Nuke™ 11.3v6 Online Help. Copyright © 2019 The Foundry Visionmongers Ltd. All Rights Reserved. Use of this document and the Nuke software is subject to an End User License Agreement (the “EULA”), the terms of which are incorporated herein by reference. This document and the Nuke software may be used or copied only in accordance with the terms of the EULA. This document, the Nuke software and all intellectual property rights relating thereto are and shall remain the sole property of The Foundry Visionmongers Ltd. (“The Foundry”) and/or The Foundry’s licensors.

The EULA is available here: End User License Agreement (EULA)

The Foundry assumes no responsibility or liability for any errors or inaccuracies that may appear in this document and this document is subject to change without notice. The content of this document is furnished for informational use only.

Except as permitted by the EULA, no part of this document may be reproduced, stored in a retrieval system or transmitted, in any form or by any means, electronic, mechanical, recording or otherwise, without the prior written permission of The Foundry. To the extent that the EULA authorizes the making of copies of this Reference Guide, such copies shall be reproduced with all copyright, trademark and other proprietary rights notices included herein. The EULA expressly prohibits any action that could adversely affect the property rights of The Foundry and/or The Foundry's licensors, including, but not limited to, the removal of the following (or any other copyright, trademark or other proprietary rights notice included herein):

Nuke™ compositing software © 2019 The Foundry Visionmongers Ltd. All Rights Reserved.

Nuke™ is a trademark of The Foundry Visionmongers Ltd.

Digital Domain ® is a registered trademark of Digital Domain, Inc.

Primatte™ keyer tool © 1997-2019 Photron USA, Inc. All Rights Reserved.

Primatte™ is a trademark of IMAGICA Corp.

Primatte™ patent is held by IMAGICA Corp.

In addition to those names set forth on this page, the names of other actual companies and products mentioned in this User Guide (including, but not limited to, those set forth below) may be the trademarks or service marks, or registered trademarks or service marks, of their respective owners in the United States and/or other countries. No association with any company or product is intended or inferred by the mention of its name in this User Guide.

ACADEMY AWARD ® is a registered service mark of the Academy of Motion Picture Arts and Sciences.

Linux ® is a registered trademark of Linus Torvalds.

Windows ® is the registered trademark of Microsoft Corporation.

Mac, Mac OS X, macOS, Sierra, High Sierra, Mojave, Shake, Final Cut Pro, and QuickTime are trademarks of Apple, Inc., registered in the U.S. and other countries.

Adobe ® and Photoshop ® are either registered trademarks or trademarks of Adobe Systems Incorporated in the United States and/or other countries.

Maya ® is a registered trademark of Autodesk, Inc., in the USA and other countries.

Houdini ® is a registered trademark of Side Effects Software, Inc.

Boujou is a trademark of 2d3 Ltd.

3D-Equalizer is a trademark of Science.D.Visions.

RenderMan ® is a registered trademark of Pixar.

Cineon™ is a trademark of Eastman Kodak Company.

Thank you to Diogo Girondi for providing icons for the Nuke user interface and Tim Baier for proofreading.
## Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Getting Started</td>
<td>80</td>
</tr>
<tr>
<td>Meet the Nuke Product Family</td>
<td>81</td>
</tr>
<tr>
<td>Nuke Products</td>
<td>81</td>
</tr>
<tr>
<td>About Nuke</td>
<td>82</td>
</tr>
<tr>
<td>About NukeX</td>
<td>83</td>
</tr>
<tr>
<td>About Nuke Assist</td>
<td>84</td>
</tr>
<tr>
<td>Using Gizmos, Groups, Precomps, Knobs, and Python</td>
<td>86</td>
</tr>
<tr>
<td>About Nuke Non-commercial</td>
<td>87</td>
</tr>
<tr>
<td>Key Concepts</td>
<td>88</td>
</tr>
<tr>
<td>Understanding the Workflow</td>
<td>88</td>
</tr>
<tr>
<td>Working with Multiple Image Formats</td>
<td>89</td>
</tr>
<tr>
<td>Channel Operations</td>
<td>90</td>
</tr>
<tr>
<td>8-, 16-, and 32-Bit Image Processing</td>
<td>91</td>
</tr>
<tr>
<td>Compositing in 3D</td>
<td>91</td>
</tr>
<tr>
<td>Render Farms and Frame Servers</td>
<td>92</td>
</tr>
<tr>
<td>Installation and Licensing</td>
<td>93</td>
</tr>
<tr>
<td>Operating Systems</td>
<td>93</td>
</tr>
<tr>
<td>Windows</td>
<td>94</td>
</tr>
<tr>
<td>System Requirements</td>
<td>94</td>
</tr>
<tr>
<td>Minimum Hardware Requirements</td>
<td>94</td>
</tr>
<tr>
<td>Requirements for GPU Acceleration</td>
<td>95</td>
</tr>
<tr>
<td>NVIDIA</td>
<td>95</td>
</tr>
<tr>
<td>AMD</td>
<td>96</td>
</tr>
<tr>
<td>Multi-GPU Processing</td>
<td>96</td>
</tr>
</tbody>
</table>
Editing Nodes 158
Cloning Nodes 159
Disabling and Deleting Nodes 160
Connecting Nodes 160
Node Indicators 163
Displaying Node Information 164
Customizing the Node Display 165
Creating Node Tool Sets 167

Using the Tab Menu 168
Adding Nodes by Sub-string 168
Adding and Removing Favorites 170
Managing Weighting 171

Navigating Inside the Node Graph 173
Panning 173
Zooming, Fitting in the Node Graph 174
Bookmarking the Pan and Zoom Level 175
Bookmarking Nodes 175
Searching for Nodes 176
Cleaning up the Node Graph 177

Properties Panels 178
Managing the Properties Panel 178
Properties Panel Controls 179
Displaying Controls 182
Using Input Fields 183

Using the Color Controls 185
Using the Color Swatch 185
Using the In-Panel Color Picker 186
Using the Floating Color Picker Window 187
Using the Channel Selector
Color Slider Functions
Customizing a Node’s Defaults with Node Presets
To Create a Node Preset:
Loading and Deleting a Preset
Animating Parameters
Working with Animated Parameters
Animated Parameters in the Dope Sheet and the Curve Editor
Using the Dope Sheet
Using the Dope Sheet Interface
Viewing Keys in the Dope Sheet
The Dope Sheet and Time Nodes
Multiple Branches of Nodes in the Dope Sheet
Selecting Keyframes
Copying and Pasting
Deleting and Adding Keyframes
Editing Clips in the Dope Sheet
Synchronizing the Frame Range
Using the Curve Editor
Editing Curves
Compositing Viewers
Adding Viewer Nodes
Connecting Viewer Nodes
Toggling Views
Panning and Zooming the Viewer Window
Panning the Frame
Re-centering the Frame
Zooming In and Out
Restoring the Zoom to 100%
Showing / Hiding Floating Viewers
Showing / Hiding Viewer Toolbars
Locking the Viewer Zoom Level
Viewing Overscan in the Viewer 219

Using the Viewer Controls 220
Compositing Viewer Controls 220
Jumping to a Specific Frame 222
Synchronizing Viewer Playback 223
Pausing the Display 223
Displaying a Single Channel 223
Layer and Channel Dropdown Menus 224
Displaying Alpha Mattes 225
Viewer Info Bar 226
Adjust Display Gain and Gamma 227
2D / 3D Toggle and Camera Controls 228
Selection Modes 228
Monitor Output Toggle 230

Controlling Zoom and Resolution 231
Using the Zoom Menu 231
Proxy Mode 231
Lowering the Display Resolution of Individual Viewers 232
Pixel Aspect Ratio 232
Full-frame processing 233
Region of Interest (ROI) 234

Viewer Overlays and Input Processes 236
Guides and Masks 236
Adding Custom Guides and Masks 240
Using the Viewer Composite Display Modes 240
Input Process and Viewer Process Controls 242

