

This sets the size of the sphere light.

The Common Controls

Nodes that handle USD operations share several identical controls in the Inspector. This section describes controls that are common among USD nodes.

Transform Tab

The Transform tab can be found in the Inspector in most USD nodes. The controls are the same as the uTransform node, which can be applied separately.

Translation

Transform controls are used to position the USD shape in 3D space.

Rotation Order

Use these buttons to select the order used to apply the rotation along each axis of the object. For example, XYZ would apply the rotation to the X-axis first, followed by the Y-axis, and then the Z-axis.

Rotation

Use these controls to rotate the shape around its pivot point. If the Use Target checkbox is selected, then the rotation is relative to the position of the target; otherwise, the global axis is used.

Pivot

A pivot point is the point around which an object rotates. Normally, an object rotates around its own center, which is considered to be a pivot of 0,0,0. These controls can be used to offset the pivot from the center.

Scale

If the Lock X/Y/Z checkbox is checked, a single scale slider is shown. This adjusts the overall size of the object. If the Lock checkbox is unchecked, individual X, Y, and Z sliders are displayed to allow scaling in any dimension.

Settings Tab

The Settings tab in the Inspector can be found on every tool in the USD category. The controls are consistent and work the same way for each tool.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node will be displayed in the Inspector. Dragging a connected node from the node tree into the field will hide that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line will reappear.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Chapter 120

VR Nodes

This chapter details the Virtual Reality (VR) nodes available in Fusion.

> The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.
>
> For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

VR Nodes

You can create and fix spherical (360°) video, often described as Virtual Reality, or VR, using Fusion's set of VR nodes. Dome productions, planetariums, and other special-venue theaters have utilized the flexibility of Fusion and its 3D system to produce and deliver special content for years.

The equirectangular (lat-long) format often used for 360° video is similar to how a globe is represented by a flat world map, with the poles at the top and bottom edges of the image and the forward viewpoint at the center.

> **TIP:** You can create stereo VR using two stacked Lat Long images, one for each eye.
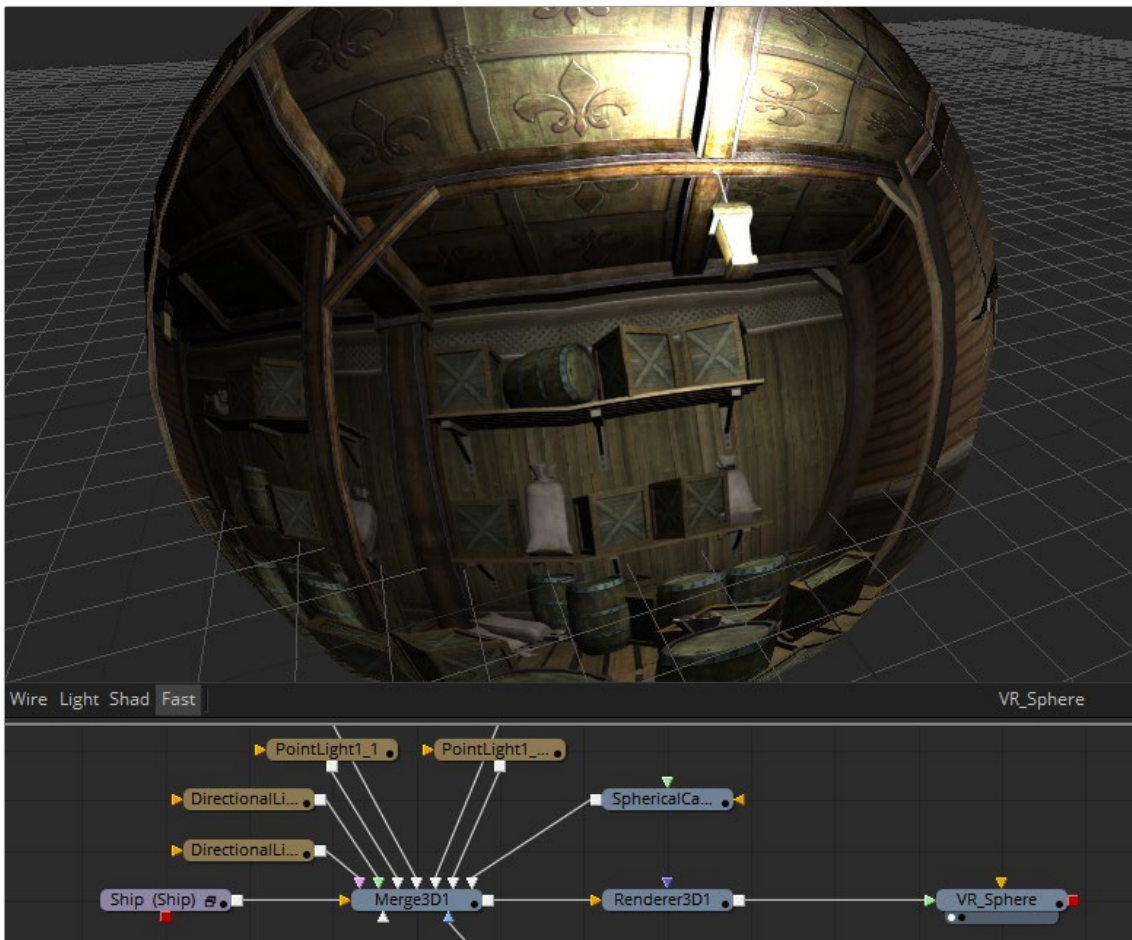
A VR Sphere displayed in the Node Editor and Fusion's 3D viewer

Fusion supports several common spherical image formats and can easily convert between them.

— **VCross and HCross:** VCross and HCross are the six square faces of a cube laid out in a cross, vertically or horizontally, with the forward view in the center of the cross in a 3:4 or 4:3 image.

— **VStrip and HStrip:** VStrip and HStrip are the six square faces of a cube laid vertically or horizontally in a line, ordered as Left, Right, Up, Down, Back, Front (+X, -X, +Y, -Y, +Z, -Z) in a 1:6 or 6:1 image.

— **LatLong:** LatLong is a single 2:1 image in an equirectangular mapping.

You can display both spherical video and live 3D scenes from the comp directly to headsets, including those from Oculus Rift and HTC Vive.

Fusion's "Fix it in post" tools for VR make it easy to perform several important tasks that are common in these types of productions.

Lat Long Patcher [LLP]

The Lat Long Patcher node

> **NOTE:** The VR category and Lat Long node are available only in Fusion Studio and DaVinci Resolve Studio.

Lat Long Patcher Node Introduction

Equirectangular stitched images often need patches, paintwork, or other VFX applied. The Lat Long Patcher extracts and de-warps a section of a lat-long (equirectangular) image to be treated, and can warp and merge fixes back over the original. You can quickly pick a section of the spherical image to patch or paint, and then apply it back to the original image. Note that matching rotations are used in both Extract and Apply modes, allowing a node's operation to be easily reversed by a copy or instance with the same rotation settings.

Input

The Lat Long Patcher node includes two inputs. The orange input accepts a 2D image in an equirectangular format, where the X-axis represents 0–360 degrees longitude, and the Y-axis represents –90 to +90 degrees latitude. The effect mask input is provided, although rarely used on VR nodes.

— **Image Input:** The orange image input accepts a equirectangular (lat-long) 2D RGBA image.
— **Effect Mask:** The effect mask input is provided, although rarely used on VR nodes.

Basic Node Setup

The Loader node connects to the input on a Lat Long Patcher node. The output of a Lat Long Patcher node is set to Extract. It is then connected to whatever image-processing operation is required. A second Lat Long Patcher node set to Apply takes an input from the processed extraction and merges it over the top of the original source.
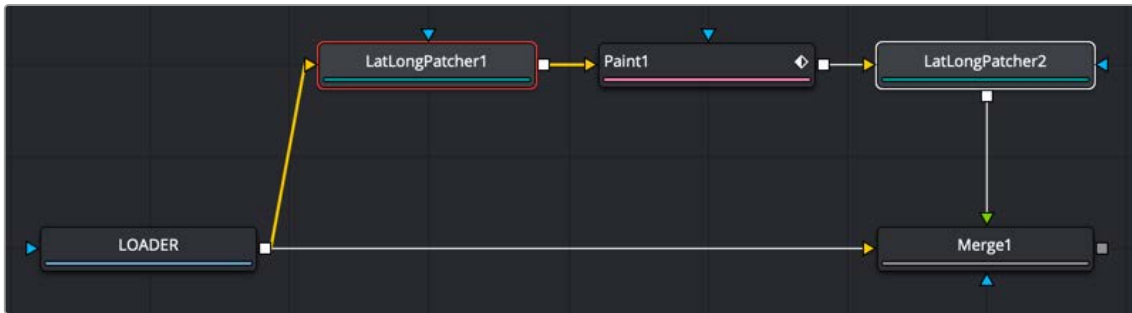
Two Lat Long Patchers used to repair a section

Inspector
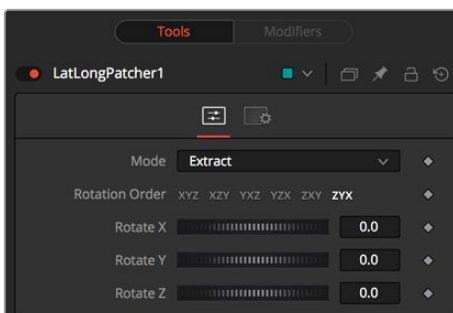
The Lat Long Patcher Controls tab

Controls Tab

The Controls tab is used to extract and later reapply a section from an equirectangular image. Rotation controls allow you to select the exact portion you need to repair.

Mode

— **Extract:** Pulls a de-warped 90-degree square image from the equirectangular image.

— **Apply:** Warps and merges a 90-degree square image over the equirectangular image. Because the square image's alpha is used, this allows, for example, paint strokes or text drawn over a transparent black background to be applied to the original equirectangular image, avoiding any double-filtering from de-warping and re-warping the original.

Rotation Order

These buttons choose the ordering of the rotations around each axis. For example, XYZ rotates first around the X axis (pitch/tilt), then around the Y axis (pan/yaw), and then around the Z axis (roll). Any of the six possible orderings can be chosen.

Rotation

These dials rotate the spherical image around each of the X, Y, and Z axes, offering independent control over pitch/tilt, pan/yaw, and roll, respectively.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other VR nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.
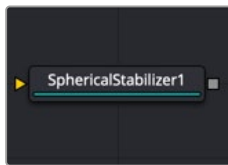
Pano Map [PaM]

The Pano Map node

> **NOTE:** The VR category and Pano Map node are available only in Fusion Studio and DaVinci Resolve Studio.

Pano Map Node Introduction

The Pano Map node converts images from one spherical layout to another, such as from a cube map to an equirectangular format. The node can also perform rotations of the spherical images when converting.

Input

The Pano Map node includes two inputs. The orange input accepts a 2D image in an equirectangular, cube map or other spherical formats. The effect mask input is provided, although rarely used on VR nodes.

— **Image Input:** The orange Image input accepts a spherical formatted 2D RGBA image that gets converted to another spherical format.

— **Effect Mask:** The effect mask input is provided, although rarely used on VR nodes.

Basic Node Setup

In the example below, a Loader node containing a Lat Long image connects to the input on a Pano Map node. The Pano Map node is used to convert the image to an H-Cross format. It is then connected to whatever image-processing operation is required.
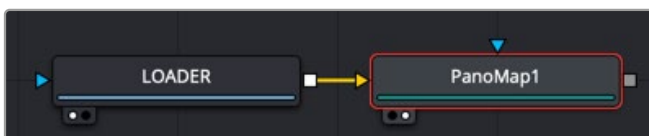
A Pano Map used to convert from one VR format to another

Inspector
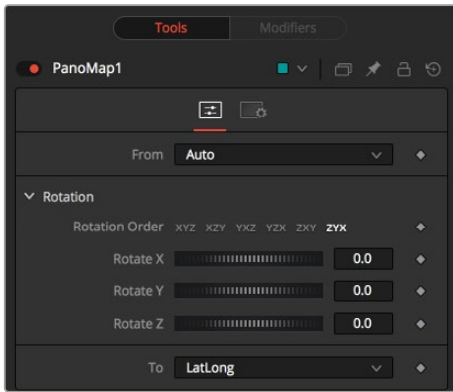
The Pano Map Controls tab

Controls Tab

The Controls tab is used to determine the format of the input image and the desired output format.

From/To

— **Auto:** Auto detects the incoming image layout from the metadata and image frame aspect.

— **VCross and HCross:** VCross and HCross are the six square faces of a cube laid out in a cross, vertically or horizontally, with the forward view in the center of the cross in a 3:4 or 4:3 image.

— **VStrip and HStrip:** VStrip and HStrip are the six square faces of a cube laid vertically or horizontally in a line, ordered as Left, Right, Up, Down, Back, Front (+X, -X, +Y, -Y, +Z, -Z) in a 1:6 or 6:1 image.

— **LatLong:** LatLong is a single 2:1 image in equirectangular mapping.

Rotation Order

These buttons choose the ordering of the rotations around each axis. For example, XYZ rotates first around the X axis (pitch/tilt), then around the Y axis (pan/yaw), and then around the Z axis (roll). Any of the six possible orderings can be chosen.

Rotation

These dials rotate the spherical image around each of the X, Y, and Z axes, offering independent control over pitch/tilt, pan/yaw, and roll, respectively.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other VR nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Spherical Camera [3SC]

The Spherical Camera node

Spherical Camera Node Introduction

The Spherical Camera is not located in the VR category of the Effects Library but in the 3D category. However, it is commonly used in creating and fixing VR content, so it is referenced here. The Spherical Camera allows the 3D Renderer node to output an image covering all viewing angles, laid out in several different formats. This image may be used, for example, as a skybox texture or reflection map or viewed in a VR headset. The Image Width setting in the 3D Renderer sets the size of each square cube face so that the resulting image may be a multiple of this size horizontally and vertically.

For more detail on the Spherical Camera node, refer to Chapter 29, "3D Nodes," in the Fusion Reference Manual or Chapter 89 in the DaVinci Resolve Reference Manual.

Spherical Stabilizer

The Spherical Stabilizer node

NOTE: The VR category and Spherical Stabilizer node are available only in Fusion Studio and DaVinci Resolve Studio.

Spherical Stabilizer Node Introduction

VR live action often uses handheld cameras, causing shaky footage to be a common problem. The Spherical Stabilizer node automatically identifies and tracks visible features in the footage, and then analyzes their movement to identify pan, tilt, and roll rotations. After tracking, it is then possible to smooth out or stabilize the rotation of the footage.

Inputs

The Spherical Stabilizer node has a single orange input.

— **Image:** This orange image input node requires an image in a spherical layout, which can be any of Lat Long (2:1 equirectangular), Horizontal/Vertical Cross, or Horizontal/Vertical Strip.

Basic Node Setup

In the example below, a 2:1 Lat Long image is connected to the input of the Spherical Stabilizer node. Once the image is stabilized, the output of the Spherical Stabilizer node is a steadied clip.
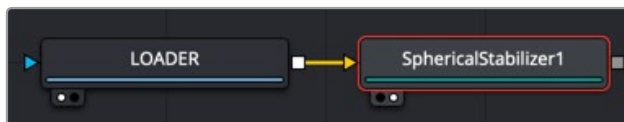
Spherical Stabilizer set up to steady a 2:1 Lat Long clip

Inspector
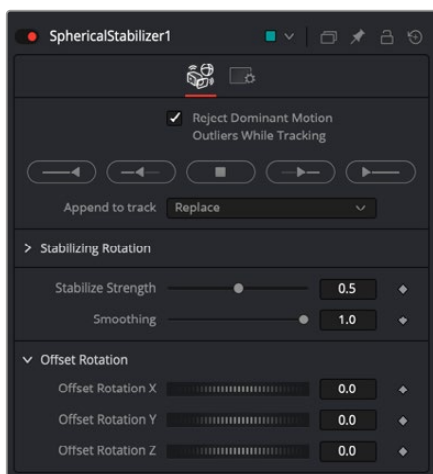
The Spherical Stabilizer Controls tab

Controls Tab

The Controls tab contains parameters to initiate the tracking and modify the results for stabilization or smoothing.

Reject Dominant Motion Outliers While Tracking

With this control activated (the default setting), features that move contrary to the majority of other features are ignored. This helps ignore the movement of subjects in the shot, preferring stable and consistent markers from the surrounding environment.

Track Controls

These buttons initiate tracking and analysis of the shot. Be aware that the reference frame used for stabilization is set to the first frame tracked.

— Track Backward from End Frame starts tracking backward from the end of the current render range.

— Track Backward from Current Time starts tracking backward from the current frame.

— Stop ceases tracking, preserving all results so far.

— Track Forward from Current Time starts tracking forward from the start of the current render range.

— Track Forward from Start Frame starts tracking forward from the current time.

Append to Track

— Replace causes the Track Controls to discard any previous tracking results and replace them with the newly-created track.

— Append adds the new tracking results to any earlier tracks.

Stabilization Strength

This control varies the amount of smoothing or stabilization applied, from 0.0 (no change) to 1.0 (maximum).

Smoothing

The Spherical Stabilizer node can eliminate all rotation from a shot, fixing the forward viewpoint (Still mode, 0.0) or gently smooth out any panning, rolling, or tilting to increase viewer comfort (Smooth mode, 1.0). This slider allows either option or anything in between.

Offset Rotation

Often a shot is not entirely level and needs the horizon to be realigned, or perhaps a desired pan should be reintroduced after fully stabilizing the shot. The Offset Rotation controls allow additional manual control of the Spherical Stabilizer's rotation of the footage, for pitch/tilt (X), pan/yaw (Y), and roll (Z), respectively. Rotation is always performed in the order X, Y, Z.

Common Controls

Settings Tab

The Settings tab in the Inspector is duplicated in other VR nodes. These common controls are described in detail in the following "The Common Controls" section.

The Common Controls

Nodes that handle VR operations share a number of identical controls in the Inspector. This section describes controls that are common among VR nodes.

Inspector
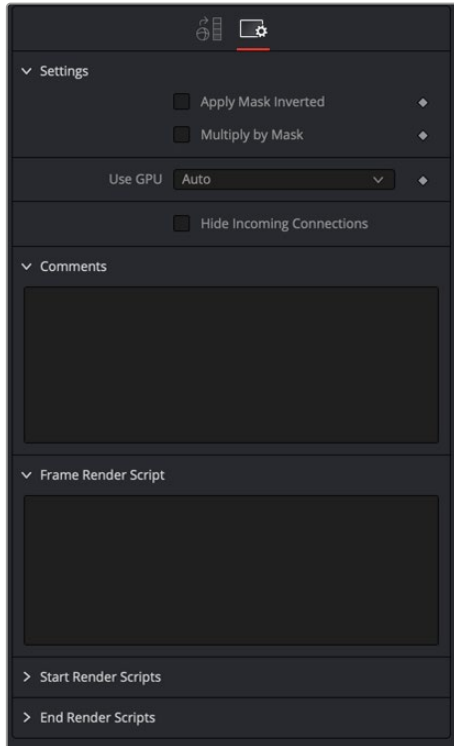
The VR Common controls

Settings Tab

The Settings tab in the Inspector can be found on every tool in the VR category. The controls are consistent and work the same way for each tool.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off hardware-accelerated rendering using the graphics card in your computer. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node are displayed in the Inspector. Dragging a connected node from the node tree into the empty field hides that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line reappears.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Chapter 121

Warp Nodes

This chapter details the Warp nodes available in Fusion.