Using the File Browser 247
To Use the Navigation Controls 248
Path Name Field
To Use the Path Name Field: 248
To Use the Filter Dropdown Menu and Sequences Checkbox 248
To Preview Files in the File Browser 249
To Select Multiple Files with the File Browser 250
To Open Incomplete Sequences in Separate Read Nodes 250

Undoing and Redoing 251

Progress Bars 252

Handling Errors 253

Using Nuke Studio's Timeline Environment 254
Achieving Real-time Playback 254
Windows and Linux Certified Hardware 254

About Clips and Shots 257
Source Clips 257
Shots 258
A Source Clip Opened in the Timeline View 258

Clip and Shot Properties 259
Setting Source Clip Ranges 260
Using Relative Paths to Media 261

Ingesting Media 263

Using Drag-and-Drop 265

Using the File Browser 266
To Use the Navigation Controls 267
Path Name Field 267
To Use the Filter Dropdown Menu and Sequences Checkbox 268
To Preview Files in the File Browser 268
To Select Multiple Files with the File Browser

Sorting and Searching Media
Sorting the Project Panel
Searching for Media
Setting Poster Frames

Color-coding Source Clips and Shots
Setting Default Colors
Setting Colors by Selection
Setting Colors by File Type

Reconnecting and Refreshing Clips

Localizing Media
Setting Localization Preferences
Managing Localization
Updating On Demand Clips
Clearing Localized Files

Using the Timeline Viewer
Deleting Media

Timeline Playback Tools
In and Out Points
Playback Controls
Timeline Viewer Tools
Using In and Out Markers
Using In and Out Markers

Working with Colors
To apply colorspace changes to shots:

Previewing on a Broadcast Monitor
To preview output on an external broadcast video monitor: 312

Using Scopes 314
Histogram 314
Waveform 316
Vector 318

About Anamorphic Media 320

About QuickTime Media 321

About RED Media 323
To modify the RED Rocket options: 323

Compositing with Nuke 325
Organization of the Section 325

Managing Scripts 327
Setting Up Your Script 327
Name, Time Span, and Frame Rate 327
Setting the Default Project Directory 328

Full-Size Formats 329
To Set Up a Full-Size Format 329

Proxy Mode 330
Proxy Format and Proxy Scale 330
To Set Up Proxy Formats 331
To Set Up a Proxy Scale 332

Read Nodes and Proxy Files 332
To Define Which File (Full-Res or Proxy) is Used in the Proxy Mode: 333

Write Nodes and Proxy Files 333

Using Proxy Mode to Enlarge a Script 333
Using Small Proxy Files 334
Using Large Proxy Files 334
Rendering the Scaled-up Output 334

To Toggle Between Full Resolution and Proxy Mode 334

Loading Image Sequences 335
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Importing Image Sequences</td>
<td>335</td>
</tr>
<tr>
<td>To Import an Image Sequence from an External File Browser</td>
<td>337</td>
</tr>
<tr>
<td>Notes on Importing QuickTime Files</td>
<td>337</td>
</tr>
<tr>
<td>Notes on Importing AVI Files</td>
<td>338</td>
</tr>
<tr>
<td>Notes on Importing OpenEXR Files</td>
<td>338</td>
</tr>
<tr>
<td>Notes on Importing PSD Files</td>
<td>339</td>
</tr>
<tr>
<td>Naming Conventions</td>
<td>340</td>
</tr>
<tr>
<td>Changing the Relation Between the Current Frame and the Frame Read In</td>
<td>340</td>
</tr>
<tr>
<td>Using Expressions</td>
<td>340</td>
</tr>
<tr>
<td>Specifying a Start Frame for a Clip</td>
<td>341</td>
</tr>
<tr>
<td>Offsetting All Frames by a Constant Value</td>
<td>341</td>
</tr>
<tr>
<td>Reformatting Image Sequences</td>
<td>343</td>
</tr>
<tr>
<td>To Insert a Reformat Node:</td>
<td>343</td>
</tr>
<tr>
<td>Image Caching</td>
<td>344</td>
</tr>
<tr>
<td>The Cache Directory</td>
<td>346</td>
</tr>
<tr>
<td>Defining the Settings for Caching</td>
<td>346</td>
</tr>
<tr>
<td>Clearing the Disk Cache</td>
<td>346</td>
</tr>
<tr>
<td>To Empty the Disk Cache</td>
<td>347</td>
</tr>
<tr>
<td>Using the DiskCache Node</td>
<td>348</td>
</tr>
<tr>
<td>To Cache Images Upstream</td>
<td>348</td>
</tr>
<tr>
<td>Localizing Files for Better Performance</td>
<td>350</td>
</tr>
<tr>
<td>Setting Localization Preferences</td>
<td>350</td>
</tr>
<tr>
<td>Managing Localization</td>
<td>353</td>
</tr>
<tr>
<td>Updating On Demand Clips</td>
<td>355</td>
</tr>
<tr>
<td>Clearing Localized Files</td>
<td>356</td>
</tr>
<tr>
<td>Saving Scripts and Recovering Back-Ups</td>
<td>358</td>
</tr>
<tr>
<td>Saving Scripts</td>
<td>358</td>
</tr>
<tr>
<td>To Save a Script</td>
<td>358</td>
</tr>
</tbody>
</table>
To Create a New Layer and/or Channel 379

Calling Channels 381
Viewing Channels in the Viewer 382
Selecting Input Channels 383
   To Select a Single Input Channel 383
   To Select Multiple Input Channels 383
Selecting Masks 383
   To Select a Channel for Use as a Matte from the Mask Input 384
   To Select a Channel for Use as a Matte from the Main Input 385

Linking Channels Using the Link Menu 386

Tracing Channels 387

Renaming Channels 388

Removing Channels and Layers 389
   To Remove a Layer or a Channel Within a Layer 389

Swapping Channels 390
Channel Outputs 391
Assigning Constants 392
Creating Swap Layers 392

Merging Images 393
Layering Images Together with the Merge Node 393
   To Layer Images with the Merge Node 393
Bounding Box Warnings 395

Merge Operations 397

Generating Contact Sheets 408
   To Generate a Contact Sheet 408

Copying a Rectangle from one Image to Another 411
   To Copy a Rectangle from One Image to Another 412
Advanced Keying 440
Under the Hood 440
View 440
Status 441

View 443
Status 443

Screen Color 446
Background Pixel 446
Edge Pixel 447
Foreground pixel 447
Biasing 448
The Bias Colors in Everyday Use 449
Picking a bias color 449
Why are there two bias colors? 450

Clip Black and White 452

Screen Gain 453
Screen Balance 454

PreBlur and Tuning 455
Tuning 455

Screen Processing 456
Two-stage keying 456
Clip Rollback 456
Dilate 457
Softness 457
Despot 458

Mattes 459
Primatte Controls

Initialize Section

Primatte Viewer Tools

Degrain Section

Degrain type
Degrain Tools Tutorial

Actions Section

Smart Select BG Color
Clean BG Noise
Clean FG Noise
Matte Sponge
Make FG Trans
Restore Detail
Spill Sponge
Spill (-)
Spill (+)
Matte (-)
Matte (+)
Detail (-)
Detail (+)
Fine Tuning Sliders
3D Sample
Simple Select BG Color
Current Color Chip
Adjust Lighting
Hybrid Render

Adjust Lighting
Adjusting overlay controls  
Adjusting the screen correct controls  
**Adjusting the Density of the Matte**  
Adjusting Density controls  
**Adjusting Spill Controls**  
**Retaining Shadows and Removing Noise**  
Adjusting Shadows controls  
Adjusting Cleanup controls  
**Adjusting Color Controls**  
**Adjusting Film Controls**  
**Choosing an Output Mode**  
**Using RotoPaint**  
Roto or RotoPaint?  
RotoPaint Quick Start  
**Connecting the RotoPaint Node**  
To Connect the RotoPaint Node  
**Working with the Toolbars**  
**Working with the Stroke/Shape List**  
To Use the Stroke/Shape List  
**Drawing Paint Strokes**  
**Using the Brush tool**  
To Use the Brush Tool  
**Using the Eraser Tool**
To Use the Eraser Tool 540