The abbreviations next to each node name can be used in the Select Tool dialog when searching for tools and in scripting references.

For purposes of this document, node trees showing MediaIn nodes in DaVinci Resolve are interchangeable with Loader nodes in Fusion Studio, unless otherwise noted.

Contents

Coordinate Space [CdS]

The Coordinate Space node

Coordinate Space Node Introduction

The Coordinate Space node changes the coordinate space of the image from rectangular to polar or from polar to rectangular.

Inputs

The two inputs on the Coordinate Space node are used to connect a 2D image and an effect mask, which can be used to limit the distorted area.

— **Input:** The orange input is used for the primary 2D image that is distorted.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the distortion to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Coordinate Space node is used below to make a circular pattern based on a Fast Noise, Mosaic Blur (DaVinci Resolve Resolve FX only), and a Transform node. The Crop node at the end is used to set the desired resolution.
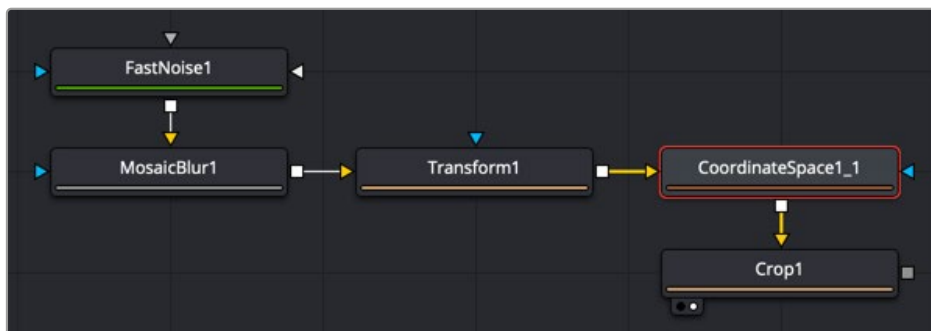
The Coordinate Space node can help create motion graphics backgrounds

> **EXAMPLE** To demonstrate a basic tunnel effect that can be achieved with this node:
>
> 1. Add a Text+ node with some text, and then animate it to move along a path from the top of the frame to the bottom.
>
> 2. Connect the output of the Text+ node to a Coordinate Space node.
>
> 3. Select Polar to Rectangular from the Shape menu.
>
> As the text moves from top to bottom along the original path, it appears to move from an infinite distance in the Coordinate Space node. It may be necessary to flip the text using a Transform node to make it appear the correct way in the Coordinate Space node. Another common use for the Coordinate Space node is to use it in pairs: two of them set to different Shape settings with a Drip or Transform node in between. When used in this way, the effect gets modified while the image remains the same.

Inspector
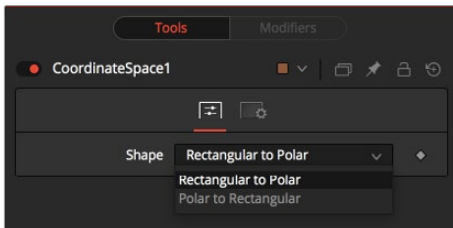
The Coordinate Space Controls tab

Controls Tab

The Controls tab Shape menu switches between Rectangular to Polar and Polar to Rectangular. Consider the following example to demonstrate the two coordinate spaces.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Warp nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Corner Positioner [CPn]

The Corner Positioner node

Corner Positioner Node Introduction

The Corner Positioner can be used to position the four corners of an image interactively. This would typically be used to replace a sign or other rectangular portion of a scene. Connect all corners to Paths or Trackers for animation purposes.

Inputs

The two inputs on the Corner Positioner node are used to connect a 2D image and an effect mask, which can be used to limit the warped area.

— **Input:** The orange input is used for the primary 2D image that is warped.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Corner Positioner to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

Below, the Corner Positioner is used to position the rectangular corners of the MediaIn2 to fit within a specific area of the MediaIn1 node. A Planar Tracker tracks the background, and then a Planar Transform is used to keep the Corner Positioner in place as the background clip moves. Once the planar tracking is completed, and the Planar Transform is created, the Planar Tracker node is no longer needed, and you can delete it.

The Corner Positioner corner pins a clip in place and uses a Planar Transform to match move it to the background.

Inspector
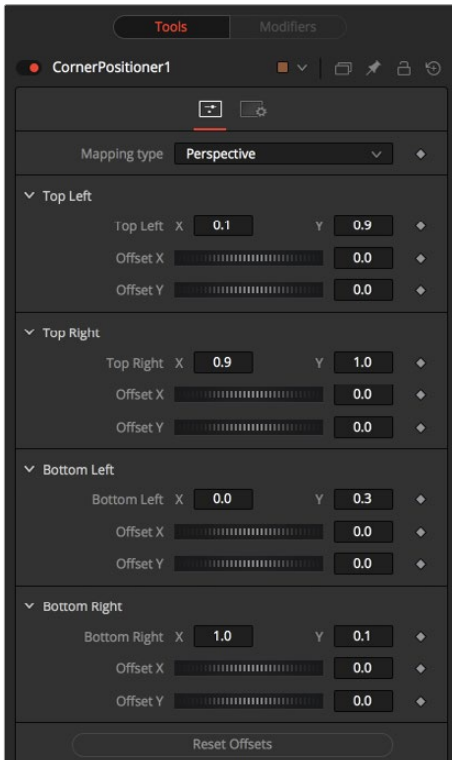
The Corner Positioner Controls tab

Controls Tab

The Controls tab includes transform and offset adjustments for the four corners of the image

Mapping Type

This determines the method used to project the image caused by the Corner Positioner. In Bi-Linear mode, a straight 2D warping takes place. In Perspective mode, the image is calculated with the offsets in 2D space and then mapped into a 3D perspective.

Corners X and Y

There are four points in the Corner Positioner. Drag these around to position each corner of the image interactively. Attach these control points to any of the usual modifiers.

The image input is deformed and perspective corrected to match the position of the four corners.

Offset X and Y

These controls can be used to offset the position of the corners slightly. This is useful when the corners are attached to Trackers with patterns that may not be positioned exactly where they are needed.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Warp nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Dent [Dnt]
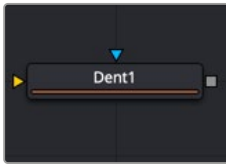
The Dent node

Dent Node Introduction

The Dent node creates a circular deformation of an image similar to a Fish Eye Lens effect, with the choice of six different Dent filters.

Inputs

The two inputs on the Dent node are used to connect a 2D image and an effect mask, which can be used to limit the warped area.

— **Input:** The orange input is used for the primary 2D image that is warped.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the Dent to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

The Dent node is used below to make a circular pattern based on a Fast Noise and a Mosaic Blur (DaVinci Resolve Resolve FX only). The Crop node at the end is used to set the desired resolution.
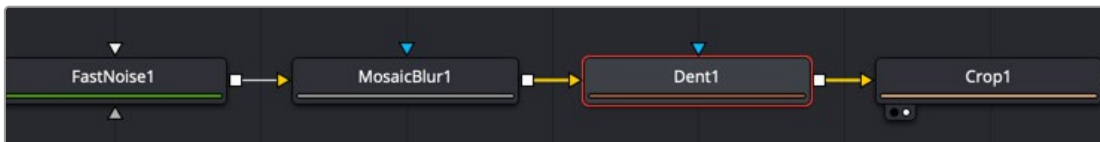
The Dent node can help create lens distortion effects or a motion graphics background.

Inspector
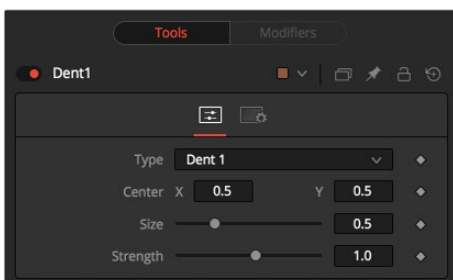
The Dent Controls tab

Controls Tab

The adjustments in the Controls tab are used to change the Dent style, position, size, and strength.

Type

Select the type of Dent filter to use from this menu. All parameters for the Dent can be keyframed.

Dent 1

This creates a bulge dent.

Kaleidoscope

This creates a dent, mirrors it, and inverts it.

Dent 2

This creates a displacement dent.

Dent 3

This creates a deform dent.

Cosine Dent

This creates a fracture to a center point.

Sine Dent

This creates a smooth rounded dent.

Center X and Y

This positions the Center of the Dent effect on the image. The default values are 0.5, 0.5, which center the effect in the image.

Size

This changes the size of the area affected by the dent. Animate this slider to make the dent grow.

Strength

This changes the overall strength of the dent.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Warp nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Displace [Dsp]

The Displace node

Displace Node Introduction

The Displace node uses a map image to displace or refract another image. This is useful for creating a vast range of effects from bevels and heat distortion to glass and water effects.

Inputs

There are three inputs on the Displace node: The primary image, the displacement map foreground image, and an effect mask.

— **Input:** The orange image input is a required connection for the primary image you wish to displace.

— **Foreground Image:** The green input is also required as the image used to displace the background. Once connected, you can choose red, green, blue, alpha, or luminance channel to create the displacement.

— **Effect Mask:** The optional blue effect mask input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the displacement to only those pixels within the mask. An effects mask is applied to the tool after it is processed.

Basic Node Setup

Below, the Displace node uses a Fast Noise to generate the Displace map. Increasing the seethe rate can produce heat distortion or flag waving effects.

The Displace node using a Fast Noise node for the Displace map

Inspector
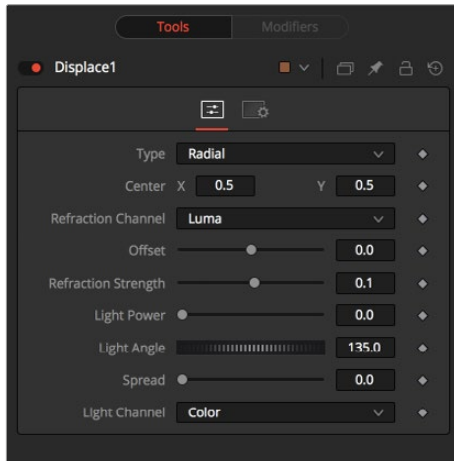
The Displace Controls tab

Controls Tab

The Controls tab is used to change the style, position, size , strength, and lighting (embossing) of the displacement.

Type

The Type menu is used to choose in what mode the Displace node operates. The Radial mode uses the map image that refracts each pixel out from the center, while X/Y mode provides control over the amount of displacement along each axis individually.

> **NOTE:** There is one set of Refraction controls while in Radial mode, and two sets in XY mode-one for each of the X and Y channels.

Center (Radial Only)

The Center control defines the point from which pixels are displaced toward or away.

Refraction Channel

This drop-down menu controls which channel from the foreground image is used to displace the image. Select from Red, Green, Blue, Alpha, or Luminance channels. In XY mode, this control appears twice, once for the X displacement and once for the Y displacement.

Refraction Strength (Radial)

Controls the strength of the refraction. Higher values cause stronger or more pronounced refraction.

X and Y Refraction (X/Y)

Two separate sliders appear to control the Refraction strength along the X- and Y-axis separately. Otherwise, this is exactly like Refraction Strength.

Light Power

This controls the intensity, or strength, of the simulated light, causing bright and dim areas to form according to the contour of the refraction image. Higher values cause the bright and dim areas to be more pronounced.

Light Angle

This sets the angle of the simulated light source.

Spread

This widens the Displacement effect and takes the edge off the Refraction map. Higher values cause the ridges or edges to spread out.

Light Channel

Select the channel from the refraction image to use as the simulated light source. Select from Color, Red, Green, Blue, Alpha, or Luminance channels.

> **NOTE:** The Radial mode pushes pixels inward or outward from a center point, based on pixel values from the Displacement map. The XY mode uses two different channels from the map to displace pixels horizontally and vertically, allowing more precise results. Using the XY mode, the Displace node can even accomplish simple morphing effects. The Light controls allow directional highlighting of refracted pixels for simulating a beveled look.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Warp nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Drip [DRP]
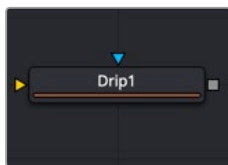
The Drip node

Drip Node Introduction

The Drip node creates a ripple effect over the entire image, which has the potential to animate outward from a central source. There are a variety of different Drip effects from which to choose.

Inputs

The two inputs on the Drip node are used to connect a 2D image and an effect mask, which can be used to limit the warped area.

— **Input:** The orange input is used for the primary 2D image that is warped.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the warping to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.
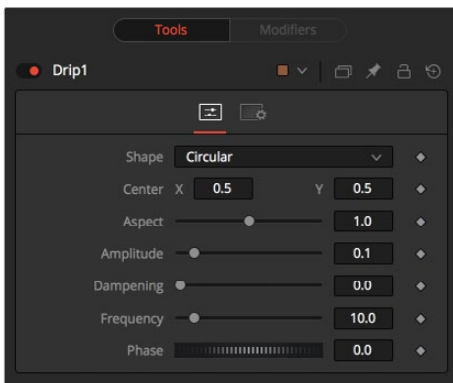
Basic Node Setup

Below, the Drip node is used to make rippling water-style effects using a MediaIn node.
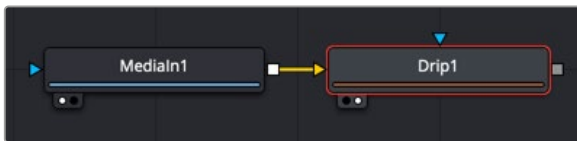
The Drip node can be connected directly after a MediaIn node or any node providing a 2D output.

Inspector

The Drip Controls tab

Controls Tab

The Controls tab is used to change the style, position, size , strength, and phase for animating the "ripples" of the Drip.

Shape

Use this control to select the shape of the Drip.

Circular

This creates circular ripples. This is the default Drip mode.

Square

This creates even-sided quadrilateral drips.

Random

This creates a randomly dispersed noise that distorts your image and is similar to a particle effect.

Horizontal

This creates horizontal waves that move in one direction.

Vertical

This creates vertical waves that move in one direction.

Exponential

This creates a Drip effect that looks like a diamond shape with inverted, curved sides (an exponential curve flipped and mirrored).

Star

This creates an eight-way symmetrical star-shaped ripple that acts as a kaleidoscope when the phase is animated.

Radial

This creates a star-shaped ripple that emits from a fixed pattern.

Center X and Y

Use this control to position the center of the Drip effect in the image. The default is 0.5, 0.5, which centers the effect in the image.

Aspect

Control the aspect ratio of the various Drip shapes. A value of 1.0 causes the shapes to be symmetrical. Smaller values cause the shape to be taller and narrower, while larger values cause shorter and wider shapes.

Amplitude

The Amplitude of the Drip effect refers to the peak height of each ripple. Use the slider to change the amount of distortion the Drip applies to the image. A value of 0.0 gives all ripples no height and therefore makes the effect transparent. A maximum Amplitude of 10 makes each ripple extremely visible and completely distorts the image. Higher numbers can be entered via the text entry boxes.

Dampening

Controls the Dampening, or falloff, of the amplitude as it moves away from the center of the effect. It can be used to limit the size or area affected by the Drip.

Frequency

This changes the number of ripples emanating from the center of the Drip effect. A value of 0.0 indicates no ripples. Move the slider up to a value of 100 to correspond with the density of desired ripples.

Phase

This controls the offset of the frequencies from the center. Animate the Phase value to make the ripple emanate from the center of the effect.

Common Controls

Settings Tab

The Settings Tab in the Inspector is also duplicated in other Warp nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Grid Warp [Grd]

The Grid Warp node

Grid Warp Node Introduction

The Grid Warp node is a 2D deformation grid with flexible vertices. The image is deformed so that the source grid matches the destination grid.

Inputs

The two inputs on the Grid Warp node are used to connect a 2D image and an effect mask, which can be used to limit the warped area.

— **Input:** The orange input is used for the primary 2D image that is warped.

— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the warping to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

Below, two Grid Warp nodes are used to warp different areas of a frame. This can be useful for chaining the framing in a shot or adding slight movement to a still. Using the Copy Src to Dest button, only the area modified by Grid Warp 2 is pasted as the foreground in the Merge.
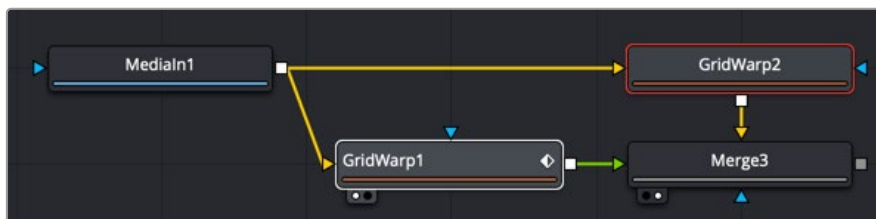
The Grid Warp can be used to shift areas within a shot for reframing or adding animation to a still.

Inspector
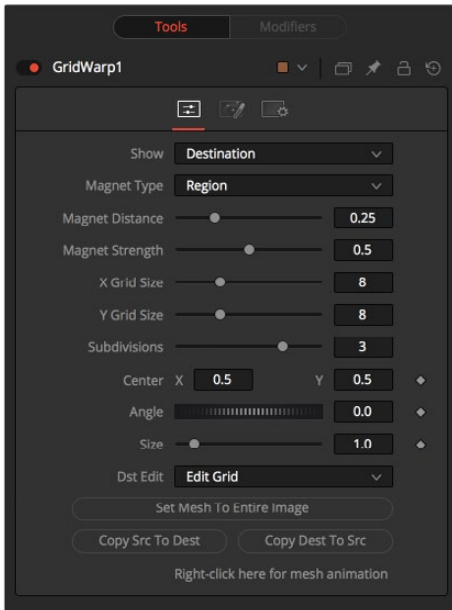
The Grid Warp Controls tab

Controls Tab

The Controls tab contains parameters that configure the onscreen grid as well the type of distortion applied when a control point on the grid is moved.

Source and Destination

The Source and Destination buttons determine whether the source grid or destination grid is currently active. Only one grid can be displayed or manipulated at a time. The selected button is highlighted to indicate that it is the currently active grid.

All other controls in this tab affect the grid selected by this control.

Selection Type

These three buttons determine the selection types used for manipulating the points. There are three options available.

Selected

When in Selected mode, adjustments to the grid are applied only to the currently selected points. This mode is identical to normal polyline operation.

Region

In Region mode, all points within the area around the mouse pointer move when the mouse button is clicked. New points that enter the region during the move are ignored. Choosing this option exposes Magnet Distance and Magnet Strength controls to determine the size and falloff of the area.

Magnetic

In Magnetic mode, all points within the area around the mouse pointer move when the mouse button is clicked. New points that enter the region during the move are affected as well. Choosing this option exposes Magnet Distance and Magnet Strength controls to determine the size and falloff of the area.

Magnet Distance

The default node for selecting and manipulating the grid is a Magnet node. The magnet is represented in the viewer by a circle around the mouse pointer. The Magnet Distance slider controls how large the region of effect for the magnet is, as in the size of the circle. Drag on the grid and any vertex within the range of the slider moves.

To increase the size of the magnet, increase the value of this slider. Alternately, adjust the size of the magnet by holding down the D key while dragging the mouse.

Magnet Strength

The Magnet Strength slider increases or decreases the falloff of the magnet cursor's effect. At a setting of 0.0, the magnetic cursor has no effect, and vertices do not move at all. As the values increase, the magnet causes a greater range of motion in the selected vertices. Use smaller values for a more sensitive adjustment and larger values for broad-sweeping changes to the grid.

X and Y Grid Size

The X and Y Grid Size sliders control the number of divisions in the grid. Where the X and Y divisions intersect, a control vertex is created.

Be aware that changing either of these controls after applying changes in the grid resets the entire grid. Set the X and Y grid sizes to the appropriate resolution before making detailed adjustments to the grid.

Subdivision Level

The Subdivision Level determines how many subdivisions there are between each set of divisions. Subdivisions do not generate vertices at intersections. The more subdivisions, the smoother the deformation is likely to be, but the slower it is to render.

Center

The Center coordinates determine the exact center of the grid. The onscreen Center control is invisible while editing the grid. Select the Edit Rect mode, and the grid center becomes visible and available for editing.

Use the Center control to move the grid through a scene without affecting the animation applied to the individual vertices. For example, while deforming lips, track the motion of the face with a Tracker, and connect the grid center to the Tracker. This matches the grid with slight movements of the head while focusing on the deformation of the lips.

Angle

This Angle control rotates the entire grid.

Size

The Size control increases or decreases the scale of the grid.

Edit Buttons

There are four edit modes available, each of which can be selected by clicking on the appropriate button.

Edit None

Set the grid to Edit None mode to disable the display of all onscreen controls.

Edit Grid

The Edit Grid mode is the default mode. While this mode is enabled, the grid is drawn in the viewer, and the control vertices of the grid can be manipulated directly.

Edit Rectangle

When the grid is in Edit Rectangle mode, the onscreen controls display a rectangle that determines the dimensions of the grid. The sides of the rectangle can be adjusted to increase or decrease the grid's dimension. This mode also reveals the onscreen Center control for the grid.

Edit Line

The Edit Line mode is beneficial for creating grids around organic shapes. When this mode is enabled, all onscreen controls disappear, and a spline can be drawn around the shape or object to be deformed. While drawing the spline, a grid is automatically created that best represents that object. Additional controls for Tolerance, Over Size, and Snap Distance appear when this mode is enabled. These controls are documented below.

Set Mesh to Entire Image

The Set Mesh to Entire Image button automatically resets the size of the grid to the exact dimensions of the image. Any adjustments to vertices within the grid are reset.

Copy Buttons

These two buttons provide a technique for copying the exact shape and dimensions of the source grid to the destination, or the destination grid to the source. This is particularly useful after setting the source grid to ensure that the destination grid's initial state matches the source grid before beginning a deformation.

Point Tolerance

This control is visible only when the Edit Line mode is enabled. The Point Tolerance slider determines how much tessellation the grid applies to match the density of points in the spline closely. The lower this value, the fewer vertices there are in the resulting grid, and the more uniform the grid appears. Higher values start applying denser grids with variations to account for regions in the spline that require more detail.

Oversize Amount

This control is visible only when the Edit Line mode is enabled. The Oversize Amount slider is used to set how large an area around the spline should be included in the grid. Higher values create a larger border, which can be useful when blending a deformation back into the source image.

Snap Distance

This control is visible only when the Edit Line mode is enabled. The Snap Distance slider dictates how strongly the drawn spline attracts surrounding vertices. If a vertex is close enough to a spline's edge, the vertex moves to line up with the spline. The higher the value, the farther the reach of the spline.

## Right-Click Here for Mesh Animation

The grids are static by default. Right-clicking on the Right-Click Here for Mesh Animation label provides a contextual menu with options for animating the grid or connecting it to another grid in the composition.

The grid uses a Polychange spline. Any adjustment to the control points adds or modifies the keyframe for all points on that spline.

Right-Click Here for Shape Animation

This label appears only in the Edit Line mode. Right-clicking on the Right-Click Here for Shape Animation label reveals a pop-up menu used to animate the shaping polyline or to connect it to other polylines.
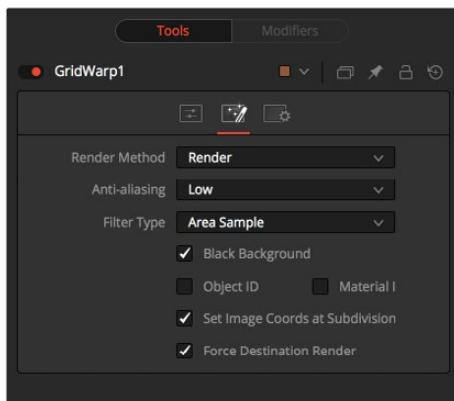
The Grid Warp Render tab

Render Tab

The Render tab controls the final rendered quality and appearance of the warping.

Render Method

The Render Method drop-down menu is used to select the rendering technique and quality applied to the mesh. The three settings are arranged in order of quality, with the first, Wireframe, as the fastest and lowest of quality. The default mode is Render, which produces final resolution, full-quality results.

Anti-Aliasing

The Anti-Aliasing control appears only as a checkbox when in Wireframe Render mode.

In other modes, it is a drop-down menu with three levels of quality. Higher degrees of anti-aliasing improve image quality dramatically but vastly increase render times. The Low setting may be an appropriate option while setting up a large dense grid or previewing a node tree, but rarely for a final render.

Filter Type

When the Render Method is set to something other than Wireframe mode, the Filter Type menu is visible and set to Area Sample. This setting prevents the grid from calculating area samples for each vertex in the grid, providing good render quality. Super Sample can provide even better results but requires much greater render times.

Wireframe Width

This slider appears only when the Render Method is set to Wireframe. It determines the width of the lines that make up the wireframe.

Anti-Aliased

This checkbox appears only when the Render Method is set to Wireframe. Use this checkbox to enable/disable anti-aliasing for the lines that make up the wireframe.

Black Background

The Black Background checkbox determines whether pixels outside of the grid in the source image are set to black or if they are preserved.

Object ID and Material ID

Enable the Object ID or Material ID checkboxes to have the grid output the proper ID channel in the final render.

Set Image Coordinates at Subdivision Level

This checkbox defaults to enabled and sets the image coordinates at the subdivision level.

Force Destination Render

This checkbox defaults to enabled and forces a destination render.

Contextual Menu Options

The Grid Warp node places a submenu for both source and destination grids in the viewer's contextual menu. Both menus have the exact same name, where only the menu for the active grid is populated with options. The other menu is empty. The contextual menu options are also available from the toolbar that appears in the viewer.
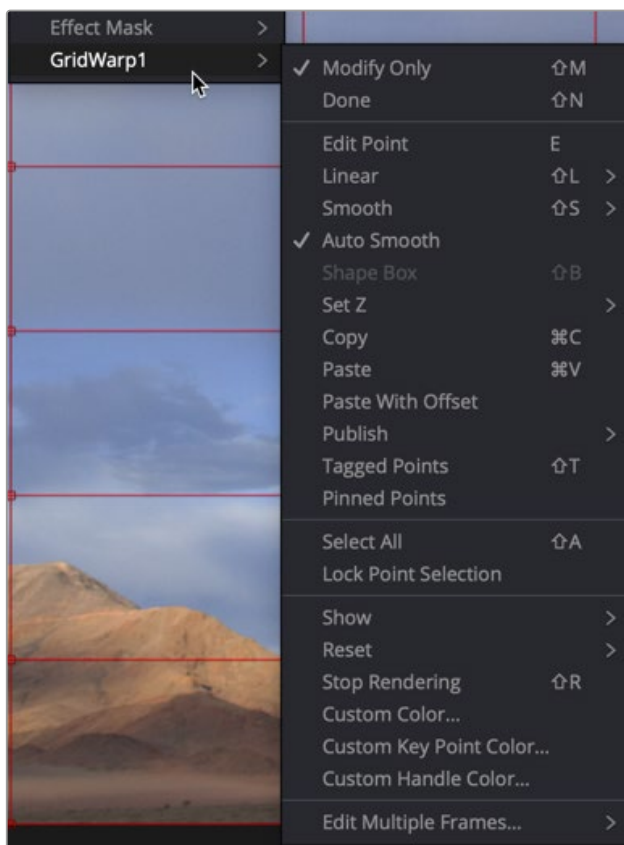
Grid Warp Contextual Menu options

Modify Only/Done

These two options set the mesh to Modify Only and Done modes, respectively. Select Modify Only to edit the mesh or Modify Done to prevent any further changes to a mesh.

Smooth/Linear

Use Smooth and Linear options to apply or remove smoothing from selected vertices.

Auto Smooth Points

When Auto Smooth Points is enabled, the vertices in the grid are automatically smoothed whenever they are moved. This is generally enabled by default.

Z Under/Z Same/Z Over

When two vertices in a grid overlap, one ends up getting clipped by the other. Z Under, Z Same, and Z Over are used to select which vertices are rendered on top and which are rendered behind.

Select All

This option selects all points in the mesh.

Show Key Points, Handles, Grid, and Subdivisions

Use these four options to enable or disable the display of the grid, key points (vertices), Bézier handles, and subdivisions in the viewers.

Reset Selected Points

This resets Selected Points (vertices) to their default positions.

Reset All Points

This resets all points (vertices) in the mesh to their default positions.

Stop Rendering

This option stops rendering, which disables all rendering of the Grid Warp node until the mode is turned off. This is frequently useful when making a series of fine adjustments to a complex grid.

Grid Warp Toolbar

Whenever the Grid Warp node is selected and is in Edit Grid mode, the Grid Warp toolbar is displayed in the views. This toolbar provides a variety of options for manipulating and adjusting the grid. The toolbar buttons in this toolbar are described in the preceding "Contextual Menu Options" section.
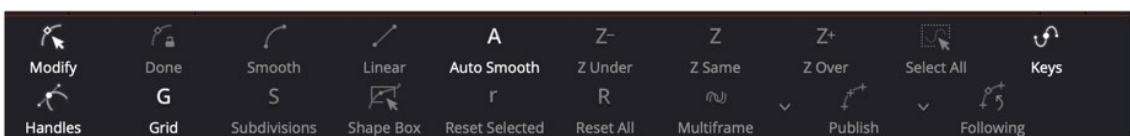
The Grid Warp viewer toolbar

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Warp nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Lens Distort [Lens]

The Lens Distort node

Lens Distort Node Introduction

This node can be used to remove or add lens distortion in an image. The lens distortion in an image depends on the lens, quality, number of elements, and many other factors.

One reason to remove lens distortion is to composite with an undistorted layer. For example, compositing a 3D element over a distorted live-action layer will cause unwanted effects like straight lines not matching up on the foreground and background. The resulting composite will not look believable.

Inputs

The two inputs on the Lens Distort node are used to connect a 2D image and an effect mask, which can be used to limit the distorted area.

— **Input:** The orange input is used for the primary 2D image that is distorted.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the distortion to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

A simplified example below applies the Lens Distort in Undistort mode to the MediaIn1 live-action layer, composites the 3D elements, and finally applies the Lens Distort at the end with the same settings, but this time in Distort mode to get the original look and distortion back into the image.
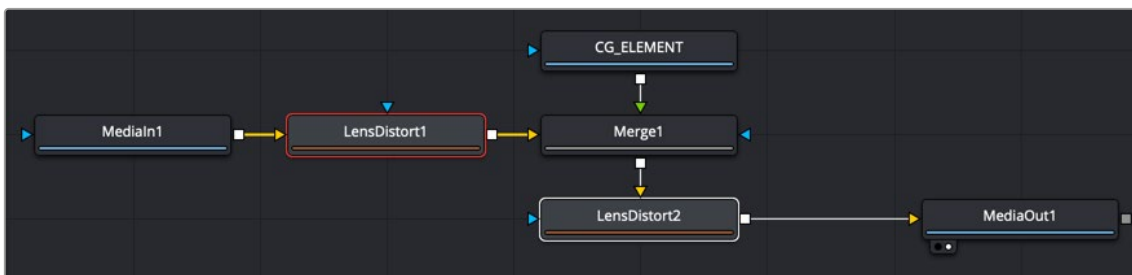
Lens Distort applied on the live-action media at the beginning of the node tree, and once again at the end

Inspector
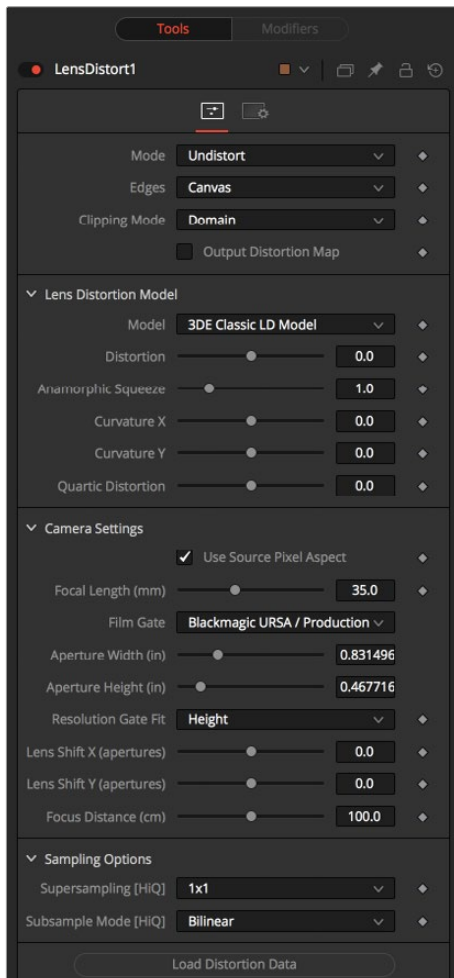
The Lens Distort Controls tab

Controls Tab

The Controls tab presents various ways to customize or build the lens distortion model you want. Camera Settings allow you to specify the camera used to capture the content.

Mode

Undistort removes the lens distortion to create a flattened image. Distort brings the original lens distortion back into the image.

Edges

Determines how samples that fall outside the frame are treated.

— **Canvas:** Pixels outside the frame are set to the default canvas color. In most cases, this is black with no alpha.

— **Duplicate:** Pixels outside the frame are duplicated. This results in "smeared" edges but is useful when, for example, applying a blur because in that case black pixels would result in the unwanted blurring between the actual image and the black canvas.

Clipping Mode

— **Domain:** Retains all pixels that might be moved out of the frame for later re-distorting.

— **Frame:** Pixels moved outside the frame are discarded.

Output Distortion Map

Outputs the location of pixels as a warped screen-coordinate map.

Camera Settings

The options known from the Camera 3D are duplicated here. They can either be set manually or connected to an already existing Camera 3D.

Lens Distortion Model

Select the appropriate 3D Equalizer Lens Distortion model here: 3DE Classic Model, 3DE4 Anamorphic, 3DE4 Radial Fisheye, or 3DE4 Radial. Please consult the 3D Equalizer manual for further explanation. The sliders in the 3DE Classic LD Model are most likely best suited for manually applying (un)distortion, without having imported lens data.

Supersampling [HiQ]

Sets the number of samples used to determine each destination pixel. As always, higher supersampling leads to higher render times. 1×1 bilinear is usually of sufficient quality, but with high lens distortion near the edges of the lens, there are noticeable differences to higher settings.

Supersampling Mode [HiQ]

The type of sample done for each supersample. Nearest leads to a crisper but more aliased image. Bi-Linear gives a blurrier result.

Load Distortion Data

Allows the user to load a Lens Distortion profile created, for example, by the 3D Equalizer.

How to Manually Determine Lens Distortion

In an ideal world, one would have exact lens parameters from each lens that was used during the shoot, and one could use those values to undistort the image. However, in the real world, those parameters have not been taken on set or don't match. Another approach is to use software like 3D Equalizer, which analyzes the footage and delivers a dataset that can be imported into the Lens Distort node right away.

And finally, one could try to manually eyeball the amount of lens distortion using the control sliders. To do that, one could either look for horizontal or vertical lines in the footage that are supposed to be straight and straighten them out using the controls, or shoot a full-frame checkerboard pattern on set as a reference.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Warp nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Perspective Positioner [PPn]

The Perspective Positioner node

Perspective Positioner Node Introduction

The Perspective Positioner is the complementary node to the Corner Positioner node. It "unpins" an image by positioning corner points on a perspective distorted area, thereby removing the perspective from the image. This function can also be used to wobble and warp the image by animating the points over time.

Inputs

The two inputs on the Perspective Positioner node are used to connect a 2D image and an effect mask, which can be used to limit the transformed area.

— **Input:** The orange input is used for the primary 2D image that is transformed.
— **Effect Mask:** The blue input is for a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the transform to only those pixels within the mask. An effects mask is applied to the tool after the tool is processed.

Basic Node Setup

In the example below, the Perspective Positioner is used to unpin a perspective distorted area of the MedianIn2 in order to paint on the flat texture. The MediaIn2 is then corner pinned back into place. The Perspective Positioner and Corner Positioner do not concatenate, so some softness is introduced with these nodes.
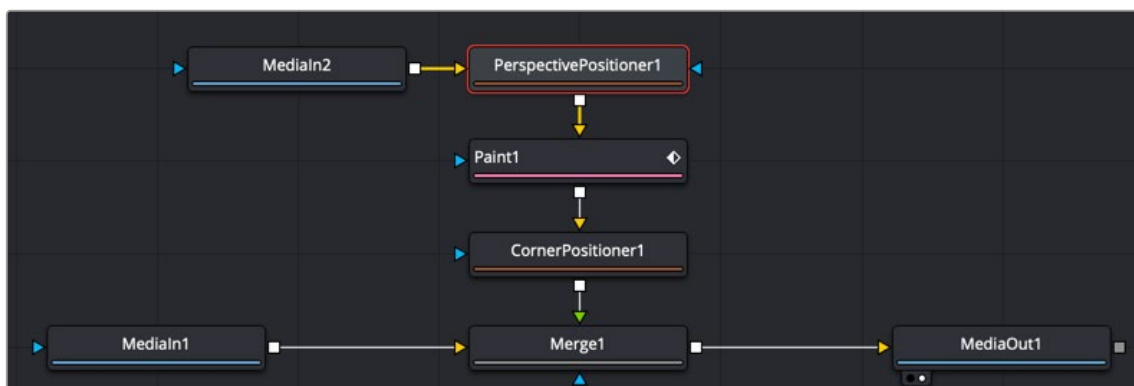
The Perspective Positioner unpins an image to paint on a texture map.