Using the Clone Tool 541
To Use the Clone Tool 541

Using the Reveal Tool 543
To Use the Reveal Tool 543

Using the Blur Tool 545
To Use the Blur Tool 545

Using the Sharpen Tool 546
To Use the Sharpen Tool 546

Using the Smear Tool 547
To Use the Smear Tool 547

Using the Dodge Tool 548
To Use the Dodge Tool 548

Using the Burn Tool 549
To Use the Burn Tool 549

Drawing Shapes 550

Using the Bezier and Cusped Bezier Tools 552
To Use the Bezier or Cusped Bezier Tools 552

Using the B-Spline Tool 554
To Use the B-Spline Tool 554

Using the Ellipse, Rectangle, and Cusped Rectangle Tools 556
To Use the Ellipse, Rectangle, and Cusped Rectangle Tools 556

Using the Open Spline Tool 558
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>To Use the Open Spline Tool</td>
<td>558</td>
</tr>
<tr>
<td>Setting Default RotoPaint Tools and Settings</td>
<td>561</td>
</tr>
<tr>
<td>To Set a Tool as Your Default Tool:</td>
<td>561</td>
</tr>
<tr>
<td>To Set Your Default Tool Properties:</td>
<td>562</td>
</tr>
<tr>
<td>Selecting the Output Format and Channels</td>
<td>564</td>
</tr>
<tr>
<td>Selecting Existing Strokes/Shapes for Editing</td>
<td>566</td>
</tr>
<tr>
<td>To Adjust Display Properties for Selected Shapes/Strokes</td>
<td>566</td>
</tr>
<tr>
<td>To Select an Entire Stroke/Shape</td>
<td>567</td>
</tr>
<tr>
<td>To Select a Spline</td>
<td>567</td>
</tr>
<tr>
<td>To Select Only Points on a Stroke/Shape</td>
<td>568</td>
</tr>
<tr>
<td>To Select Feather Points</td>
<td>568</td>
</tr>
<tr>
<td>To Select Points Using a Transform Handle</td>
<td>568</td>
</tr>
<tr>
<td>Editing Existing Stroke/Shape Attributes</td>
<td>569</td>
</tr>
<tr>
<td>Editing Color</td>
<td>569</td>
</tr>
<tr>
<td>Editing Opacity</td>
<td>569</td>
</tr>
<tr>
<td>Selecting a Source for Your Stroke/Shape</td>
<td>570</td>
</tr>
<tr>
<td>Editing Blending Mode</td>
<td>570</td>
</tr>
<tr>
<td>Transforming Strokes/Shapes/Groups</td>
<td>572</td>
</tr>
<tr>
<td>To Transform a Stroke/Shape Using a Transform Handle Jack:</td>
<td>573</td>
</tr>
<tr>
<td>To Transform Points Using a Transform Box:</td>
<td>573</td>
</tr>
<tr>
<td>To Transform Onion Skin Source</td>
<td>573</td>
</tr>
<tr>
<td>Adjusting Mask Controls</td>
<td>575</td>
</tr>
<tr>
<td>Editing Shape-Specific Attributes</td>
<td>576</td>
</tr>
<tr>
<td>Adding and Removing Feather</td>
<td>576</td>
</tr>
<tr>
<td>Editing Stroke-Specific Attributes</td>
<td>577</td>
</tr>
<tr>
<td>Selecting a Source Image</td>
<td>577</td>
</tr>
</tbody>
</table>
Editing Brush Type 577
Editing Brush Size 577
Editing Brush Spacing 578
Editing Brush Hardness 578
Adjusting Write On 579

Editing Clone or Reveal Attributes 580

Editing Existing Stroke/Shape Timing 581

Editing Existing Stroke/Shape Stack Order 582

Editing Existing Stroke/Shape Splines 583
To Add a Point to a Stroke/Shape 583
To Move a Point 583
To Move Several Points Together 583
To Delete a Point 584
To Delete an Entire Stroke/Shape 584

Copying, Cutting, and Pasting Stroke Attributes 585
To Copy Stroke/Shape Attributes 585
To Cut Stroke/Shape Attributes 585
To Paste Stroke/Shape Attributes 586

Point Cusping, Smoothing, Expressions, and Links 587
To Cusp or Smooth Points 587
To Add Expressions to Points 587
To Link Other Controls to a Point’s Position 588

Animating Strokes/Shapes 589
To animate a stroke/shape using autokey 589
To animate strokes/shapes manually 589
Viewing Spline Keyframes 591
  To View Keyframes on the Timeline 591

Deleting or Rippling Keyframes 592
  To Delete a Keyframe 592
  To Delete All Keyframes for a Stroke/Shape 592
  To Ripple Keyframes 592

Copying, Cutting, and Pasting Animations 594
  To Copy Animations 594
  To Cut Animations 594
  To Paste Animations 595

Adding Motion Blur 597
  Adding Blur to Shapes 597
  Adding Global Motion Blur 599

Viewing Points in the Curve Editor and the Dope Sheet 601

Copying, Pasting, and Cutting Stroke Positions 602
  To Copy Stroke/Shape Positions 602
  To Cut Stroke/Shape Positions 602
  To Paste Stroke/Shape Positions 603

Copying, Pasting, and Cutting Point Positions 604
  To Copy Point Values 604

Pasting Point Positions 605
  To Paste Point Values 605

Cutting Point Positions 606
  To Cut Point Values 606
RotoPaint and Stereoscopic Projects 607
Selecting the View to Draw On 607
Selecting the View to Clone From 607

Reproducing Strokes/Shapes in Other Views 608

Editing Strokes/Shapes in One View Only 609
To Edit Stroke/Shape Points 609
To Edit Stroke/Shape Attributes 609

Applying a Stereo Offset 611

Where Are the Bezier and Paint Nodes? 612

Tracking and Stabilizing 613
Before Tracking 613
Quick Start 613

Connecting the Tracker Node 614

Adding Track Anchors 615
Positioning Track Anchors 615

Tracking Preferences and Viewer Tools 618

Automatic vs. Keyframe Tracking 620

Automatic Tracking 621
Calculating Auto-Tracks 621
Troubleshooting Auto-Tracks 622
Dealing with Occlusions 623

Keyframe Tracking 625
Calculating Keyframe Tracks 625
Importing Tracking Data 626
Troubleshooting Keyframe Tracks 628
Dealing with Occlusions 629

Applying Tracking Data 631
Applying Tracking Data Using Tracker’s Controls 631
  Stabilizing Elements 631
  Match-moving Elements 632
  Removing or Adding Jitter 633
Applying Tracking Data Using Linking Expressions 634
  Creating Linking Expressions 634
  To Link Animated Parameters with a Tracker Node 635
Transforming Masks with Tracking Data 635
Using the CornerPin2D Node 637
  To Use the CornerPin2D Node 638

Transforming Elements 640

Transforming in 2D 641
Using the 2D Transformation Overlay 641
Translating Elements 642
  Using the Transform Node 642
  Using the Position Node 643
Rotating Elements 643
Scaling Elements 644
Skewing Elements 645
To Invert a Transform Effect 645
Bounding Box Warnings 646