Inspector
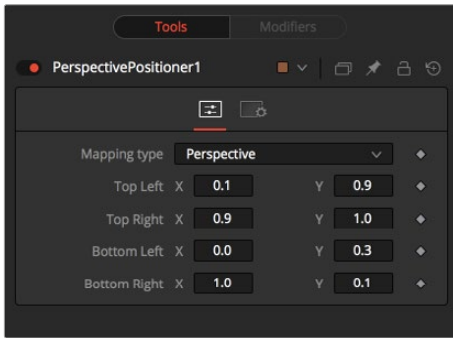
The Perspective Positioner Controls tab

Controls Tab

The Controls tab contains parameters for selecting vector channels and controlling how much distortion they apply to an image.

Mapping Type

The Mapping Type menu is used to select the type of transform used to distort the image. Bi-Linear is available for support of older projects. Leaving this on Perspective is strongly suggested since the Perspective setting maps the real world more accurately.

Corners X and Y

There are the four control points of the Perspective Positioner. Interactively drag these in the viewers to position each corner of the image. You can refine their position using the Top, Bottom, Left, and Right controls in the Inspector.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Warp nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Vector Distortion [Dst]
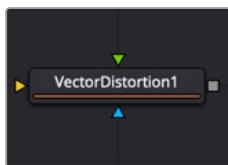
The Vector Distortion node

Vector Distortion Node Introduction

The Vector Distortion node distorts the main source image along the X- and Y-axis separately, based on the vector channel data in the source image or vector channels from a second reference image.

Inputs

There are three inputs on the Vector Distort node for the primary 2D image, the distort image with vector channels and an effect mask.

— **Input:** The orange image input is a required connection for the primary image you wish to distort. If this image has vector channels, they are used in the distortion.

— **Distort:** The green input is an optional distort image input used to distort the background image based on vector channels. Once connected, it overrides vector channels in the input image.

— **Effect Mask:** The optional blue effect mask input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the displacement to only those pixels within the mask. An effects mask is applied to the tool after it is processed.

Basic Node Setup

In the example below, the MediaIn1 node uses an Optical Flow node to generate the vector channels, which are then passed on to the Vector Distort node. MediaIn2 is distorted and composited over the background.
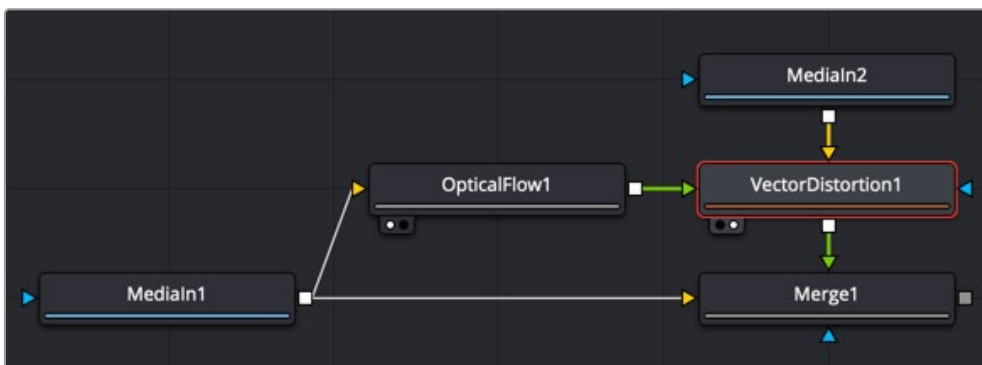
The Vector Distort node relies on the image having vector channels or an Optical Flow node to generate them.

Inspector
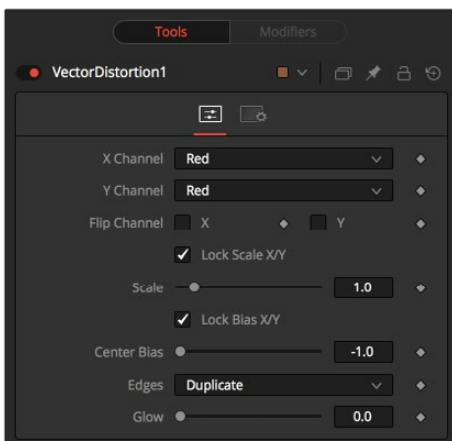
The Vector Distortion Controls tab

Controls Tab

The Controls tab contains parameters for selecting vector channels and controlling how much distortion they apply to an image.

X Channel and Y Channel

These two menus are used to select which channel of the (green) distort image input is used to distort the X and Y channels. If no distort reference image is connected, then channels from the main orange input are used instead.

Flip Channel X and Flip Channel Y

Use these checkboxes to flip the direction of the distortion along the specified axis.

Lock Scale X/Y

Select this checkbox to separate the Scale slider into separate Scale X and Scale Y sliders.

Scale

Use the Scale slider to apply a multiplier to the values of the distortion reference image.

Lock Bias X/Y

Select this checkbox to separate the Bias slider into separate Bias X and Bias Y sliders.

Center Bias

Use the Center Bias slider to shift or nudge the distortion along a given axis.

Edges

This menu determines how the edges of the image are treated.

— **Canvas:** This causes the edges that are revealed by the shake to be the canvas color—usually transparent or black.
— **Duplicate:** This causes the edges to be duplicated, causing a slight smearing effect at the edges.

Glow

Use this slider to add a glow to the result of the vector distortion.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Warp nodes. These common controls are described in detail at the end of this chapter in "The Common Controls" section.

Vortex [Vtx]

The Vortex node

Vortex Node Introduction

The Vortex effect appears as a swirling whirlpool in specified regions of the image. The Vortex can be made to move and grow by animating the various controls.

Inputs

There are two inputs on the Vortex node for the primary 2D image and the effect mask.

— **Input:** The orange image input is a required connection for the primary image you wish to swirl.
— **Effect Mask:** The optional blue effect mask input expects a mask shape created by polylines, basic primitive shapes, paint strokes, or bitmaps from other tools. Connecting a mask to this input limits the swirling vortex to only those pixels within the mask. An effects mask is applied to the tool after it is processed.

Basic Node Setup

Below, the Vortex is applied to text for creating motion graphics. Since the Vortex will cause the text to swirl outside the text boundary, a Set Domain node is used to expand the text boundary, ensuring that the text is not cropped when the Vortex is applied.

The Vortex is used to create swirling whirlpool effects.

Inspector
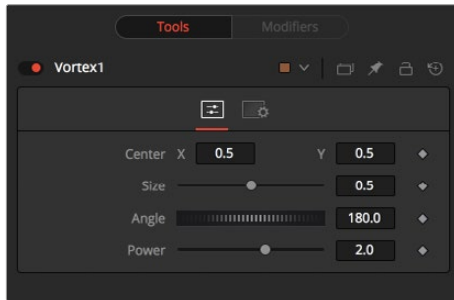
The Vector Distortion Controls tab

Controls Tab

The Controls tab contains parameters for adjusting the position, size, and strength of the Vortex effect.

Center X and Y

This control is used to position the center of the Vortex effect on the image. The default is 0.5, 0.5, which positions the effect in the center of the image.

Size

Size changes the area affected by the Vortex. You can drag the circumference of the effect in the viewer or use the Size slider.

Angle

Drag the rotation handle in the viewer or use the thumbwheel control to change the amount of rotation in the Vortex. The higher the angle value, the greater the swirling effect.

Power

Increasing the Power slider makes the Vortex smaller but tighter. It effectively concentrates it inside the given image area.

Common Controls

Settings Tab

The Settings tab in the Inspector is also duplicated in other Warp nodes. These common controls are described in detail in the following "The Common Controls" section.

The Common Controls

Nodes that handle Warp operations share a number of identical controls in the Inspector. This section describes controls that are common among Warp nodes.

Inspector
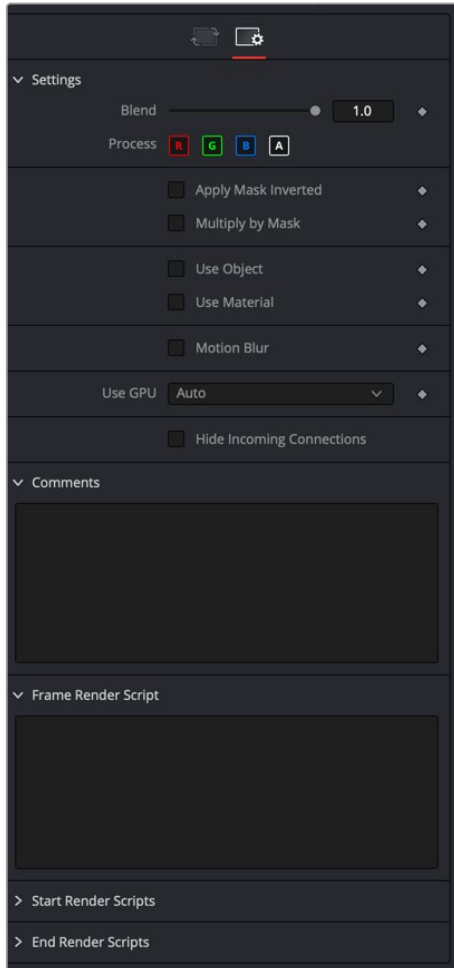
Warp Common Controls tab

Settings Tab

The Settings tab in the Inspector can be found on every tool in the Warp category. The Settings controls are even found on third-party Warp-type plugin tools. The controls are consistent and work the same way for each tool.

Blend

The Blend control is used to blend between the tool's original image input and the tool's final modified output image. When the blend value is 0.0, the outgoing image is identical to the incoming image. Normally, this will cause the tool to skip processing entirely, copying the input straight to the output.

Process When Blend Is 0.0

The tool is processed even when the input value is zero. This can be useful if the node is scripted to trigger a task, but the node's value is set to 0.0.

Red/Green/Blue/Alpha Channel Selector

These four buttons are used to limit the effect of the tool to specified color channels. This filter is often applied after the tool has been processed.

For example, if the Red button on a Blur tool is deselected, the blur will first be applied to the image, and then the red channel from the original input will be copied back over the red channel of the result.

There are some exceptions, such as tools for which deselecting these channels causes the tool to skip processing that channel entirely. Tools that do this will generally possess a set of identical RGBA buttons on the Controls tab in the tool. In this case, the buttons in the Settings and the Controls tabs are identical.

Apply Mask Inverted

Enabling the Apply Mask Inverted option inverts the complete mask channel for the tool. The mask channel is the combined result of all masks connected to or generated in a node.

Multiply by Mask

Selecting this option will cause the RGB values of the masked image to be multiplied by the mask channel's values. This will cause all pixels of the image not included in the mask (i.e., set to 0) to become black/transparent.

Use Object/Use Material (Checkboxes)

Some 3D software can render to file formats that support additional channels. Notably, the EXR file format supports Object ID and Material ID channels, which can be used as a mask for the effect. These checkboxes determine whether the channels will be used, if present. The specific Material ID or Object ID affected is chosen using the Object and Material ID sliders explained below.

Correct Edges

This checkbox appears only when the Use Object or Use Material checkboxes are selected. It toggles the method used to deal with overlapping edges of objects in a multi-object image. When enabled, the Coverage and Background Color channels are used to separate and improve the effect around the edge of the object. If this option is disabled (or no Coverage or Background Color channels are available), aliasing may occur on the edge of the mask.

For more information on the Coverage and Background Color channels, *see Chapter 78, "Understanding Image Channels,"* in the DaVinci Resolve Reference Manual, or Chapter 16 in the Fusion Reference Manual.

Object ID/Material ID (Sliders)

Use these sliders to select which ID will be used to create a mask from the object or material channels of an image. Use the Sample button in the same way as the Color Picker: to grab IDs from the image displayed in the viewer. The image or sequence must have been rendered from a 3D software package with those channels included.

Motion Blur

— **Motion Blur:** This toggles the rendering of Motion Blur on the tool. When this control is toggled on, the tool's predicted motion is used to produce the motion blur caused by the virtual camera's shutter. When the control is toggled off, no motion blur is created.

— **Quality:** Quality determines the number of samples used to create the blur. A quality setting of 2 will cause Fusion to create two samples to either side of an object's actual motion. Larger values produce smoother results but increase the render time.

— **Shutter Angle:**L Shutter Angle controls the angle of the virtual shutter used to produce the motion blur effect. Larger angles create more blur but increase the render times. A value of 360 is the equivalent of having the shutter open for one full frame exposure. Higher values are possible and can be used to create interesting effects.

— **Center Bias:** Center Bias modifies the position of the center of the motion blur. This allows for the creation of motion trail effects.

— **Sample Spread:** Adjusting this control modifies the weighting given to each sample. This affects the brightness of the samples.

Use GPU

The Use GPU menu has three settings. Setting the menu to Disable turns off GPU hardware-accelerated rendering. Enabled uses the GPU hardware for rendering the node. Auto uses a capable GPU if one is available and falls back to software rendering when a capable GPU is not available.

Hide Incoming Connections

Enabling this checkbox can hide connection lines from incoming nodes, making a node tree appear cleaner and easier to read. When enabled, empty fields for each input on a node will be displayed in the Inspector. Dragging a connected node from the node tree into the field will hide that incoming connection line as long as the node is not selected in the node tree. When the node is selected in the node tree, the line will reappear.

Comments

The Comments field is used to add notes to a tool. Click in the empty field and type the text. When a note is added to a tool, a small red square appears in the lower-left corner of the node when the full tile is displayed, or a small text bubble icon appears on the right when nodes are collapsed. To see the note in the Node Editor, hold the mouse pointer over the node to display the tooltip.

Scripts

Three Scripting fields are available on every tool in Fusion from the Settings tab. They each contain edit boxes used to add scripts that process when the tool is rendering. For more details on scripting nodes, please consult the Fusion scripting documentation.

Modifiers

This chapter details the modifiers available in Fusion.

Contents

Modifiers

Modifiers are extensions to a node's standard set of parameters found in the Inspector; in fact, modifiers are designed to control other parameters. They can be as simple as a motion path or linking two parameters. However, they can also be elaborate expressions, procedural functions, external data, third-party plugins, or scripted Fuses. You can add modifiers by right-clicking over a parameter in the Inspector and choosing a modifier from the menu. Alternatively, you can right-click a control in the viewer. Not all modifiers are displayed in the right-click menu for all parameters. Some modifiers work only on specific parameter types.

> **NOTE:** Text3D and Text+ have additional text-specific modifiers, which are covered in their nodes' sections.
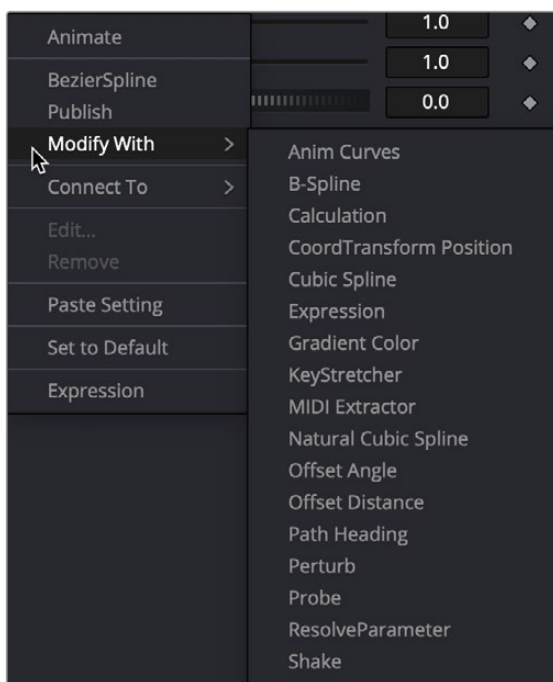
Modifiers right-click menu

Anim Curves

The Animation Curves modifier (Anim Curves) is used to dynamically adjust the timing, values, and acceleration of an animation, even if you decide to change the duration of a comp. Using this modifier makes it infinitely easier to stretch or squish animations, create smooth motion, add bouncing properties, or mirror animations curves without the complexity of manually adjusting splines.
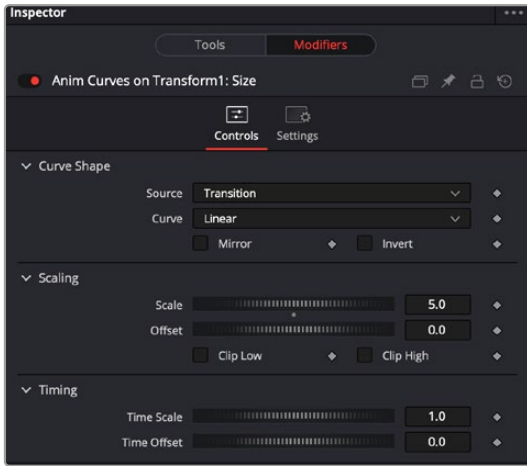
The Inspector for the Anim Curves modifier

When creating Fusion templates for the Edit and Cut page in DaVinci Resolve, the Anim Curves modifier allows the keyframed animation you've created in Fusion to stretch and squish appropriately as the transition, title, or effect's duration changes on the Edit and Cut page Timelines.

Curve Shape Controls

The controls for the Anim Curves modifier appear in the modifier's tab of the Inspector. The Curve Shape controls determine the acceleration or shape of the animation curve.

— **Source:** This drop-down menu has three options based on how the comp is created from DaVinci Resolve's Edit page.

— **Transition:** This setting is automatically selected when the comp is created from an Edit page transition effect. If the duration of the transition is updated in the Edit page, the timing of the animation updates as well.

— **Duration:** Use this setting when the comp is created from a clip on the Edit page. The animation timing will update if the clip's duration changes by trimming.

— **Custom:** Displays an Input dial to manually control the timing.

— **Input:** This dial is only visible when Source is set to Custom. It is used to change the input keyframe value.

— **Curve:** The Curve drop-down menu selects the interpolation method used between keyframes. The three choices are: linear, easing, or custom.

— **Linear:** The default Linear interpolation method maintains a fixed, consistent acceleration between keyframes.

— **Easing:** Displays interpolation menus for both the start of the curve (In) and the end of the curve (Out).