Choosing a Filtering Algorithm 648

How Nodes Concatenate 653

Applying Core Transformations in 2.5D 656
Using the 3D Transformation Handles 656
Adding a Card3D Node 657
Translating Elements 657
To Translate an Element Using the Card3D Node 657
Rotating Elements 657
Scaling Elements 658
Skewing Elements 658
To Skew an Element Using the Card3D Node 658
Specifying the Order of Operations 659
To Choose the Operation Order for Scales, Rotations, and Translations 659
To Choose the Operation Order for Rotations 659
Choosing a Filtering Algorithm 659
Adding Motion Blur 660
To Add Motion Blur 660
To Add Motion Blur to an Image Rendered in a Third-party Application 661
To Create Motion Blur with the VectorBlur Node 662
Replicating the Input Image Across the Output 664
To Use the Tile Node 664
Warping Images 666
Quick Start 666
Warping 667
Warping Images Using the GridWarp Node 667
To Warp an Image Using the GridWarp Node 667
Warping an Image Using the SplineWarp Node 675
To Warp an Image Using the SplineWarp Node 675
Using Boundary Shapes and Pins to Limit Warp 683
Morphing 686
Morphing One Image into Another Using the GridWarp Node 687
Morphing One Image into Another Using the SplineWarp Node 689
Using the cookie-cutter to create traveling masks 691
Transforming and Animating Warps and Morphs 694
Transforming Warps 694
Using the Tracker Node 694
Using the PlanarTracker Node 694
Using Expressions with GridWarp 695

Animating Warps 696
To Animate a Warp Using the GridWarp Node 696
To Animate a Warp Using the SplineWarp Node 696
Linking Transforms Using the SplineWarp Node 697

Temporal Operations 698
Quick Start 698

Distorting Time 700

Simple Retiming 701
To Retime All Frames in a Clip 701
To Retime a Range of Frames in a Clip 701

Interpolation 702
Frame-Blending 702
To Insert a FrameBlend Node 703

OFlow Retiming 705
To Retime with OFlow 705
Refining the Results 706

Warping Clips 709
To Warp a Clip 710

Global Frame Range and Speed 711

Applying a TimeBlur Filter 712
Applying Motion Blur to a Clip 712

Editing Clips 713

Offsetting and Slipping Clips 713
To Offset a Clip Using the TimeOffset Node 713
To Slip a Clip Using the TimeClip Node 714
Cutting Clips
To Cut a Clip

Splicing Clips
To Splice Slips

Freeze Framing Clips
To Output a Freeze Frame
To Output a Frame Interval

Working with Color

Using Scopes
Histogram
Using the Histogram Node
To Define Tonal Range with the Histogram Node
Waveform
Vectorscope

Using the Pixel Analyzer
Analyzing Pixel Selections
Analyzing Full Frames
Applying Analysis Data

Making Tonal Adjustments
Sampling White and Black Points
To Define Tonal Range with the Grade Node
Making Basic Corrections
Using ColorCorrect Sliders
To Adjust Contrast, Gain, Gamma or Offset with the ColorCorrect Node
Adjusting Black Levels with the Toe Node
Using ColorLookup Curves
To Make Basic Corrections with the ColorLookup Node

Making Hue, Saturation, and Value Adjustments
Correcting HSV
To make HSV corrections with the HSVTool node
Correcting Hue Only
Suppressing spill 743
Correcting Saturation Only 744
   To make saturation corrections with the Saturation node 744

Masking Color Corrections 745
To Mask a Color Correction 745

Applying Grain 747
Using Synthetic Grain 747
   To add synthetic grain with the Grain node 747
Using Practical Grain 748
   To Create Film Stock Sequences 748
   To Add Scanned Grain with the ScannedGrain Node 749
   To Mix the Grain and Backplate 750

Applying Mathematical Operations 751
Clamping Channel Values 751
Offsetting Channel Values 752
Inverting Channel Values 752
Multiplying Channel Values 753
Applying Expressions to Channel Values 753

Transforming the Color Space 755

Overriding the Default Cineon Conversion 756
To Override the Default Cineon Conversions Using Log2Lin 756
To Convert Between Logarithmic and Linear Color Spaces 756
To Create a 3D LUT in Log Color Space (for example Cineon) 757

Making Other ColorSpace Conversions 758
To Convert an Element in Nuke’s Native Color Space into Another Color Space 758
Converting Color Spaces with the OCIOColorSpace Node 758

Changing the Viewer Color Space 760
Adding Context Viewer Processes
Filtering and Spatial Effects
Applying Convolves
Using the Convolve Node
Simulating Depth-of-Field Blurring
Quick Start
Connecting ZDefocus
Adjusting Blur Settings
Adjusting the Shape of Out-of-Focus Highlights
Using a Predefined Disc Image
Using a Predefined Bladed Image
Using a Custom Filter Image
Enhancing Out-of-Focus Highlights
Masking Blur Effects
Creating Effects
Quick Start
Background Reflections on Foreground Elements
Using the LightWrap Node
Creating Star Filter Effects on Image Highlights
Using the Glint Node
Creating Text Overlays
Preparing a Text Overlay
Entering Text
Example Variables and Entities
Fonts and Font Properties
Selecting a Font
Adjusting Font Size and Spacing
Justifying Fonts 796

Updating the Font Cache 799

Transforming Text 800
  Transforming Selections 800
  Transforming Groups 801

Animating Transforms 802
  Animating Selections 802
  Animating Groups 802

Changing the Text Color 803

Masking Regions of the Viewer 804

Adding Shadows to Text 805

Adding a Drop Shadow 808

Analyzing and Matching Clips 811
  Introduction 811

Cropping Black Edges 813

Analyzing the Intensity of a Frame Sequence 814

Removing Flicker 815

Analyzing Exposure Differences 816

Tracking the Brightest and Darkest Pixels 817

3D Compositing 818
  Overview 818

Setting Up a Scene 820
The Scene Node
  To Add a Scene Node

The ScanlineRender Node
  To Add a ScanlineRender Node

The Camera Node

Using the 3D Viewer
  Switching to the 3D Viewer
  To Navigate in the 3D Viewer
  To Change the 3D Viewer Display Properties
  To Look Through a Camera
  To Lock the 3D Camera View
  To Use the Interactive 3D Camera View Mode

3D Scene Geometry

Using Built-in Primitive Geometry
  Working with Cards
    To Add a Card Object:
      Deforming Card Objects
  Working with Cubes
    To Add a Cube
    To Translate a Cube's Sides
  Working with Cylinders
    To Add a Cylinder
    Adjusting Geometric Resolution
  Working with Spheres
    To Add a Sphere
    Adjusting Geometric Resolution

Importing Geometry and Point Clouds from Other Applications

Importing Geometry from OBJ Files
  To import an OBJ object

Importing Geometry and Point Clouds from FBX Files
  Importing Meshes from FBX Files
  Importing Point Clouds from FBX Files
Importing Geometry and Point Clouds from Alembic Files
To Import Meshes and Point Clouds from an Alembic File

Creating Point Clouds and Geometry from Scratch

Creating a Position Pass Using the DepthToPosition Node
To Create a Position Pass using DepthToPosition

Creating a Dense Point Cloud Using the PositionToPoints Node
To Create a Point Cloud Using PositionToPoints

Creating a Point Cloud Using the DepthToPoints Node
To Create a Point Cloud Using DepthToPoints

Object Display Properties
To Edit an Object’s Display Attributes

3D Selection Tools
Selection Modes
Selecting Nodes
Selecting Vertices on a 3D Object
To select vertices in the Viewer
To Save or Restore Selected Vertices Using the GeoSelect Node
Selecting Faces on a 3D Object
Selecting 3D Objects
Matching Position, Orientation, and Size to 3D Selection

Merging Objects
To Merge Your 3D Objects

Modifying Object Shapes
Modifying Objects Using the EditGeo Node
Rendering a Wireframe Overlay on Your Geometry 881

Merging Two Shader Nodes 885
To Merge Two Shaders 885

Merging a Material with the Objects Behind 887
To Merge a Material with the Objects Behind 887

Replacing Material Channels with a Constant Color 891

Projecting Textures onto Objects 893
Projecting Textures with the UVProject Node 893
Projecting Textures with the Project3D Node 894

Importing UDIM Patches 895

Lighting 898

Working with Lights 899
Inserting Direct Lights 899
Inserting Point Lights 899
Inserting Spot Lights 900
Inserting Environment Lights 901
Inserting Direct Lights, Point Lights, or Spot Lights Using the Look Input 903
Importing Lights from FBX Files 907

Casting Shadows 909
To Cast Shadows: 910
Adjusting Shadows When Using ScanlineRender 911
Adjusting Shadows When Using PrmanRender 914

Manipulating Object Normals 916
To Manipulate Object Normals 916
Relighting a 2D Image Using 3D Lights
To Relight a 2D Image Using Relight

Cameras

Working with Cameras
To Add a Camera
To Look Through a Camera
To Lock the 3D Camera View
To Use the Interactive 3D Camera View Mode
To Edit a Camera's Lens Characteristics
Using the Look Input

Projection Cameras
First a Little Math...
Setting Up the Projection Camera Script
To Add a Projection Camera
To View a 3D Scene over a 2D Background Image

Importing Cameras from FBX Files
Importing Cameras from an FBX File

Importing Cameras from Alembic Files
To Import Cameras from an Alembic File:

Importing Cameras from Boujou
To Import a Camera from Boujou:

Locating a 3D Point from an Animated Camera
Connecting the PointsTo3D Node
Setting Reference Frames
Calculating the 3D Position

Transforming Geometry, Cameras, and Lights
Using the Transform Handles
- To Move an Object with the Transform Handles
- To Rotate an Object with the Transform Handles
- To Scale an Object with the Transform Handles

Transforming from the Node Properties Panel
- To Set Transformation Options
- To Transform an Object from Its Panel

Transformations and the Pivot Point
- To Move the Pivot Point

Parenting to Axis Objects
- To Add an Axis Object
- Using the Look Input

Using the TransformGeo Node
- To Have One 3D Object Always Face Another

Applying Tracks to an Object
- To apply a channel file to an object

Importing Transforms from FBX Files
- Importing Transforms from an FBX File

Importing Transforms from Alembic Files
- To Import Transforms from an Alembic file:

Adding Motion Blur to the 3D Scene

Adding Motion Blur Using a Renderer

Adding Motion Blur Using VectorBlur
- To use VectorBlur with ScanlineRender
- To use VectorBlur with RayRender
To use VectorBlur with MotionBlur3D 962
To use VectorBlur with Third-Party Motion Vectors 964

Exporting Geometry, Cameras, Lights, Axes, or Point Clouds 965

Rendering a 3D Scene 966
Choosing a Render Node 966
Rendering Out a Scene 966
To Add Motion Blur to the 3D Scene 967

Stereoscopic Scripts 968
Quick Start 968

Setting Up Views for the Script 970
Creating Views Automatically 970
Creating and Managing Views Manually 970

Loading Multi-View Images 974
To Read Images in 974
Combining Views from Different Files into a Single Output 975

Displaying Views in the Viewer 977
To Display a Particular View 977
To Display Two Views Next to Each Other 978
To Display a Blend Between Two Views 978

Selecting Which Views Display Changes 980
Splitting Views Off 980
To Show Separate Values for Each View 981
To Unsplit Views 982
Selecting the View to Process When Using the RotoPaint Node 982

Performing Different Actions on Different Views 983
To Extract a View for Processing 983

Reproducing Changes Made to One View 985
Reproducing X and Y Values 985

Swapping Views 987
To Rearrange Views 987

Converting Images into Anaglyph 988
To Convert Your Images into Anaglyph 988

Changing Convergence 990
To Change the Convergence Point of a Stereo Image 991
Using the Same Convergence Point Throughout a Sequence 992

Previewing Stereoscopic Images 994
Previewing Using the Viewer Stereo Modes 994
Displaying OpenGL Stereo in Comp Viewers 994
  Enabling OpenGL Stereo Output 995
  Windows 995
  Linux 996
  Switching to OpenGL Stereo Output 996
Flipbooking Stereo Sequences 997

Rendering Stereoscopic Images 999
Rendering EXR Files 999
Rendering Other File Formats 999

Deep Compositing 1001
About Deep Compositing 1001
Quick Start 1001

Reading in Deep Footage 1003

Importing DTEX Files 1004
Importing Scanline OpenEXR Files 1004
Creating Deep Data

Converting a 2D Image Sequence to a Deep Frame Using Input Frames

Converting a 2D Image to a Deep Image

Recoloring Depth Data

Using ScanlineRender to Generate Deep Data

Merging Deep Images

Creating Holdouts

Creating a Flattened Holdout with the DeepHoldout Node

Creating a Deep Holdout with the DeepMerge Node

Creating 2D and 3D Elements from Deep Images

Creating a 2D Image from a Deep Image

Creating a Point Cloud from a Deep Image

Modifying Deep Data

Color Correcting Deep Images

Adjusting the Effect of Deep Color Correction

Modifying Your Deep Images with Expressions

Cropping, Reformatting, and Transforming Deep Images

Cropping Deep Images

Reformatting Deep Images

Transforming Deep Samples

Sampling Deep Images

Writing Deep Data

Working with File Metadata

Metadata
Previewing in a Nuke Viewer
To Enable the ROI Render Feature
To Edit the Position or Size of Current ROI
To Disable the ROI Render Feature

Flipbooking Sequences
Setting the Viewer Cache Location and Size
To Use HieroPlayer as Nuke’s Flipbook
Flipbooking a Sequence
Capturing the Viewer

Previewing on an External Broadcast Video Monitor
To Preview Output on an External Broadcast Video Monitor
To Disable Viewing on an External Broadcast Video Monitor

Rendering Output
Render Resolution and Format

Output (Write) Nodes
To Render a Single Write Node
To Render Selected or All Write Nodes in the Script
Selecting Which Views to Render
Notes on Rendering QuickTime Files
  Advanced QuickTimes Options
Notes on Rendering OpenEXR Files

Rendering Using the Frame Server
Frame Server Preferences
Using the Frame Server on External Machines
Configuring the Frame Server on External Machines
Frame Server Logs
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>File Name Conventions for Rendered Images</td>
<td>1074</td>
</tr>
<tr>
<td>Writing Versions of Rendered Images</td>
<td>1074</td>
</tr>
<tr>
<td>Changing the Numbering of Rendered Frames</td>
<td>1075</td>
</tr>
<tr>
<td>Using Expressions</td>
<td>1075</td>
</tr>
<tr>
<td>Specifying a Frame Number for the First Frame in the Clip</td>
<td>1076</td>
</tr>
<tr>
<td>Offsetting All Frame Numbers by a Constant Value</td>
<td>1076</td>
</tr>
<tr>
<td>Using a Write Node to Read in the Rendered Image</td>
<td>1078</td>
</tr>
<tr>
<td>To Use a Write Node to Read in the Rendered Image</td>
<td>1078</td>
</tr>
<tr>
<td>What is the Hash Value?</td>
<td>1079</td>
</tr>
<tr>
<td>Render Farms</td>
<td>1080</td>
</tr>
<tr>
<td>Organizing Scripts</td>
<td>1081</td>
</tr>
<tr>
<td>Displaying Script Information</td>
<td>1082</td>
</tr>
<tr>
<td>Using Visual Diagnostics</td>
<td>1083</td>
</tr>
<tr>
<td>Enabling Visual Diagnostics</td>
<td>1083</td>
</tr>
<tr>
<td>Filtering Profile Data</td>
<td>1087</td>
</tr>
<tr>
<td>Exporting and Importing Profile Data</td>
<td>1090</td>
</tr>
<tr>
<td>Exporting Profile Data</td>
<td>1090</td>
</tr>
<tr>
<td>Importing Profile Data</td>
<td>1091</td>
</tr>
<tr>
<td>Using Performance Timing</td>
<td>1092</td>
</tr>
<tr>
<td>Enabling Performance Timings</td>
<td>1092</td>
</tr>
<tr>
<td>Outputting Performance Timings Onscreen</td>
<td>1092</td>
</tr>
<tr>
<td>Writing Performance Timings to File</td>
<td>1094</td>
</tr>
<tr>
<td>File Name Search and Replace</td>
<td>1095</td>
</tr>
<tr>
<td>To Search for a File Name or File Path and Replace It</td>
<td>1095</td>
</tr>
</tbody>
</table>
To Check if an Environment Variable Exists 1145
   From Inside Nuke 1145
   In the Windows Environment 1145
   In the Mac or Linux Environment 1145