— **Custom:** Opens a mini Spline Editor to customize the interpolation from the start of the animation to the end.

— **Mirror:** Plays the animation forward, and after reaching the end, it returns to the starting value. This causes the initial animation to be twice as fast, since the second half of the comp is used for the reverse animation.

— **Invert:** Flips the animation curve upside-down so that the values start high and end low.

Scaling

The Scale parameters modify the animation values using relative adjustments.

— **Scale:** This number is a multiplier applied to the value of the keyframes. If the Scale value is 2 and a keyframe has a value of 0, it remains 0. If the Scale value is 2 and a keyframe has a value of 10, the result is as if the keyframe is set to 20. This can be thought of as the ending value for the animation. It is best to set this while viewing the last frame in the comp.

— **Offset:** The offset is added to the keyframe values and can be thought of as the starting value for the animation. It is best to set this while viewing the first frame in the comp.

— **Clip Low:** Ensures the output value never dips below 0.0.

— **Clip High:** Ensures the output value never exceeds 1.0.

Timing

The Timing parameters adjust the animation timing using relative values.

— **Time Scale:** Stretches or squishes the animation, causing it to run faster or slower. A value of 1.0 keeps the animation running for the comp's duration (unless you have customized the animation using other controls in the Modifier).

— **Time Offset:** This value delays the animation as a fraction of its total duration. A value of 0.0 applies no delay. A value of 0.5 delays the animation starting point midway into the comp's duration.

Using the Anim Curves Modifier to Create a Custom Transition

To understand how to use the Anim Curves modifier for a transition, let's create a simple scaling dissolve.

1 Add a standard cross dissolve in the Edit page Timeline.

2 Right-click over the transition and choose Convert to Fusion Cross Dissolve.

3 Right-click over the transition again and choose Open in Fusion page.

4 Add a Transform node to MediaIn1 and to MediaIn2.

5 Select the Transform node attached to the MediaIn2.

6 In the Inspector, right-click the Size control, then choose Modify With > Anim Curves from the contextual menu. Adding this modifier to the Size control will cause the slider to animate from 0 to 1 for the Cross Dissolve's duration.

7 Select the Transform node attached to the MediaIn1.

8 In the Inspector, right-click the Size control, then choose Modify With > Anim Curves from the contextual menu.

9 At the top of the Inspector, click the Modifier tab and click the Invert button. Inverting the animation curves causes MediaIn1 to scale opposite of MediaIn2.

10 In the Modifiers tab, set the Curve drop-down menu to Easing, and experiment with the different ease-in ease-out curve types from the In/Out drop-down menus.

Once you create a Macro from this node tree and save it as a Transition template, you can apply it in the Edit page Timeline. If you change the transition duration in the Edit page, the animation timing will update appropriately.

Using the Anim Curves Modifier with Paths

To understand how to use the Anim Curves modifier's with a Path modifier, let's use the premise that you want to create text that falls from the top of the frame and bounces as it reaches the bottom of the frame.

1   In Fusion, create two keyframes that cause text to start at the top of the frame and drop to the bottom. This automatically creates a Path modifier.

2   In the Inspector's Modifier tab, right-click over the Displacement parameter and choose Insert > Anim Curves. The animation is normalized to the duration of the comp.

3   Set the Source menu to Duration, since this is not a transition and we are not customizing the duration.

4   From the Curve menu, choose Easing, then for the Out menu, choose Bounce.

5   Play the animation to see the Bounce animation.

6   To make the bounce occur halfway down the frame, change the Scale to .05.

7   To make the animation run twice as fast, enter 2.0 in the Time Scale parameter.

Once you create a macro from this node tree and save it as a Title template, you can apply it in the Edit page Timeline. If you change the title's duration in the Edit page, the animation timing will update appropriately.

> **TIP:** To view the resulting animation curve in the Spline Editor, select the parameter name in the Spline Editor's header. The spline is updated as you change the controls.
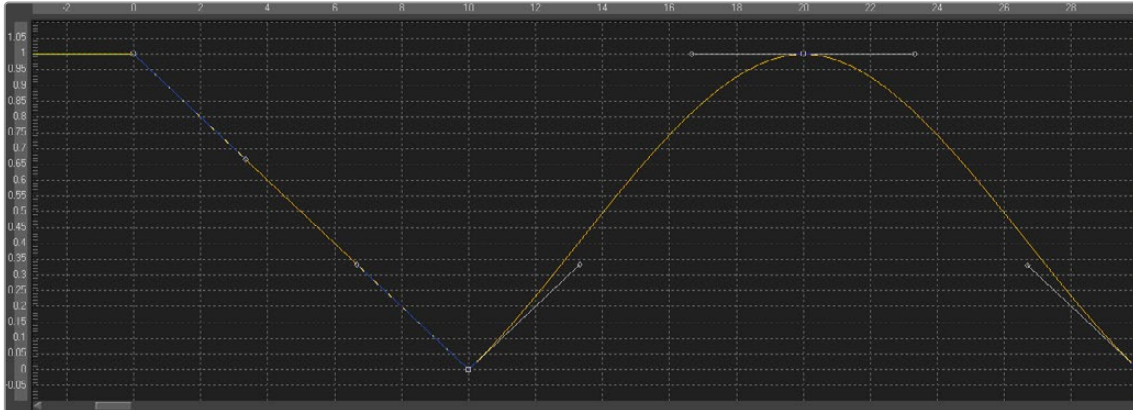
Bézier Spline

The Bézier Spline is one of the animation modifiers in Fusion and is typically applied to numerical values rather than point values. It is automatically applied when you keyframe a parameter or each time you right-click a number field and select Animate.
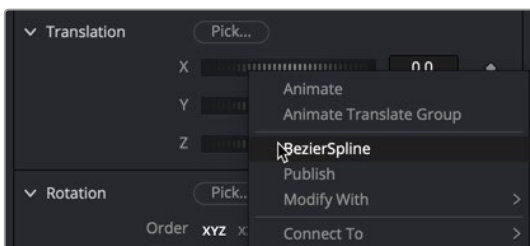
Bézier Spline modifier menu

Usage

You can add the Bézier Spline to the Spline Editor by right-clicking a number field and selecting BezierSpline. Since this is the most common choice for animation splines, it is separated from the Modify With menu for quicker access. Selecting BezierSpline from the menu adds a keyframe at the current location and displays a Bézier Spline in the Spline Editor.
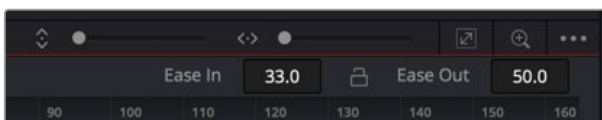

Bézier Spline Editor

Unlike most modifiers, this modifier has no actual Controls tab in the Inspector. However, the Spline Editor displays the Bézier Spline, and it can be controlled there. The Bézier Spline offers individual control over each control point's smoothness using Bézier handles. The smoothness is applied in multiple ways:

— To make the control points smooth, select them, and press Shift-S. The handles can be used to modify the smoothness further.

— To make the control points linear, select them, and press Shift-L. These operations can also be performed using the contextual menu.

— Select the control point(s), right-click, and select Smooth or Linear. The menu also allows the user to smooth a spline using a convolution analysis called a Savitzky-Golay filter. Select the control point(s), right-click, and select Smooth Points -Y Dialog.

Ease In/Out

Traditional Ease In/Out can also be modified by using the number field virtual sliders in the Spline Editor. Select the control points you want to modify, right-click, and select Ease In/Out... from the contextual menu. Then use the number field virtual sliders to control the Ease In/Out numerically.
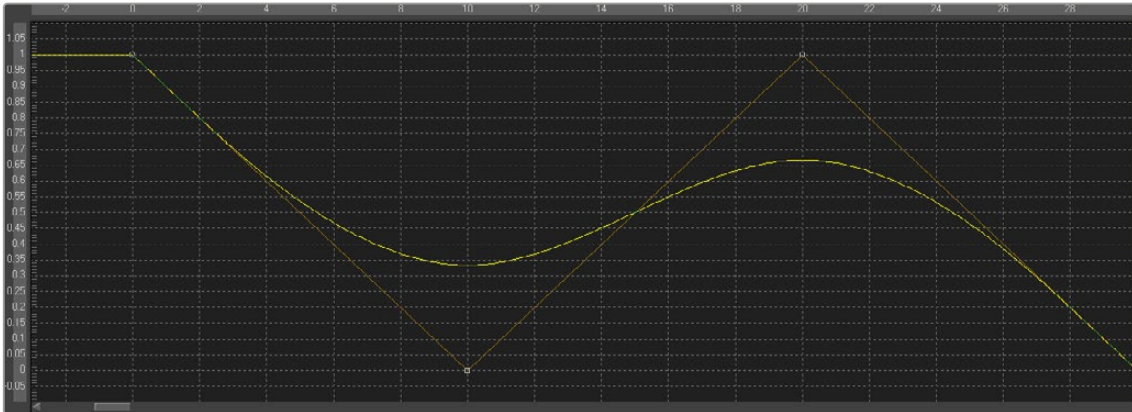

Spline Ease In/Out modifier

B-Spline

An alternative to the Bézier Spline, B-spline is another animation modifier in Fusion and is typically applied to numerical values rather than point values. It is applied by right-clicking a parameter and selecting Modify With > B-Spline.

Usage

B-Spline Editor

— This animation spline modifier has no actual Controls tab. However, the Spline Editor displays the B-spline, and it can be controlled there. Notice that, though the actual value of the second keyframe is 0, the value of the resulting spline is 0.33 due to the unique smoothing and weighing algorithms of a B-spline.

— The weight can be modified by clicking the control point to select it, holding the W key, and moving the mouse left and right to lower or increase the tension. This is also done with multiple selected control points simultaneously.

Calculation

Calculations are used to create indirect connections between parameters. A Calculation can perform a mathematical expression based on two operands, where each operand can be connected to another parameter or set manually.

Additionally, using Time offsets and Scale controls in the Time tab, the Calculation control can access values of a parameter at times other than the current time.

The Calculation's most common use is for connecting two parameters when one value range or scope is inappropriate for the other parameter.

> **NOTE:** The Expression modifier is essentially a more flexible version of the Calculation modifier, with a single exception. It is far easier to manipulate the timing of the operands provided in the Calculation modifier than it is to do so with an Expression.

Inspector
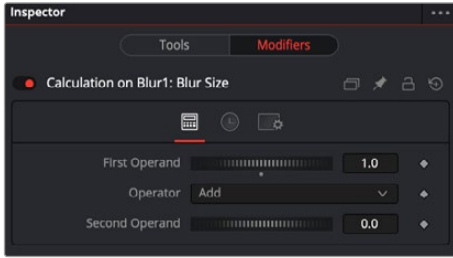
Calculation Calc tab

Calc Tab

The Calc tab includes two dials used for the connected parameter and value that gets mathematically combined. The Operator menu selects how the Second Operand value combines with the parameter's value.
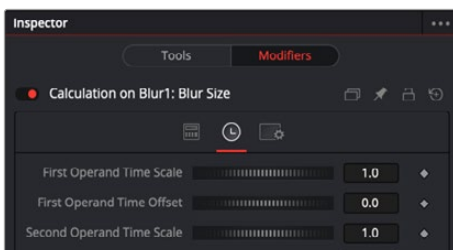
First and Second Operand

These sliders are connected to published or animated parameters or manually set to the desired values for the calculation.

Operator

Select from the mathematical operations listed in this menu to determine how the two operands are combined. Clicking the drop-down arrow opens the menu with the following options:

— Add

— Subtract (First - Second)

— Multiply

— Divide (First / Second)

— Divide (Second / First)

— Subtract (Second - First)

— Minimum

— Maximum

— Average

— First only

Calculation Time tab

Time Tab

The Time tab is used to modify the time of the Calculation modifier. The controls here retime the speed of the effect or offset it in time.

First and Second Operand Time Scale

These sliders multiply the frame number and return the value of the operands at the multiplied frame number. A value of 1 returns the value of the operand at frame x when the composition is on frame x. For example, if the first operand is animated with a value of 1 to 50 from frame 0 to 10, then a scale of 0.5 would cause the calculation to return a value of 25 at frame 10 (effectively slowing the animation by half for the purposes of the calculation).

First Operand and Second Operand Time Offset

These sliders return the value of the operand at the Time Offset specified. A value of 10 would return the value of the operand 10 frames forward in time, and -10 would return the value of the operand 10 frames back in time. See the example below for a practical example.

EXAMPLE

The following example uses a calculation to apply blur to a Text node in inverse proportion to the size of the text.

1. Open a new composition that starts on frame 0 and ends on frame 100.

2. At frame 0, add a Text+ node to the composition.

3. Enter a small amount of text and set the size to 0.05

4. Click the Keyframe button to the right of the Size slider to add a keyframe.

5. Move to frame 100 and set the Size value to 0.50.

6. Connect a Blur node after the Text+ node.

7. View the Blur node in one of the viewers.

   To have the blur decrease in strength as the text gets bigger, a simple "pick whip"-style parameter linking does not work. The controls cannot be directly connected together because the values of the Text Size control are getting bigger instead of smaller.

8. Right-click the Blur Size and select Modify With > Calculation from the contextual menu.

   This adds a Calculation modifier to the Blur node. At the top of the Inspector, a new set of controls appears in the Modifiers tab while the Blur node is selected.

9. At the top of the Inspector, select the Modifiers tab (F11).

10. Right-click the First Operand slider and select Connect To > Text 1 > Size from the contextual menu.

    Although the Blur Size is now connected to the Text Size parameter, this connection isn't very useful. The maximum value of the Blur Size control is 0.5, which is hardly noticeable as a blur.

11. Set the Operator drop-down menu to Multiply.

12. Set the Second Operand slider to 100.

    Now the first operand is multiplied by 100, and adjusting the dial gives you a much blurrier blur.

13. Switch to the Time tab of the modifier and set the First Operand Time Scale to -1.0.

    Normally, the first operand gets the value of the control it is connected to from the same frame as the current time. So at frame 10, the first operand is set to the same value as the Text size at frame 10. By setting this value to -1, the value is read from one frame back in time whenever the current time of the composition advances by 1 frame. However, this means that the Calculation would be reading the value of the Text size at frame -10 when we are at frame 10 in the composition.

14. To correct for this, set the First Operand Time Offset slider to 100.

15. Return to the Tools tab at the top of the Inspector and press Play (Spacebar) to watch how the value of the Blur Size relates to the value of the Text Size.

CoordTransform Position

Because of the hierarchical nature of 3D in Fusion, the original position of an object in a 3D scene often fails to indicate the current position of the object. For example, an image plane might initially have a position at 1, 2, 1, but then be scaled, offset, and rotated by other tools further downstream in the node tree, ending up with an absolute location of 10, 20, 5. This can complicate connecting an object further downstream in the composition directly to the position of an upstream object. The Coordinate Transform modifier can be added to any set of the XYZ coordinate controls to calculate the current position of a given object at any point in the scene hierarchy. To add a Coordinate Transform modifier, right-click the numeric input on any node, and select Modify With > CoordTransform Position from the contextual menu.

Inspector
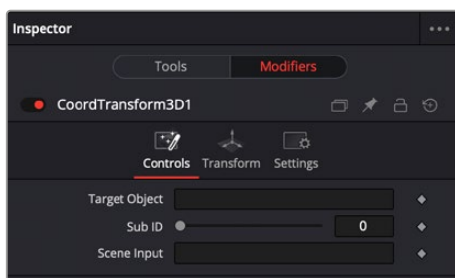
The Coordinate Transform
modifier Controls tab

Controls Tab

The Controls tab has two fields for the target and scene input. The target is for the node continuing the original coordinates, while the scene input is used for the scene with the new coordinates.

Target Object

This control is connected to the 3D tool that produces the original coordinates to be transformed. To connect a tool, drag the node from the Node Editor into the text edit control, or right-click the control and select the tool from the contextual menu. It is also possible to type the tool's name directly into the control.

SubID

The SubID slider can be used to target an individual sub-element of certain types of geometry, such as an individual character produced by a Text 3D tool or a specific copy created by a Duplicate 3D tool.

Scene Input

This control should be connected to the 3D tool, which outputs the scene containing the object at the new location. To connect a tool, drag and drop a tool tile from the Node Editor into the text edit control, or right-click the control and select an object from the Connect To pop-up menu.

Cubic Spline

The Cubic Spline is another animation modifier in Fusion that is normally applied to numerical values rather than point values. It can be applied by right-clicking a numerical control and selecting Modify With > Natural Cubic Spline.

Usage

Being an animation spline, this modifier has no actual Controls tab. However, its effect can be seen and influenced in the Spline Editor.
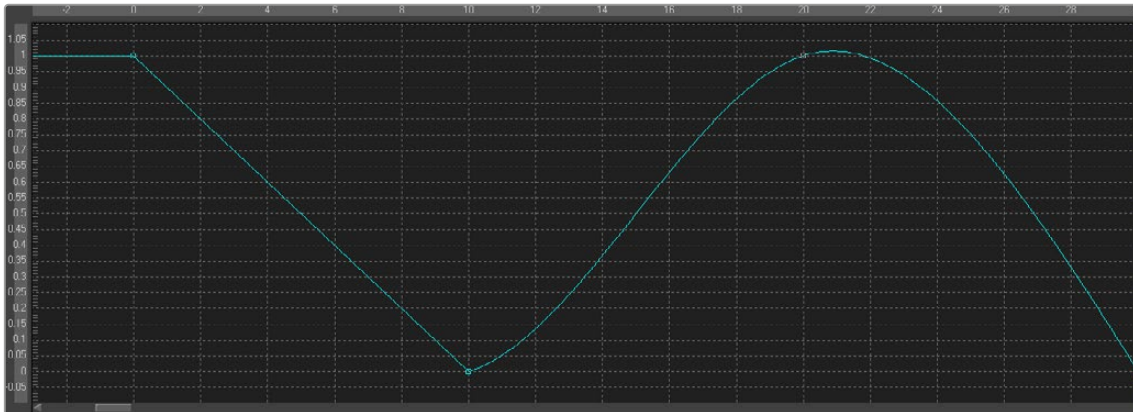
Cubic Spline Editor

Custom Poly

The Custom Poly modifier can be added to Polygon masks or paths. Similar in function to the Custom and pCustom tools, existing points can be repositioned in the polyline, or replaced completely with a new set of points. The expressions are evaluated for each point on the output polygon. The modifier can be applied by right-clicking on the "Right-click here for shape animation" text at the bottom of the Polygon controls, and selecting Insert > Custom Poly from the contextual menu.
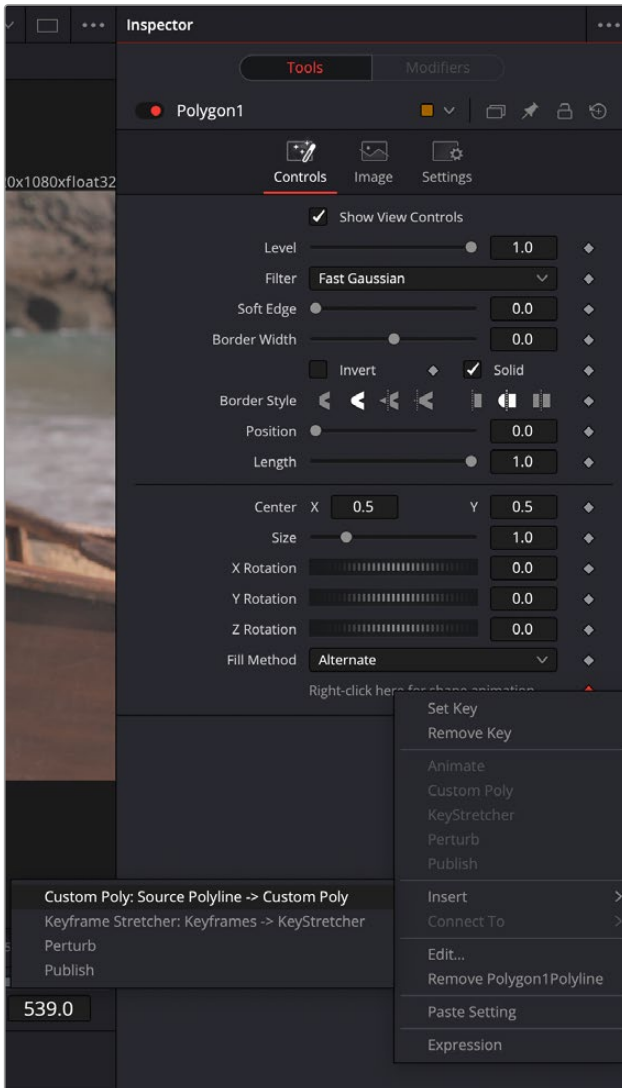
Selecting the Custom Poly modifier

Inspector

The Custom Poly Controls tab

Controls Tab

The Custom Poly Controls tab are available once you select the modifiers tab on the Polygon's inspector. It has a single point input, and number variables to use in animating the expressions. The default is one point and four numbers, but they can be expanded to a maximum of nine. Additional points and numbers can be added from the Config tab.
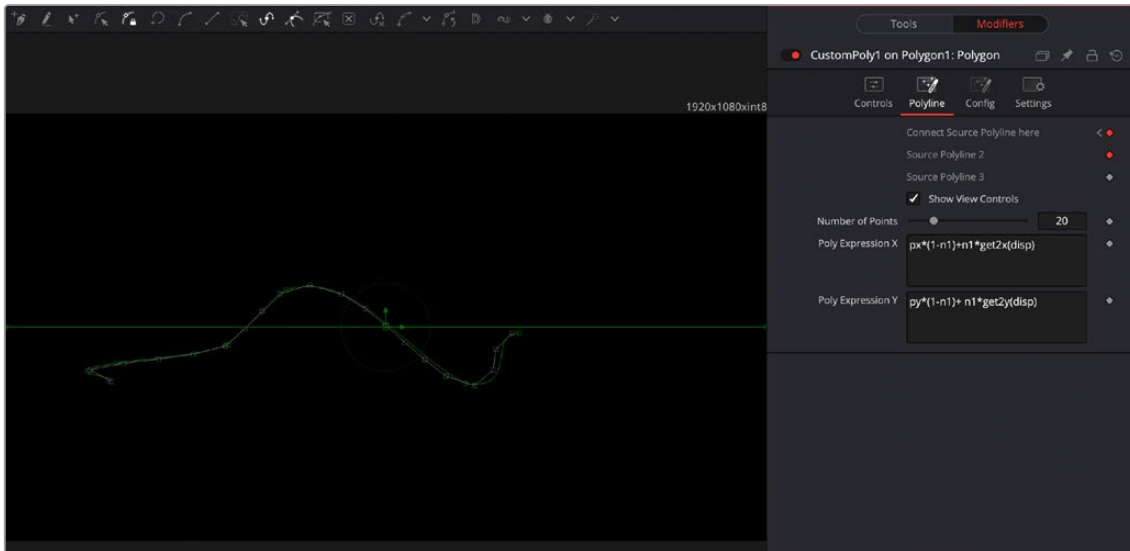
The Custom Poly Polyline tab

Polyline Tab

The polyline tab exposes the controls to connect and modify the polyline.

Connect Source Polyline here

Allows you to connect other polylines to the modifier.

Show View Controls

This checkbox toggles the polyline controls visibility in the viewer.

Number of Points

The number of output points can be set, controlling the amount of custom subdivision of the polyline. A value of zero uses the number of original source points.

Poly Expression X/Y

Fields for entering the mathematical expressions. For example, px*(1-n1)+n1*get2x(disp) and py*(1-n1)+ n1*get2y(disp).

It uses most of the same expression variables as the Expression modifier below (i.e., n1..n9, p1x..p9x, p1y..p9y, math functions, etc.), and it adds:

— px,py for the current point on the source polygon
— disp for the point's displacement on the polyline (0.0 is start, 1.0 is end)
— index for the current point's index (zero-based)
— num for the number of output points
— getx(disp), gety(disp) to get values from anywhere on the polyline
— getx_at(disp, time), gety_at(disp, time) to get values from polylines at other times
— Similarly, get2x/y(), get3x/y(), get2x/y_at(), get3x/y_at() for the second and third source polys

Config Tab

The config tab allows you to select the amount of Points and Numbers shown in the modifier (to a maximum of 9), and to give them custom names for easier organization.

Expression

An expression is a variable or a mathematical calculation added to a parameter, rather than a straight numeric value. You can add an expression to any parameter in Fusion, or you can add the Expression modifier, which adds several tabs to the modifier Inspector. Adding this modifier to a parameter adds the ability to manipulate that parameter based on any number of controls, either positional or value-based. This modifier offers exceptional flexibility compared to the more limited Calculation or Offset modifiers, but it is unable to access values from frames other than the current time.

The Expression modifier accepts nine value inputs and nine position inputs that are used as part of a user-defined mathematical expression to output a value.

To add the Expression modifier to a parameter, right-click the parameter in the Inspector and choose Modify With > Expression from the contextual menu. The type of value returned by the Expression depends entirely on the type of control it is modifying.

When used with a value control (like a slider), the Expression in the Number Out tab is evaluated to create the result. When used to modify a positional control (like Center), the Point Out tab controls the result.

The Inspector's Modifiers tab contains the controls for the Expression modifier, described below.

Inspector
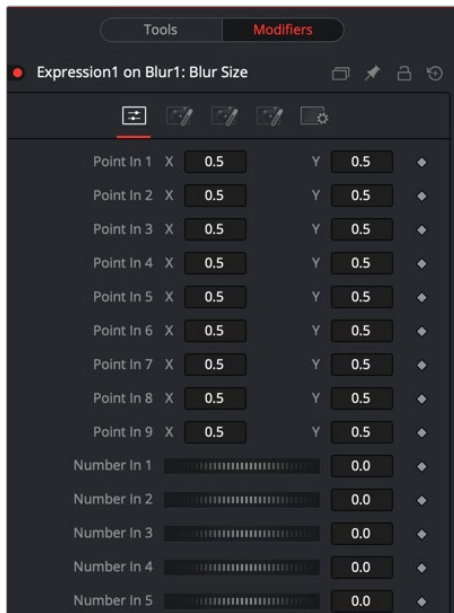
Expressions modifier controls

Controls Tab

This tab provides nine number controls and nine point controls. The values of the number controls can be referred to in an expression as n1 through n9. The X-coordinate of each point control can be referred to as p1x through p9x, while the Y-coordinate is p1y through p9y.

These values can be set manually, connected to other parameters, animated, and even connected to other Expressions or Calculations.
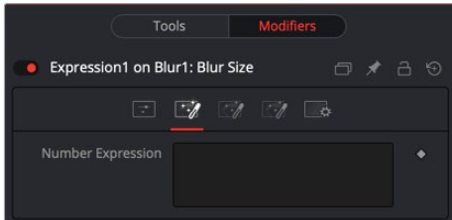

The Number Out tab

Number Out Tab

Mathematical formulas are entered using the Number In and Point In values from the Controls tab. The output modifies the control in which the Expression was applied. See below for the syntax to use in this field.
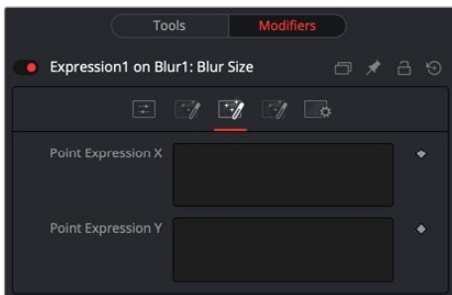
The Point Out tab

Point Out Tab

The two text boxes in this tab use mathematical formulas, accessing the Number In and Point In values from the Controls tab. The output value modifies the control in which the Expression was applied. The Expression in the top text box control is used to calculate the X-axis value, and the bottom text box is used to calculate the Y-axis control. See below for the syntax to use in this field.
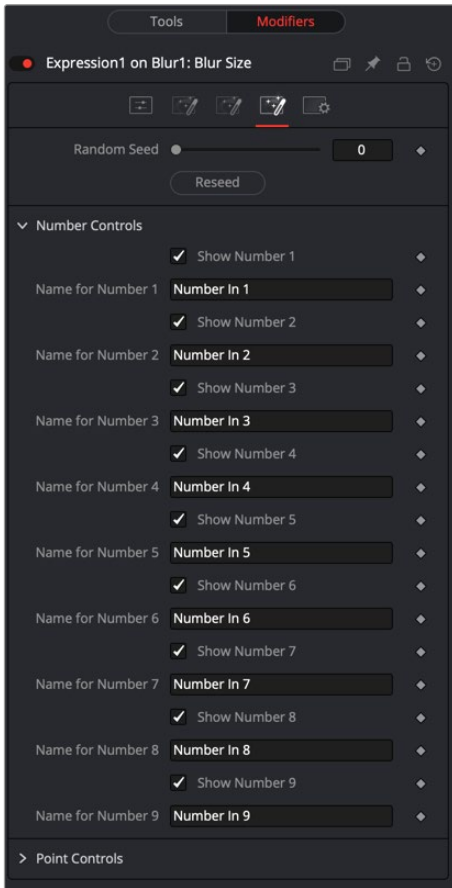
The Expressions modifier Config tab

Config Tab

A good expression is reused over and over again. As a result, it can be useful to provide more descriptive names for each parameter or control and to hide the unused ones. The Config Tab of the Expressions modifier can customize the visibility and name for each of the nine point and number controls.

Random Seed

The Random Seed control sets the starting number for the Rand() function. The rand(x, y) function produces a random value between X and Y, producing a new value for every frame. As long as the setting of this Random Seed slider remains the same, the values produced at frame x are always the same. Adjust the Seed slider to a new value to get a different value for that frame.

Show Number or Point X

There are eighteen of these checkbox controls, one for each of the nine Number and nine Point inputs. Enable this checkbox to display the control for Number x or Point x in the Controls tab.

Name for Number or Point X

There are eighteen of these edit controls, one for each of the nine Number and nine Point inputs. Type a new name for the input into this edit control to assign a new name for the Input's label in the Controls tab.

Expression Syntax Formulas

Formulas are entered into the Number Out or Point Out tabs as part of an expression. They can be made up of the following functions:

n1...n9	The value of Number Input 1..9.
p1x...p9x	The X of Positional Control 1..9.
p1y...p9y	The Y of Positional Control 1..9.
time	The current time (frame number).
pi	The value of pi.
e	The value of e.
log(x)	The base-10 log of x.
ln(x)	The natural (base-e) log of x.
sin(x)	The sine of x (x is degrees).
cos(x)	The cosine of x (x is degrees).
tan(x)	The tangent of x (x is degrees).
asin(x)	The arcsine of x, in degrees.
acos(x)	The arccosine of x, in degrees.
atan(x)	The arctangent of x, in degrees.
atan2(x, y)	The arctangent of x,y, in degrees.
abs(x)	The absolute (positive) value of x.
int(x)	The integer (whole) value of x.
frac(x)	The fractional value of x.
sqrt(x)	The Square Root of x.
rand(x, y)	A random value between x and y.
rands(x, y, s)	A random value between x and y, based on seed s.
min(x, y)	The minimum (lowest) of x and y.
max(x, y)	The maximum (highest) of x and y.
dist(x1, y1, x2, y2)	The distance between point x1,y2 and x2,y2.
dist3d(x1,y1,z1,x2,y2,z2)	The distance between 3D points x1,y2,z1 and x2,y2,z2
noise(x)	A smoothly varying Perlin noise value based on x
noise2(x, y)	A smoothly varying Perlin noise value based on x and y
noise3(x, y, z)	A smoothly varying Perlin noise value based on x, y and z
if(c, x, y)	Returns x if c not 0, otherwise y.

Expression Syntax Operators

Operators are used to evaluate statements. They are combined with functions to perform logical and mathematical calculations in the Number Out and Point Out tabs.

x + y	x plus y.
x - y	x minus y.
x > = y	1.0 if x is less than y, otherwise 0.0.
x > y	1.0 if x is greater than y, otherwise 0.0.
!x	1.0 if x = 0, otherwise 0.0.
-x	(0.0 - x).
+x	(0.0 + x) (effectively does nothing).
x ^ y	x raised to the power of y.
x y	x multiplied by y.
x y	x divided by y.
x % y	x modulo y, (remainder of (x divided by y)).
x > = y	1.0 if x is less than or equal to y, otherwise 0.0.
x > = y	1.0 if x is greater than or equal to y, otherwise 0.0.
x = y	1.0 if x is exactly equal to y, otherwise 0.0.
x == y	1.0 if x is exactly equal to y, otherwise 0.0 (identical to above).
x < > y	1.0 if x is not equal to y, otherwise 0.0.
x != y	1.0 if x is not equal to y, otherwise 0.0 (identical to above).
x & y	1.0 if both x and y are not 0.0, otherwise 0.0.
x && y	1.0 if both x and y are not 0.0, otherwise 0.0 (identical to above).
x | y	1.0 if either x or y (or both) are not 0.0, otherwise 0.0.
x || y	1.0 if either x or y (or both) are not 0.0, otherwise 0.0 (identical to above).

EXAMPLE 1

To make a numeric control equal to the Y value of a motion path, add an expression to the desired target control and connect the Path to Point In 1. Enter the formula:

`p1y`

into the Number Out field.

From Image

The From Image modifier only works on gradients, like the gradient on a Background node. It takes samples of an image along a user-definable line and creates a gradient from those samples.

Unlike other modifiers, From Image is not located in the Modify With menu. This modifier can be applied by right-clicking a Gradient bar in the Inspector and selecting From Image.

Inspector
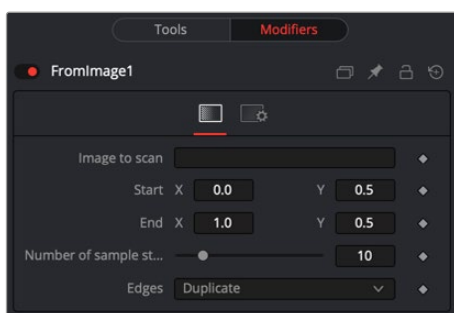
From Image modifier

Controls Tab

The From Image controls tab in the Inspector is used to select the node that contains the image you want to sample. It allows you to define the starting and ending point in the image as well as how many color samples to use in creating the gradient.

Image to Scan

Drop into this box the node from the Node Editor that you want to be color sampled.

Start X/Y, End X/Y

These two point controls define the Start and End points of the line along which the samples are taken from the image defined in the Image to Scan box.

The points can also be moved directly in the viewer.

Number of Sample Steps

This defines how many individual color samples are taken along the line. You can also see the result of this setting when you look at the actual node's Gradient control. The more sample steps you define here, the more individual color stops appear on the Gradient control. It is also possible to first create a gradient using the From Image modifier and then remove that modifier from the Gradient control again. The created gradient stays intact and can then be fine tuned by hand.

Edges

Edges determines how the edges of the image are treated when the sample line extends over the actual frame of the image to be sampled.

Black

This outputs black for every point on the sample line outside of the image bounds.

Wrap

This wraps the edges of the line around the borders of the image.
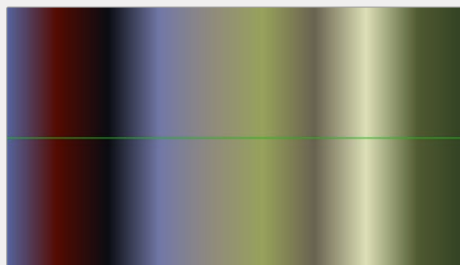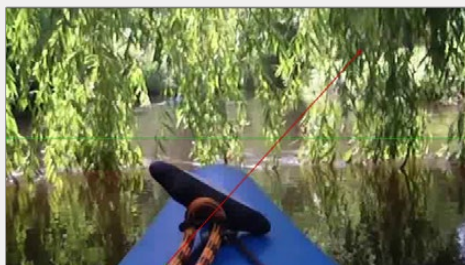
Duplicate

This causes the edges of the image to be duplicated as best as possible, continuing the image beyond its original size.

Color

This outputs a user-definable color instead of black for every point on the sample line outside of the image bounds.


EXAMPLE

The source image on the left shows the color selection line in red. The image on the right shows the resulting gradient from that selection.

Gradient Color

The Gradient Color modifier allows you to create a customized gradient and map it into a specific time range to control a value. Use the Start and End time controls to set the frames for the animation. If the Start and End time values are set to 0, then the modifier returns the value at the starting point of the gradient. You can use the Offset control to animate the gradient manually.

It can be applied by right-clicking a parameter and selecting Modify With > Gradient Color.

Inspector
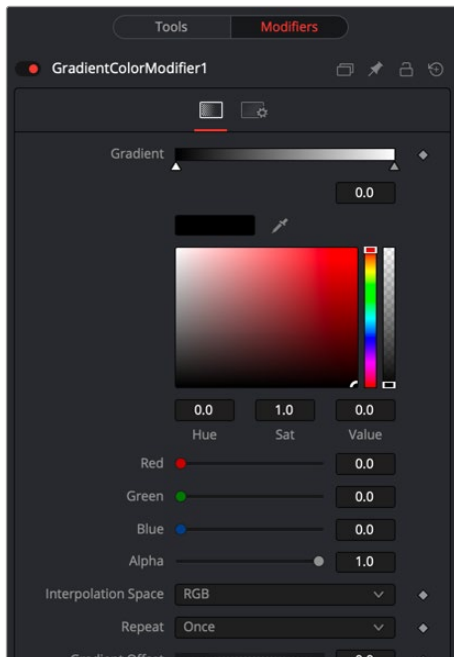
The Gradient modifier Controls tab

Controls Tab

The Controls tab consists of a Gradient bar where you add and adjust points of the gradient. Start Time and End Time thumbwheels at the bottom of the Inspector determine the time range the gradient is mapped into.