To Display a List of Set Environment Variables 1145
   On Windows 1145
   On Mac or Linux 1146

Nuke Environment Variables 1147

Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts 1154

Defining the Nuke Plug-in Path 1156

Loading OFX Plug-ins 1157

Defining Common Favorite Directories 1158
   To Define a Common Set of Favorite Directories 1158
      Example 1 1158
      Example 2 1159

Handling File Paths Cross Platform 1161

Setting Default Values for Controls 1162

Defining Custom Menus and Toolbars 1163
   To Add a Toolbar 1164
      Example 1 1165
      Example 2 1165
   To Define a Menu or Toolbar Option 1165
      Example 1 1167
      Example 2 1167
      Example 3 1168

Defining Common Image Formats 1169
   Defining an Image Format 1169
      Example
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Creating and Accessing Gizmos</td>
<td>1170</td>
</tr>
<tr>
<td>Creating Gizmos</td>
<td>1170</td>
</tr>
<tr>
<td>Managing Gizmo Controls</td>
<td>1172</td>
</tr>
<tr>
<td>Adding Knobs Using Drag-and-Drop</td>
<td>1172</td>
</tr>
<tr>
<td>Editing Knobs</td>
<td>1175</td>
</tr>
<tr>
<td>Organizing Knobs</td>
<td>1177</td>
</tr>
<tr>
<td>Adding Tabs</td>
<td>1178</td>
</tr>
<tr>
<td>Hiding and Removing Knobs</td>
<td>1179</td>
</tr>
<tr>
<td>Adding Knobs Manually</td>
<td>1181</td>
</tr>
<tr>
<td>To Pick Existing Controls</td>
<td>1181</td>
</tr>
<tr>
<td>To Create New Controls:</td>
<td>1183</td>
</tr>
<tr>
<td>To Delete Controls:</td>
<td>1185</td>
</tr>
<tr>
<td>Accessing Gizmos in Nuke</td>
<td>1192</td>
</tr>
<tr>
<td>Sourcing Custom Plug-ins and Generic Tcl Scripts</td>
<td>1194</td>
</tr>
<tr>
<td>Nuke Custom Plug-ins</td>
<td>1194</td>
</tr>
<tr>
<td>To Source a Custom Plug-in</td>
<td>1194</td>
</tr>
<tr>
<td>Sourcing Tcl Procedure</td>
<td>1194</td>
</tr>
<tr>
<td>To Source a Generic Tcl Procedure</td>
<td>1194</td>
</tr>
<tr>
<td>Template Scripts</td>
<td>1195</td>
</tr>
<tr>
<td>Creating and Using a Template Script</td>
<td>1195</td>
</tr>
<tr>
<td>Defining Common Preferences</td>
<td>1196</td>
</tr>
<tr>
<td>Defining Default Preferences</td>
<td>1196</td>
</tr>
<tr>
<td>Deleting (and Resetting) the Preferences</td>
<td>1197</td>
</tr>
<tr>
<td>Altering a Script’s Lookup Tables (LUTs)</td>
<td>1198</td>
</tr>
<tr>
<td>Managing Nuke’s Native LUTs</td>
<td>1199</td>
</tr>
<tr>
<td>To Display LUT Curves</td>
<td>1199</td>
</tr>
<tr>
<td>To Create a New LUT</td>
<td>1199</td>
</tr>
<tr>
<td>To Edit LUTs</td>
<td>1200</td>
</tr>
</tbody>
</table>
To Reset the LUT Curves Back to Their Initial Default Shapes 1200
To Remove LUTs 1200

Selecting the LUT to Use 1202
Default LUT Settings 1202
To Change the Default LUT Settings 1203

Example Cases 1204
Working in Video Colorspace 1204
Linear Data in 16-Bit Files 1204
Cineon Displays 1204
Color Management 1204

OCIO Color Management 1205

Creating Custom Viewer Processes 1207

Using a Gizmo as a Custom Viewer Process 1209
To Register a LUT in the Project Settings as a Viewer Process 1209
To Create a Viewer Process Gizmo 1210
To Register a Custom Viewer Process 1211

Applying Custom Viewer Processes to Images 1213
To Apply Your Custom Viewer Process to Images Displayed in a Viewer 1213
To View the Controls of the Currently Active Viewer Process 1213

Expressions 1214
Quick Start 1214

Linking Expressions 1215
Referencing Values from Another Parameter 1216
Linking Channels and Formats Using Expressions 1218

Adding Mathematical Functions to Expressions 1219
Converting Expressions Between Scripting Languages

The Script Editor and Python
Quick Start

Using the Script Editor
Opening the Script Editor
Input and Output Panes
Entering a Statement
Moving Through and Clearing the Script History
Clearing the Output Pane

Automating Procedures
Saving Statements in a Python Module
Opening a Python Script in the Script Editor
Importing and Executing a Python Script

Nuke as a Python Module

Getting Help
More Documentation
Viewing More Examples
Using the Help Statement
Python on the Web

Timeline Editing in Nuke Studio

Using Tags
Using Quick Tags
Tagging Using the Viewer
Tagging Shots
Saving and Loading Projects
Autosaved Projects
Managing Timelines
Adding Tracks to the Timeline
Adding Clips to the Timeline
Audio and the Timeline
WAV Shots
Displaying Audio Waveforms
Audio Scrubbing
Synchronizing Audio and Video
PulseAudio on Linux
Stopping PulseAudio
Restarting PulseAudio
Using Reference Media
Comparing Media
Caching Frames in the Playback Cache
Caching Frames in the Disk Cache
Caching Sequence Ranges
Caching Selected Shot Ranges
Caching In/Out Ranges
Clearing Cached Frames
Clearing All Cached Frames
Clearing Sequences
Clearing Selected Shot Ranges
Clearing In/Out Ranges

Viewing Multi-Format Timelines
Setting the Sequence Format
Reformatting Shots

Refreshing and Replacing Shots

Setting Soft Trims

Enabling and Disabling Shots

Adding Transitions
Invalid Transitions

Retiming Shots

Using Freeze Frames

Blending Tracks on the Timeline
Adding New Blend Tracks
Converting Tracks to Blend Tracks
Masking Blended Tracks

Stereoscopic and Multi-View Projects

Creating Views in a Project
Creating and Managing Views

Importing Source Clips
Importing Single-View Clips 1338
  File Name Variables 1338
  Directory Name Variables 1340

Importing Multi-View Clips 1342
  Multi-View QuickTime Files 1342

Displaying Views in the Viewer 1343
Displaying a Particular View 1343
Displaying Two Views Next to Each Other 1343
Displaying a Blend Between Two Views 1344
Displaying OpenGL Stereo in Timeline Viewers 1344
  Enabling OpenGL Stereo Output 1344
  Windows 1344
  Linux 1346
  Switching to OpenGL Stereo Output 1346

Displaying Views in the Timeline 1348
Splitting Views to Separate Tracks 1348
  Displaying Two Views Next to Each Other 1350
  Displaying a Blend Between Two Views 1350

Applying Changes to Selected Views 1352
Splitting Views to Tracks 1352
Splitting Views in the Properties Panel 1353
Showing Separate Values for Each View 1354
Unsplitting Views 1355