Gradient

The Gradient control consists of a bar where it is possible to add, modify, and remove points of the gradient. Each point has its color. It is possible to animate the color as well as the position of the point. Furthermore, a From Image modifier can be applied to the gradient to evaluate it from an image.

Gradient Interpolation Method

The gradient is linearly interpolated from point to point in RGB color space by default. This can sometimes result in unwanted colors. Choosing another color space may provide a better result.

Repeat

Defines how the left and right borders of the gradient are treated.

Gradients set to Once, Repeat, and Ping Pong from top to bottom, respectively, and shifting the gradient to the left.

— **Once:** When using the Gradient Offset control to shift the gradient, the border colors keep their values. Shifting the default gradient to the left results in a white border on the left; shifting it to the right results in a black border on the right.
— **Repeat:** When using the Gradient Offset control to shift the gradient, the border colors are wrapped around. Shifting the default gradient to the left results in a sharp jump from white to black; shifting it to the right results in a sharp jump from black to white.
— **Ping Pong:** When using the Gradient Offset control to shift the gradient, the border colors ping-pong back and forth. Shifting the default gradient to the left results in the edge fading from white back to black; shifting it to the right results in the edge fading from black back to white.

Gradient Offset

Allows you to pan through the gradient.

Time Controls

The Start Time and End Time thumbwheels determine the time range the gradient is mapped into. This is set in frames. The same effect can be achieved by setting the Gradient to Once and animating the offset thumbwheel.

Key Stretcher Modifier

The Keyframe Stretcher modifier is primarily used when creating title templates in Fusion for use in DaVinci Resolve's Edit page or Cut page. The Keyframe Stretcher modifier is added to an animated parameter, so the keyframes in the animated parameter stretch when the template is trimmed in the Timeline. It can be applied by right-clicking a parameter and selecting Modify with > KeyStretcher.

For more information on the Keyframe Stretcher Modifier controls, see the Keyframe Stretcher Node in Chapter 49, "Miscellaneous Nodes" in the Fusion Reference Manual or Chapter 110 in the DaVinci Resolve Reference Manual.

MIDI Extractor

The MIDI Extractor modifier provides the ability to modify the value of a control using the values stored in a MIDI file. This modifier relies on some knowledge of MIDI, which is beyond the scope of this manual.

The value produced by the modifier is extracted from the MIDI event selected in the Mode menu. Each mode can be trimmed so that only specific messages for that event are processed—for example, only some notes are processed, while others are ignored. The value of the event can be further scaled or modified by additional factors, such as Scale, Velocity, Attack, and Decay.

It can be applied by right-clicking a parameter and selecting Modify With > MIDI Extractor.

Inspector

The MIDI Extractor modifier Controls tab

Controls Tab

The Controls tab is used to load the MIDI file, modify its timing, and determine which MIDI messages and events trigger changes in the Fusion parameter.

MIDI File

This browser control is used to specify the MIDI file that is used as the input for the modifier.

Time Scale

Time Scale is used to specify the relationship between time as the MIDI file defines it and time as Fusion defines it. A value of 1.0 plays the MIDI events at normal speed, 2.0 plays at double speed, and so on.

Time Offset

Time Offset adjusts the sync between the MIDI file's timing and Fusion's timing. If there is an unexpected delay, or if the MIDI file should start partway into or before some animation in Fusion, this control can be used to offset the MIDI data as required.

Result Offset, Result Scale

These sliders adjust the range of values produced by the modifier. By default, values between 0 and 1 (or -1 and 1 for PitchBend mode) are generated. This does not always suit the node/parameter, and scale can be used to make this range larger (such as * 0.0 - 2.0). Offset is used to provide some constant value as a base.

Result Curve

The Result Curve can also be used to adjust the output. However, this adjusts the curve of the result. By default, for any input MIDI data, the results fall linearly between 0.1 and 1.0 (for example, a velocity 127 note generates 1.0, whereas 63 generates approximately 0.5).

The Result Curve applies a gamma-like curve so that middle values can produce higher or lower results while still maintaining the full scale.

Mode

This menu provides Beat, Note, Control Change, Poly AfterTouch, Channel AfterTouch, or Pitch Bend, indicating from which MIDI event the values are being read. Beat mode is slightly different in that it produces regular pulses based on the tempo of the MIDI file (including any tempo maps).

The Beat mode does not use any specific messages; it bases its event timing on the tempo map contained in the MIDI file.

Combine Events

This menu selects what happens when multiple events occur at the same time. In Notes mode, this can happen easily. For other events, this can happen if Multiple Channels are selected.

Use this to take the result from the most recent event to occur, the oldest event still happening, the highest or lowest valued event, the average, sum, or the median of all events currently occurring.

Beat (Quarters) (Beat Mode Only)

This defines how often a beat occurs when in Beat mode. This is in quarter notes, so a value of 1.0 gives a beat every quarter.

Note Range (Note and Poly Aftertouch Modes Only)

This defines what range of notes causes a value to be generated. For example, use this to pick out the kick drum from a GM drum track by setting the note range between 35–36.

Pitch Scale (Note Mode Only)

Pitch Scale defines how much the result changes with pitch. A value of 1.0 causes the result to vary from 0.0 to 1.0 over the entire range.

Velocity Scale (Note Mode Only)

This defines how much the result changes with velocity. A value of 1.0 causes the result to vary from 0.0 to 1.0 over the entire range. This is added to the result from Pitch Scale for the final result.

Control Number (Control Change Mode Only)

This specifies the MIDI controller number from which to extract events.

Envelope Controls (Note and Beat Modes Only)

These define an Envelope to follow for values before, during, and after the note or beat. Pre-Attack Time defines how long before the event it starts ramping up to the pre-attack level. Attack is the Time/Level to ramp to once the event has occurred, followed by the Decay ramp and Sustain, until the event stops. This stage is for Notes only. Beats have an instantaneous duration, so it goes straight to Release. Release is the ramp-down time after the event finishes. When trying to do a Beat, set Release to some value, or there likely will not be much of a beat.

These values can be used to follow actual sounds in the MIDI sequence or just to create interesting effects. All time values used in the MIDI Extractor are in seconds.
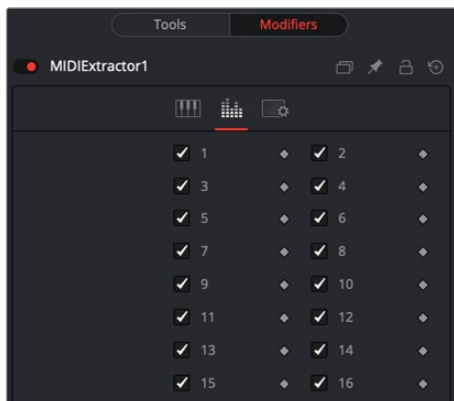
The MIDI Extractor modifier Channels tab

Channels Tab

The Channels tab is used to select the Channels used in the modifier.

Channels

Channels checkboxes select which of the 16 channels in the MIDI file are actually considered for events. This is a good way to single out a specific instrument from an arrangement.

ABOUT MIDI

A single MIDI interface allows 16 channels. Typically, these are assigned to different instruments within a device or different devices. Usually, MIDI data is 7 bits, ranging from 0–127. In Fusion, this is represented as a value between 0–1 to be more consistent with how data is handled in Fusion.

There are numerous different MIDI messages and events, but the ones that are particularly useful with this modifier are detailed below.

MIDI MESSAGES

— **Note On:** This indicates that a note (on a specific channel) is being turned on, has a pitch (0–127, with middle C being 60) and a Velocity (0–127, representing how fast the key or strings or whatever was hit).

— **Note Off:** This indicates that a note (on a specific channel) is being turned off, has a pitch (0–127, with middle C being 60) and a Velocity (0–127, representing how fast the key or strings or whatever was released).

— **Control Change:** This message indicates that some controller has changed. There are 128 controllers (0–127), each of which has data from 0–127. Controllers are used to set parameters such as Volume, Pan, amount of Reverb or Chorus, and generic things like foot controllers or breath controllers.

MIDI EVENTS

— **Channel Aftertouch:** This event defines that pressure is being applied to the keys (or strings or whatever) during a note. This represents general, overall pressure for this channel, so it simply uses a pressure value (0–127).

— **Poly Aftertouch:** This event defines that pressure is being applied to the keys (or strings or whatever) during a note. It is specific to each particular note and therefore contains a note number as well as a pressure value (0–127).

PITCH BEND

The Pitch Bend controller generally specifies the degree of pitch bending or variation applied to the note. Because pitch bend values are transmitted as a 14-bit values, this control has a range between -1 and 1 and a correspondingly finer degree of resolution.
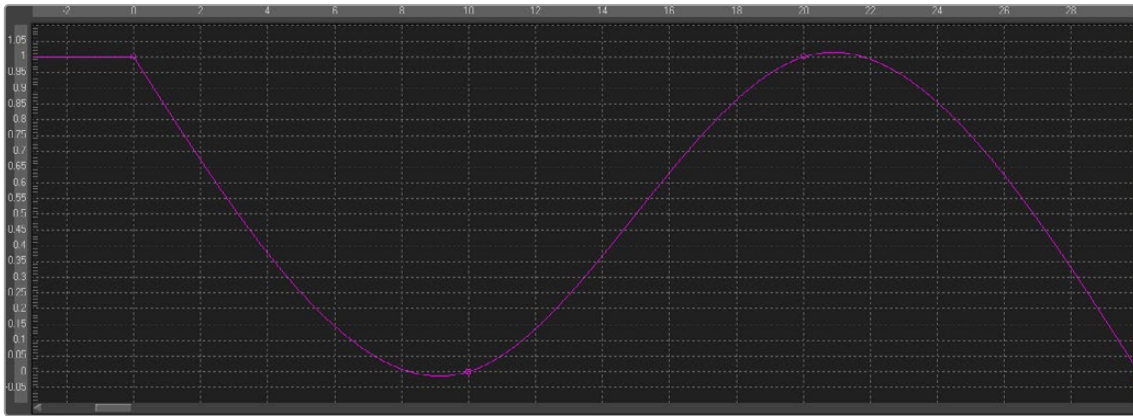
Natural Cubic Spline

The Natural Cubic Spline is one of the animation modifiers in Fusion and normally is applied to numerical values rather than point values. It can be applied by right-clicking a numerical control and selecting Modify With > Natural Cubic Spline.

NOTE: Unlike other spline types, Cubic splines have no control handles. They attempt to automatically provide a smooth curve through the control points.

Usage

Being an animation spline, this modifier has no actual Controls tab. However, its effect can be seen and influenced in the Spline Editor.

Natural Cubic Spline Editor

Offset (Angle, Distance, Position)

There are three Offset modifiers used to create variances between values. Depending on the modifier, these values relate to controls, paths, and points. The three types of Offset modifiers available in Fusion are:

— Offset Angle
— Offset Distance
— Offset Position

Offset Angle

The Offset Angle modifier outputs a value between 0 and 360 that is based on the angle between two positional controls. The Position and Offset parameters may be static, connected to other positional parameters, or connected to paths of their own. All offsets use the same set of controls, which behave differently depending on the offset type used. These controls are described below.

Offset Distance

The Offset Distance modifier outputs a value that is based on the distance between two positional controls. This modifier is capable of outputting a value based on a mathematical expression applied to a position.

Offset Position

The Offset Position modifier generates a position (X and Y coordinates) that is based on the relationship between positional controls. This modifier is the equivalent of a calculation control except that it outputs X and Y coordinates instead of a value.

It can be applied by right-clicking a control and selecting Modify With > Offset.

Inspector
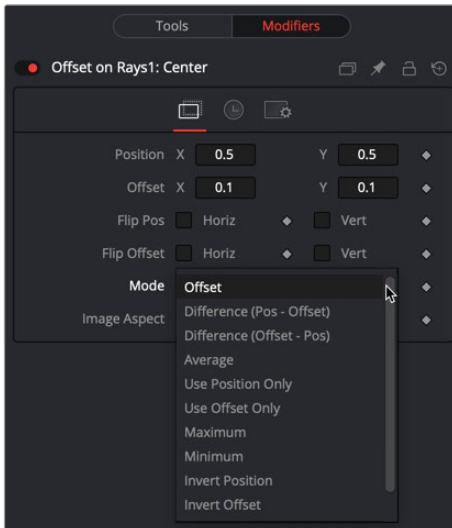
The Offset Position modifier controls

Offset Tab

The Inspector for all three Offset modifiers is identical. The Offset tab includes Position and Offset values as well as a Mode menu for selecting the mathematical operation performed by the offset control.

Position X and Y

The X and Y values are used by the Position to generate the calculation.

Offset X and Y

The X and Y values are used by the Offset to generate the calculation.

Flip Position Horizontal and Vertical

When these controls are selected, the Position is mirrored along the vertical or horizontal axis of the image.

Flip Offset Horizontal and Vertical

When these controls are selected, the Offset position is mirrored along the vertical or horizontal axis of the image.

Mode

The Mode menu includes mathematical operations performed by the Offset control. Available options include:

— Offset
— Difference (Position - Offset)
— Difference (Offset - Position)
— Average

— Use Position Only
— Use Offset Only
— Maximum
— Minimum

— Invert Position
— Invert Offset
— Invert Sugar
— Random Offset

Image Aspect

Adjust the modifier's output to compensate for the image aspect (not pixel aspect) of the project. A square image of 500 x 500 would use an Image Aspect value of 1, and a rectangular image of 500 x 1000 would use an Aspect Value of 2. The default value is always based on the current frame format selected in the preferences. To calculate image aspect, divide the width by the height. This control can also be used to create the illusion of aspect.

The Offset Time tab

Time Tab

Position Time Scale

This returns the value of the Position at the Time Scale specified (for example, 0.5 is the value at half the current frame time).

Position Time Offset

This returns the value of the Position at the Time Offset specified (for example, 10 is 10 frames back).

Offset Time Scale

This returns the value of the Offset at the Time Scale specified.

Offset Time Offset

This returns the value of the Offset at the Time Offset specified.

> **EXAMPLE**
>
> This is a simple comp to illustrate one potential use of offsets.
>
> 1. Create a new Comp 100 frames long.
> 2. Create a node tree consisting of a black background and a Text node foreground connected to a Merge.
> 3. In the Text Layout tab, use the Center X control to animate the text from the left side of the screen to the right.
> 4. Move to frame 0.
> 5. In the Text tab in the Inspector, right-click the Size control and select Modify With > Offset Distance from the contextual menu.
> 6. This adds two onscreen controls: a crosshair for the position and an X control for the offset. These onscreen controls represent the Position and Offset controls displayed in the Modifiers tab.
>   The size of the text is now determined by the distance, or offset, between the two onscreen controls.

7. Drag the X onscreen control in the viewer to see how the distance from the crosshair changes the size of the merge and by association the text.

   Both the crosshair and the X onscreen controls are animatable and can be connected to other controls.

8. Position the X centered at the bottom of the viewer.

9. In the Inspector, select the Modifiers tab.

10. In the Offset on Text size section, right-click over Position and choose Connect To > PathConnect the position value of the Offset to the existing path by right-clicking the Position control and selecting Connect To > Path1 Position.

11. Play the comp to view the animation.

12. Now, the text shrinks near the center of the path (when the distance between the offset and the path is at its minimum) and grows at its ends (where the distance between the offset and the path is at its maximum).

Path

The Path modifier uses two splines to control the animation of points: an onscreen motion path (spacial) and a Time spline visible in the Spline Editor (temporal). To animate an object's position control using a Path, right-click the Position control either in the Inspector or in the viewer and select Path from the contextual menu. This adds a keyframe at the current position. You can begin creating a path by moving the playhead and dragging the center position control in the viewer. The Spline Editor shows a displacement spline for editing the temporal value, or "acceleration," of the path.

Inspector
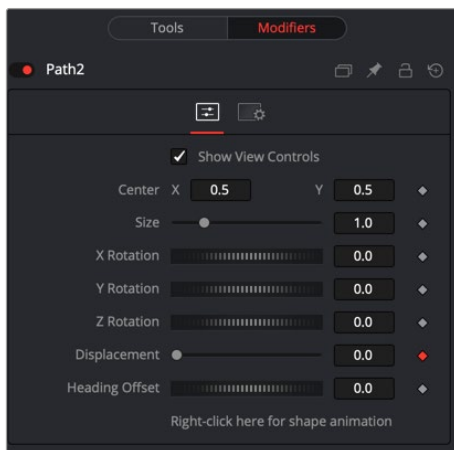
The Path modifier Controls tab

Controls Tab

The Controls tab for the path allows you to scale, reposition, and rotate the path. It also provides the Displacement parameter, allowing you to control the acceleration of an object attached to the path.

Center

The actual center of the path. This can be modified and animated as well to move the entire path around.

Size

The size of the path. Again, this allows for later modification of the animation.

X Y Z Rotation

The path can be rotated in all three dimensions to allow for sophisticated controls.

Displacement

Every motion path has an associated Displacement spline in the Spline Editor. The Displacement spline represents the position of the animated element along its path, represented as a value between 0.0 and 1.0. Displacement splines are used to control the speed or acceleration of an object's movement along the path.

To slow down, speed up, stop, or even reverse the motion of the control along the path, adjust the values of the points for the path's displacement in the Spline Editor or in the Inspector.

— A Displacement value of 0.0 in the Spline Editor indicates that the control is at the very beginning of a path.

— A value of 1.0 indicates that the control is positioned at the end of the path.

— Each locked point on the motion path in the viewer has an associated point on the Displacement spline.

— Unlocked points have a control point in the viewer but do not have a corresponding point on the Displacement spline.

Heading Offset

Connecting to the Heading adjusts the auto orientation of the object along the path. For instance, if a mask's angle is connected to the path's heading, the mask's angle will adjust to follow the angle of the path.

Right-Click Here for Shape Animation

It's possible to animate the shape of the path as well or to connect it to other path controls like Polyline Masks or Paint Strokes.

> **TIP:** Switching Default Paths
>
> You can change the default path type used when animating a position or center control to a path (if this is the preferred type of animation). Open the Preferences window and select the Global Settings. In the Default category, select the Point With menu and choose Path. The next time Animate is selected from a Position or Center control's contextual menu, a path is used.

Perturb

The Perturb modifier generates smoothly varying random values for a given parameter based on Perlin noise. It can be used to add jitter, shake, or wobble to any animatable parameter, even if the parameter is already animated. Its results are similar to those of the Shake modifier, although it uses a different set of controls that may be more intuitive. Unlike other random modifiers, you can apply the Perturb modifier to polylines, shapes, grid meshes, and even color gradients.

For example, to add camera shake to an existing path, right-click the crosshair and choose Insert > Perturb, and then adjust the Strength down to suit. Alternatively, right-clicking the path's "Right-click here for shape animation" label at the bottom of the Inspector lets you apply perturb to the path's polyline. This works best if the polyline has many points—for example, if it has been tracked or hand-drawn with the Draw Append pencil tool. A third usage option is to use the Insert contextual menu to insert the modifier onto the Displacement control. This causes the motion along the path to jitter back and forth without actually leaving the path.

> **NOTE:** Perturb can only add jitter; it cannot smooth out existing animation curves.

Inspector

The Perturb modifier Controls tab

Controls Tab

The Controls tab for Perturb is mainly used for controlling the Strength, Wobble, and Speed parameters of the random jitter.

Value

The content of this control depends on what type of control the modifier was applied to. If the Perturb modifier was added to a basic Slider control, the Value is a slider. If it was added to a Gradient control, then a Gradient control is displayed here. Use the control to set the default, or center value, for the Perturb modifier to work on.
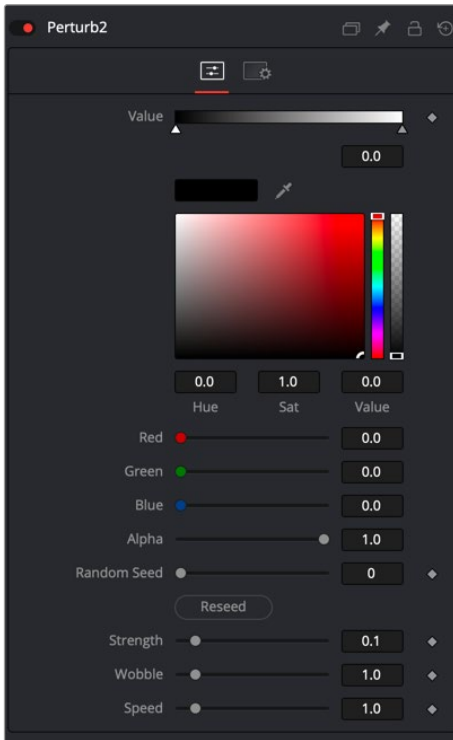
The Perturb modifier Gradient controls

Jaggedness

(Polylines and meshes only) This allows you to increase the amount of variation along the length of the polyline or mesh, rather than over time. Increasing Jaggedness gives a squigglier polyline or more tangled mesh, independent of its movement.

Phase

(Polylines and meshes only) Animating this can be used to move the ripple of a polyline or mesh along itself, from end to end. The effect can be most clearly seen when Speed is set to 0.0.

Random Seed Randomize

The Random Seed is used to "seed" the amount of jitter applied by the modifier. Two Perturb modifiers with identical settings, but different random seeds, produce two completely different results. Click the Randomize button to assign a random seed value.

Strength

Use this control to adjust the strength of the Perturb modifier's output, or its maximum variation from the primary value specified above.

Wobble

Use the Wobble control to determine how smooth the resulting values are. Less wobble implies a smoother transition between values, while more wobble produces less predictable results.

Speed

Increasing the Speed slider value speeds up the rate at which the value changes. This can increase the apparent wobbliness in a more predictable fashion than the Wobble control and make the jitter more frantic or languorous in nature.
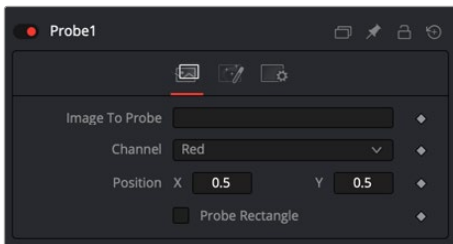
Probe

The Probe modifier is one of the most versatile modifiers in Fusion. It allows you to control any numeric parameter by the color or luminosity of a specific pixel or rectangular region of an image. Think of driving the Brightness node by probing the pixel values of flickering lights in a shot, or measuring graded LUTs to compare values.

It can be applied by right-clicking a parameter and selecting Modify With > Probe.

Inspector

The Probe modifier Controls tab

Controls Tab

The Controls tab for the Probe modifier allows you to select the node to probe, define the channel used to drive the parameter, and control the size of the probed area.

Image to Probe

Drag a node from the Node Editor to populate this field and identify the image to probe.

Channel

Select the channel you want to probe. The usual options are:

— Red
— Green
— Blue
— Alpha

Luma

Once a Probe modifier is present somewhere in your comp, you can connect other node's values to its outputs as well. The Probe allows you to connect to its values individually:

— Result
— Red
— Green
— Blue
— Alpha

Position X Y

The position in the image from where the probe samples the values.

Probe Rectangle

By default, the Probe samples only the value of a single pixel at its position. By using the Probe Rectangle mode, you can sample from a larger area of pixels based on the Evaluation method.

Width Height Controls

These determine the size of the area to be probed.

Evaluation

Sets how the pixels inside the rectangle are computed to generate the output value.

Options include:

— **Average:** All pixel values inside the rectangle are averaged.

— **Minimum:** The smallest value of all pixels inside the rectangle is used.

— **Maximum:** The highest value of all pixels inside the rectangle is used.
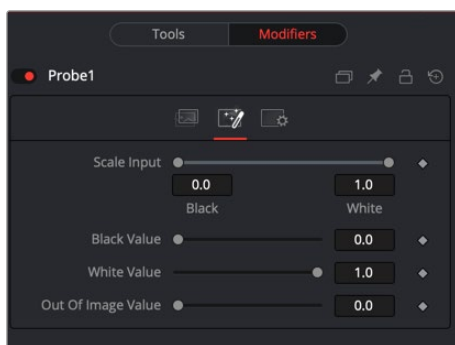
The Probe modifier Value tab

Value Tab

The Value tab controls the range or scale of the modifier adjustment, thereby adjusting the sensitivity of the Probe.

Scale Input

By default, the Probe generates the Black Value when the probed area results in a value of 0 (i.e., black), and it generates its White Value when the probed area results in a value of 1 (i.e., white). By using this range control, you can modify the sensitivity of the Probe.

Black Value

The value that is generated by the Probe if the probed area delivers the result set in Scale Input Black.

White Value

The value that is generated by the Probe if the probed area delivers the result set in Scale Input White.

Out of Image Value

The value that is generated by the Probe if the probed area is outside the frame boundaries of the probed image. If probing a rectangle, this value does not generate before the entire rectangle is outside the frame boundaries of the image to be probed.

Publish

Only parameters that are animated will be available from the Connect To menu. To connect to non-animated parameters, you must Publish them first. Animated controls are automatically published, whereas static controls must be published manually.

To publish a static control, right-click the control and select Publish from the contextual menu.
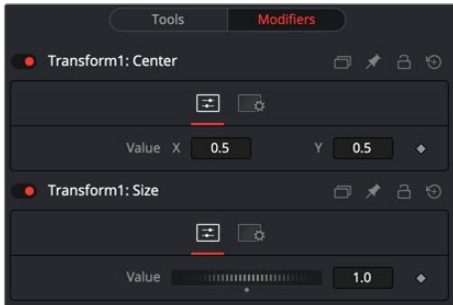
The Publish modifier Controls tab

Controls Tab

The Controls tab shows the published control available for linking to other controls.

Published Value

The display of the published control is obviously dependent on which control is published from which node.

Resolve Parameter

The Resolve Parameter Modifier is used when creating a transition template in Fusion for use in DaVinci Resolve's Edit page or Cut page. When building a transition in Fusion, the Resolve Parameter modifier is added to any control you want to animate. The Resolve Parameter modifier automatically animates the parameter for the duration of the transition, allowing you to trim the transition in the Edit page or Cut page.

For example, to create a cross dissolve, do the following:

1   From the Effects Library, add a Fusion Composition to the Edit page Timeline.

2   In Fusion, add a Dissolve node to the Node Editor.

3   In the Inspector, right-click the Background/Foreground parameter, and then choose Resolve Parameter from the Modifier contextual menu. Adding the modifier to the Background/Foreground parameter automatically updates the slider if the transition is modified back on the Edit/Cut page.

4   In the Node Editor, right-click the Dissolve node and choose Macro > Create Macro.

5   When creating a macro that's to be used as a Fusion transition it's important that two inputs and one output are selected in the Macro Editor. In this example, under the Dissolve heading, enable the Output, Background and Foreground check boxes.

6   Give the transition a name, then save the macro from the top File menu.

7 Save the macro in the following folder so it appears the Edit page or Cut page Effects Library:

— **For MacOS:** $TEMPLATE_MAC_OS_PATH/Transitions or $TEMPLATE_MAC_USER_PATH/Transitions

— **For Windows:** $TEMPLATE_WIN_OS_PATH\Transitions or $TEMPLATE_WIN_USER_PATH\Transitions

8 Quit and reopen DaVinci Resolve to update the list of transitions in the Effects Library.

9 On the Edit page, open the Effects Library. Navigate to Video Transitions Fusion Transitions, and the custom Fusion transition will be listed.

Shake

The Shake modifier is used to randomize a Position or Value control to create semi-random numeric inputs. The resulting shake can be entirely random. The motion can also be smoothed for a more gentle, organic feel.

To add the Shake modifier to a parameter, select Modify With > Shake from the parameter's contextual menu. The Shake modifier uses the following controls to achieve its effect. It can be applied by right-clicking a parameter and selecting Modify With > Shake.

Inspector
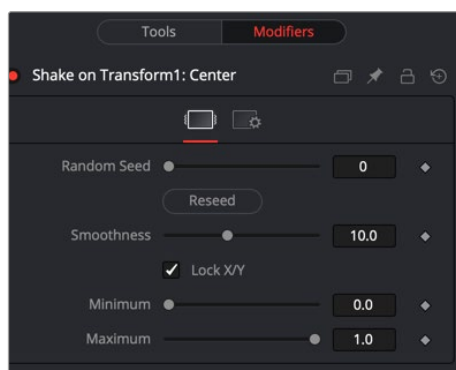
The Shake modifier Controls tab

Controls Tab

Random Seed

The Random Seed control contains the value used to seed the random number generator. Given the same seed, a random number generator always produces the same results. Change the seed if the results from the randomizer are not satisfying.

Smoothness

This control is used to smooth the overall randomness of the Shake. The higher the value, the smoother the motion appears. A value of zero generates completely random results, with no smoothing applied.

Lock X/Y

This checkbox is used to unlock the X- and Y-axis, revealing independent slider controls for each axis.

Minimum and Maximum

This control is used to determine the overall strength of the shake. The low values represent the lowest value that can be generated by the randomizer, and the high values represent the highest values. To create a Shake that moves a center crosshair anywhere within the image, set the Minimum to 0.0 and the Maximum to 1.0. To restrict the motion to a smaller shake in the bottom-right corner of the image, set the Minimum to 0.70 and the Maximum to 0.90.

> **EXAMPLE**
>
> 1. Create a new comp, and then add and view a Text node.
> 2. Type some text in the Text node.
> 3. In the viewer, right-click over the Center control of the text and choose Modify With > Shake Position.
> 4. In the Inspector, select the Modifiers tab and set the smoothing to 5.0.
> 5. Set the Minimum to 0.1 and the Maximum to 0.9.
>
>   This adds some chaotic movement to the text. However, we can change this over Go to frame 0 and in the Inspector click the Keyframe button to the right of both the Minimum and the Maximum controls.
> 6. Go to frame 0 and in the Inspector click the Keyframe button to the right of both the Minimum and the Maximum controls.
> 7. Go to frame 90 and adjust the Minimum to 0.45 and the Maximum to 0.55.
> 8. View the results.
>
>   Now, the text starts out by flying all over the screen and tightens in toward the center of the screen as the comp plays.
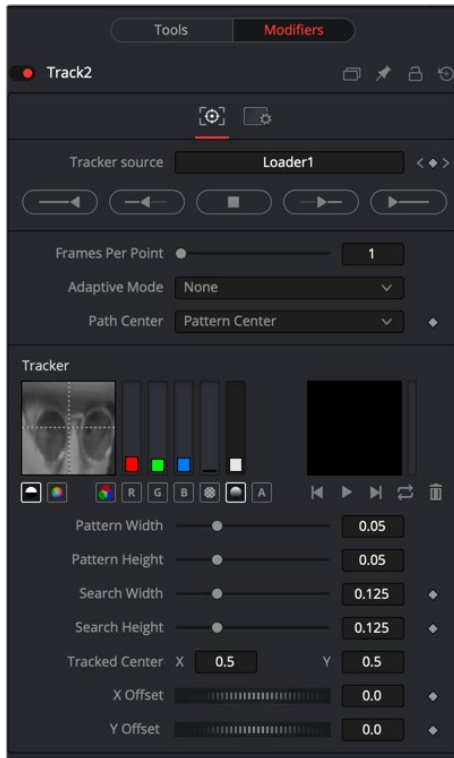
Track

Although there is a standard Tracker node, you can also use a Tracker modifier to add a tracker directly to a parameter. To apply the Tracker modifier, in the viewer right-click the Center control of of any transform, text, mask, or other positionable element. From the contextual menu, choose Object x Center > Modify With > Tracker.

Then choose one of three options:

— **Tracker Position:** Tracks a point from the source.

— **Steady Position:** Stabilizes based on a single point in the source.

— **Unsteady Position:** Adds original motion back after stabilizing.

This adds a modifier in the Inspector with a set of controls almost identical to those found in the Tracker node itself.

Inspector

The Tracker modifier

For an in-depth explanation of this node, *see Chapter 117, "Tracker Nodes,"* in the DaVinci Resolve Reference Manual, or Chapter 55 in the Fusion Reference Manual.

Tracker Modifier vs. Tracker Node

The differences between a Tracker modifier and a Tracker node are as follows:

— The Tracker modifier can only track a single pattern.

— The Tracker modifier can only output a single value and cannot be used for complex stabilization or match-moving procedures.

— The default source image for the modifier is the node immediately upstream of the node that contains the modifier (i.e., when adding a Tracker modifier to a Glow node with a Loader as its input, the Tracker Source input defaults to the output of the Loader). Set a different source image for the Tracker modifier by typing in the name of the node. Alternatively, drag the source node from the Node Editor into the Text Box control.

Vector Result

The Vector Result modifier is used to offset positional controls, such as crosshairs, by distance and angle. These can be static or animated values.

It can be applied by right-clicking a control and selecting Modify With > Vector.

Inspector
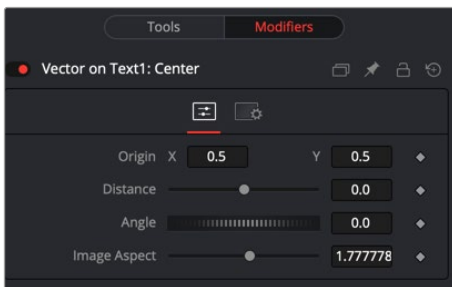
The Vector Result modifier Controls tab

Controls Tab

Origin

This control is used to represent the position from which the vector's distance and angle values originate.

Distance

This slider control is used to determine the distance of the vector from the origin.

Angle

This thumbwheel control is used to determine the angle of the vector relative to the origin.

Image Aspect

This slider control is used primarily to compensate for image aspect differences. A square image of 500 x 500 would use an Image Aspect value of 1, while a rectangular image of 500 x 1000 would use an Image Aspect value of 2. The default for this value is taken from the current Frame Format preferences using width/height. It may be necessary to modify this control to match the current image.

EXAMPLE

1. Create a 100-frame comp.

2. Create a simple node tree consisting of a black background and a Text node foreground connected to a Merge.

3. Enter some text in the Text node.

4. Select the Merge node.

5. In the viewer, right-click the Center control of the Merge and choose Modify With > Vector Result.

   This adds a crosshair onscreen control for the Vector distance and angle. The onscreen control represents the Distance and Angle controls displayed in the Modifiers tab.

6. In the Modifiers tab of the Inspector, drag the Distance control to distance the text from the Vector origin.

7. Drag the Angle thumbwheel to rotate the text around the Vector origin.

8. This is different from changing a pivot point, since the text itself is not rotating.

   These points are animatable and can be connected to other controls.

9. In the Inspector, right-click the Origin control and choose a path to add a motion path modifier to the Origin control.

10. Verify that the current frame is set to frame 0 (zero) and use the Origin controls in the Inspector or drag the Vector Origin crosshair to the bottom-left corner of the screen.

11. On the Vector Angle thumbwheel, click the Keyframe button to animate this control.

12. Set the Angle thumbwheel to a value of 10.

13. Go to frame 100 and click at the top-left corner of the screen to move the Vector Origin crosshair.

14. Set the Vector Angle thumbwheel to a value of 1000.

15. Play the comp to see the results.

   This causes the text to orbit around the path just created.

XY Path

The XY Path type uses two separate splines for the position along the X-axis and for the position along the Y-axis.

To animate a coordinate control using an XY path, right-click the control and select Modify With > XY Path from the contextual menu.

At first glance, XY paths work like Displacement paths. To describe the path, change frames and position the control where it should be on that frame, and then change frames again and move the control to its new position. Fusion automatically interpolates between the points. The difference is that no keyframes are created on the onscreen path.

Look in the Spline Editor to find the X and Y channel splines. Changes to the control's values are keyframed on these splines. The advantage to the XY path is that it becomes very easy to work with motion along an individual axis.

Inspector
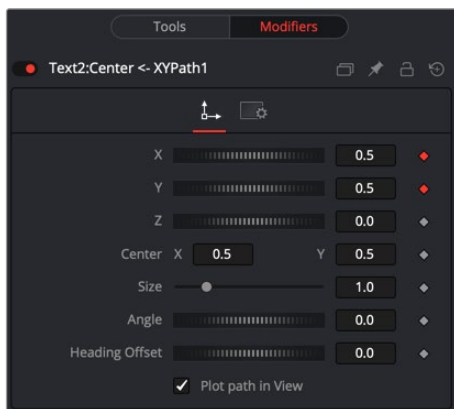
XY Path modifier controls

X Y Z Values

These reflect the position of the animated control using X, Y, and Z values.

Center

The actual center of the path. This can be modified and animated as well to move the entire path around.

Size

The size of the path. Again, this allows for later modification of the animation.

Angle

The angle of the path. Again, this allows for later modification of the animation.

Heading Offset

If another control (for example, a mask's Angle) is connected to the path's heading, this control allows for adding or subtracting from the calculated angle.

Plot Path in View

Toggles whether or not the actual path is displayed in the views.

Switching Default Paths

You can change the Default Path type to XY path (if this is the preferred type of animation). Open the Default category in the Global Preferences and locate the Point With drop-down menu. Change this from the current value to XY Path. The next time Animate is selected from a Coordinate control's contextual menu, an XY path is used instead of a Displacement path.