Soft Effects 1356
Available Soft Effects 1356

Adding Sequence-Level Soft Effects 1359
  Using the Spreadsheet View 1360

Adding Shot-Level Soft Effects 1362

Soft Effect Controls 1362

Editing Sequence-Level Soft Effects 1364
  Moving 1364
  Copying 1364
  Cloning 1364
Copying as Clone 1365
Decloning 1365
Deleting 1365

Editing Shot-Level Soft Effects 1366
  Enabling and Disabling Soft Effects 1366

Create Comp 1367
Edit Export Settings 1367
Edit Project Settings 1368
Creating and Editing a Comp 1369
Nuke Comp Colors 1371
Creating Multiple Shot Comps 1372
Create Comp Special 1374
Enabling and Disabling a Nuke Comp 1376
Saving a New Version and Updating the Comp 1376
Stereo and Multi-View Comps 1376

Annotations 1380
Workflow 1380
The Annotations Menu 1381
Adding Annotations 1381
Enabling and Disabling Annotations 1384
Editing Sequence-Level Annotations 1384
Editing Shot-Level Annotations 1385
Viewing Annotations in the Compositing Environment 1386
  Viewing the Annotation Node Group 1386
Re-Exporting Annotations from the Timeline 1387

Timeline Editing Tools 1389
Using the Multi Tool 1392
Using the Move/Trim Tool 1394
Moving Shots 1394
To Move Shots Using the Spreadsheet View:

Trimming Shots
To Trim Shots Using the Spreadsheet View:

Using the Selection Tools

Using the Slip Clip Tool
Slipping Using the Spreadsheet View

Using the Slide Clip Tool

Using the Roll Edit Tool

Using the Retime Clip Tool

Using the Razor and Join Tools
Copying Cuts Between Tracks

Insert, Overwrite, and 3-Point Editing
Inserting Clips
Overwrite Edits
3-Point Editing

Versions and Snapshots

Using Versions

Versions in Bins

Versions in Sequences

Using Snapshots

Creating Snapshots

Restoring Snapshots
Exporting from Nuke Studio 1433

Introduction to the Export Dialog 1434

Using Local and Project Presets 1436

Using the Shot Template 1436

Custom Shot Templates 1439

Multi-format Exports 1439

Adding Burn-in Text to Exports 1442

Adding Additional Nodes During Export 1444

Using the Frame Server on External Machines 1446

Configuring the Frame Server on External Machines 1446

Frame Server Logs 1448

Exporting Sequences and Shots 1450

Nuke Project File Settings 1451

Nuke Write Node Settings 1453

Tracks, Range, and Handles Settings 1456

Building VFX Tracks and Comp Clips 1458

Building Tracks From Export Structure 1458

Building Comp Clips From Export Tags 1460

Building Render Placeholders From Export Tags 1461

Exporting Multi-View Source Clips 1463

Transcoding 1465

Transcoding a Sequence 1465

Tracks and Range Settings 1467
Transcoding a Sequence as Shots

Transcoding Timeline Selections

Transcoding from the Bin View

Ad Hoc Exports

Exporting EDLs and XMLs

Using the Audio Exporter

Exporting Audio from Sequences

Exporting Audio from Shots

Exporting Audio from Source Clips

Using the Copy Exporter

Using the SymLink Generator

Advanced Compositing with NukeX and Nuke Studio

Retiming and Motion Blur

VectorGenerator

Quick Start

Connecting VectorGenerator

Viewing and Refining the Results

Kronos

Example Input Speed Calculation

Quick Start
Tracking with PlanarTracker 1532

Tracking a Plane 1533
Drawing a Plane to Track 1533
Tracking the Footage 1534
Tracker Menu Options in the Viewer 1536
Track Motion Controls 1536
Tracking Controls 1537
Clear Tracking Data Controls 1537
Planar Tracker Surface Controls 1538
Reference Frame Controls 1538
Keyframe Controls 1538
Layer Dropdown 1539
CornerPin Dropdown 1539
Tracking Tab Menu Options 1540
Settings 1540
Export 1541
Correction 1541
Reference Frame 1541

Reusing a Track Result 1543
Reusing a Tracked Plane 1543
Reusing a Single Track 1543

Placing an Image on the Planar Surface 1544

Camera Tracking 1548
Introduction 1548
Quick Start 1548

Connecting the CameraTracker Node 1550

Masking Out Regions of the Image 1551

Working with Multi-View Scripts 1553
Pre-tracking Checks 1577
Post-Tracking Refinements 1577
Retracking Partial Frame Ranges 1578

Working with User Tracks 1579

Adding and Positioning User Tracks 1580

User Tracking Methods 1582
Auto User Tracks 1582
Manual User Tracks 1583
Extracting User Tracks 1584
Importing a Tracker 1584

Tracking Assists 1585

Tracking a Scene Manually 1586

Linking Still Reference Frames 1587

Assigning 3D Survey Points 1588

Solving the Camera Position 1590

Viewing Solve Data 1591
Reviewing AutoTracks Curves 1592
Reviewing Solved User Tracks 1593
Previewing Matchmove Quality
  Method 1 1593
  Method 2 1594

Troubleshooting Solves 1597
Using Curve Thresholds to Delete Tracks 1597
Refining a Solve 1599

Updating Solves with Extended Tracking Data 1600
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Adjusting the Scene</td>
<td>1601</td>
</tr>
<tr>
<td>Setting the Ground Plane and Axes</td>
<td>1602</td>
</tr>
<tr>
<td>Setting Axes Individually</td>
<td>1603</td>
</tr>
<tr>
<td>Setting the Origin Within a Scene</td>
<td>1603</td>
</tr>
<tr>
<td>Transforming the Scene Manually</td>
<td>1605</td>
</tr>
<tr>
<td>Transforming Using Channel Files</td>
<td>1605</td>
</tr>
<tr>
<td>Transforming Using Snap To Position</td>
<td>1606</td>
</tr>
<tr>
<td>Using Scene Constraints</td>
<td>1607</td>
</tr>
<tr>
<td>Using Solve Data</td>
<td>1609</td>
</tr>
<tr>
<td>Creating Camera Nodes</td>
<td>1610</td>
</tr>
<tr>
<td>Creating a Camera Rig</td>
<td>1610</td>
</tr>
<tr>
<td>Creating a Camera Set</td>
<td>1611</td>
</tr>
<tr>
<td>Creating Scenes</td>
<td>1613</td>
</tr>
<tr>
<td>Creating a 3D Scene with ScanlineRender and LensDistortion</td>
<td>1613</td>
</tr>
<tr>
<td>Creating Point Clouds</td>
<td>1615</td>
</tr>
<tr>
<td>Creating Cards</td>
<td>1616</td>
</tr>
<tr>
<td>Combining Solves</td>
<td>1618</td>
</tr>
<tr>
<td>Placing Objects in the Scene</td>
<td>1619</td>
</tr>
<tr>
<td>Accounting for Lens Distortion</td>
<td>1620</td>
</tr>
<tr>
<td>Distorting Your CG Elements to Match Your 2D Footage</td>
<td>1620</td>
</tr>
<tr>
<td>Undistorting Your 2D Footage to Match Your CG Elements (Using LensDistortion)</td>
<td>1620</td>
</tr>
<tr>
<td>Undistorting Your 2D Footage to Match Your CG Elements (Using CameraTracker)</td>
<td>1621</td>
</tr>
</tbody>
</table>
About the Lens Model
Using MatchGrade
Extracting a Baked-In Grade
Analyzing Reference Frames
Auto-Analyzing Per Frame
Matching a Grade Between Two Different Clips
Using the Smart Vector Toolset
Quick Start
Generating Motion Vectors
Masking Foreground and Background Areas
Adding Paint to the Source
Adding an Image to the Source
Applying Motion Vectors to the Source
Warping the Vectors
Improving Warps
Warping Images Using VectorCornerPin
Warping Multiple Reference Frames
Generating Depth Maps
Quick Start
Connecting DepthGenerator
Selecting What to Output
Analyzing Depth
Connecting Particle Nodes 1725

Emitting Particles 1726

Spawning Particles with ParticleSpawn 1731

Adjusting the Speed and Direction of Particles 1732
Applying Gravity to Particles 1732
Aligning Particles 1732
Controlling Particle Speeds 1732
Attracting Particles to a Specific Point 1733

Modifying the Particles’ Movement 1734
Bouncing Particles off Objects 1734
Adding Drag to Particles 1735
Adding Turbulence Motion on Particles 1735
Adding Spiral Motion to Particles 1736
Adding a Wind Effect to Particles 1737

Adjusting Controls Common to Several Particle Nodes 1738
Particle Rendering Controls 1738
Particle Transform Controls 1738
Condition and Region Controls 1738
  Conditions tab 1738
  Region tab 1739

Adjusting Particle Properties Using Curves 1740

Adjusting Particles Using Expressions 1741
Particle Expression Functions 1742

Adjusting Particle Simulation Settings 1746

Merging Particle Streams 1747
Controlling Particles by Channel 1748
Caching Particles 1751
PrmanRender 1753
Setting Up RenderMan Pro Server and PrmanRender 1753
Using The PrmanRender Node 1754
Render Quality 1755
Shadows, Reflections, Refractions and Depth of Field 1756
Motion Blur Parameters 1757
Shader Parameters 1758
RIB Parameters 1759
Using the ModifyRIB Node 1760
Using the Reflection Node 1761
Using the Refraction Node 1762
Using F_DeFlicker2 1763
Introduction 1763
Quick Start 1764
Parameters 1765
Using F_ReGrain 1766
Introduction 1766
Quick Start 1767
Parameters 1787

Global Motion Estimation 1790
Introduction 1790

What is Global Motion Estimation? 1791
Limitations of GME 1791

Global Motion Estimation Effects 1792
Using Them 1792

Controls 1794
Parameters That Affect Analysis 1794
Parameters That Affect Rendering 1795

Widgets 1796

Using F_Align 1797
Introduction 1797

Quick Start 1799
Analyzing with the Analyse button 1799
Analyzing On The Fly 1800

Parameters 1801

Using F_RigRemoval 1803
Introduction 1803

Quick Start 1804

Occlusions 1806

Parameters 1808

Using F_Steadiness 1810
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction</td>
<td>1810</td>
</tr>
<tr>
<td>Quick Start</td>
<td>1812</td>
</tr>
<tr>
<td>Smoothing Out Camera Motion</td>
<td>1812</td>
</tr>
<tr>
<td>Locking To A Frame</td>
<td>1812</td>
</tr>
<tr>
<td>Parameters</td>
<td>1814</td>
</tr>
<tr>
<td>Using the BlinkScript Node</td>
<td>1817</td>
</tr>
<tr>
<td>About BlinkScript</td>
<td>1817</td>
</tr>
<tr>
<td>Quick Start</td>
<td>1819</td>
</tr>
<tr>
<td>Connecting the BlinkScript Node</td>
<td>1820</td>
</tr>
<tr>
<td>Loading, Editing, and Saving Kernels</td>
<td>1821</td>
</tr>
<tr>
<td>Loading Kernels</td>
<td>1821</td>
</tr>
<tr>
<td>Editing Kernels</td>
<td>1822</td>
</tr>
<tr>
<td>Saving Kernels</td>
<td>1824</td>
</tr>
<tr>
<td>Setting Kernel Parameters</td>
<td>1825</td>
</tr>
<tr>
<td>GPU or CPU?</td>
<td>1825</td>
</tr>
<tr>
<td>Specifying the Output Format</td>
<td>1826</td>
</tr>
<tr>
<td>Changing Performance Settings</td>
<td>1826</td>
</tr>
<tr>
<td>Publishing and Protecting Your Kernels</td>
<td>1827</td>
</tr>
<tr>
<td>Limitations and Known Issues</td>
<td>1829</td>
</tr>
<tr>
<td>RGBA Only</td>
<td>1829</td>
</tr>
<tr>
<td>GPU</td>
<td>1829</td>
</tr>
<tr>
<td>Crashing and Infinite Loops</td>
<td>1829</td>
</tr>
<tr>
<td>Known Issues</td>
<td>1829</td>
</tr>
<tr>
<td>OpenCL on OS X 10.8 (Mountain Lion)</td>
<td>1829</td>
</tr>
<tr>
<td>Written Tutorials</td>
<td>1830</td>
</tr>
<tr>
<td>Topic</td>
<td>Page</td>
</tr>
<tr>
<td>--------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>Importing Image Sequences</td>
<td>1852</td>
</tr>
<tr>
<td>To Read the Images</td>
<td>1852</td>
</tr>
<tr>
<td>Navigating Inside the Windows</td>
<td>1855</td>
</tr>
<tr>
<td>Panning Your View</td>
<td>1855</td>
</tr>
<tr>
<td>Zooming or Magnifying Your View</td>
<td>1855</td>
</tr>
<tr>
<td>Using the Node Graph Overview</td>
<td>1855</td>
</tr>
<tr>
<td>Framing the View in the Window</td>
<td>1856</td>
</tr>
<tr>
<td>Working with Viewers</td>
<td>1857</td>
</tr>
<tr>
<td>Displaying the Images in a Viewer Window</td>
<td>1858</td>
</tr>
<tr>
<td>Viewing Multiple Inputs</td>
<td>1860</td>
</tr>
<tr>
<td>Reformatting Images</td>
<td>1862</td>
</tr>
<tr>
<td>To Conform Images to the Project Format</td>
<td>1862</td>
</tr>
<tr>
<td>Using Proxies and “Down-res”</td>
<td>1863</td>
</tr>
<tr>
<td>To Activate Proxy Mode</td>
<td>1863</td>
</tr>
<tr>
<td>To Activate “Down-Res”</td>
<td>1863</td>
</tr>
<tr>
<td>Compositing Images</td>
<td>1865</td>
</tr>
<tr>
<td>To Composite Two Nodes</td>
<td>1865</td>
</tr>
<tr>
<td>Color-Correcting Images</td>
<td>1868</td>
</tr>
<tr>
<td>Masking Effects</td>
<td>1869</td>
</tr>
<tr>
<td>To Create and Apply a Bezier Mask</td>
<td>1869</td>
</tr>
<tr>
<td>Creating Flipbook Previews</td>
<td>1870</td>
</tr>
<tr>
<td>To Generate a Flipbook</td>
<td>1870</td>
</tr>
<tr>
<td>Rendering Final Output</td>
<td>1870</td>
</tr>
<tr>
<td>To Render the Result of Your Composite</td>
<td>1870</td>
</tr>
<tr>
<td>Using the Nuke Frame Number Variable</td>
<td>1872</td>
</tr>
<tr>
<td>Image Formats</td>
<td>1872</td>
</tr>
</tbody>
</table>
Rendering with the Active Resolution  
Rendering Multiple Channels  

Epilogue  

Tutorial 2: 2D Point Tracking  
Introduction  

One-Point, Two-Point, Three-Point, Four  

Open the Tutorial Project File  
To Open the Project File  

Tracking a Single Feature  
Setting a Tracking Anchor  
Auto-Tracking vs. Keyframe Tracking  
  Using Auto-Tracking  
  Using Keyframe Tracking  
Editing Track Data  

Tracking Obscured Features  
To Track a Feature That Moves off Screen  

Stabilizing Elements  
To Track and Stabilize  

Match-Moving Elements  
To Match-Move an Element  

Epilogue  

Tutorial 3: Keying and Mattes  
Introduction  

Open the Tutorial Project File  
To Open the Project File
Keying with Primatte 1896
To Pull a Key with Primatte (Method 1) 1896
To Pull a Key with Primatte (Method 2) 1897

Image-Based Keying 1901
To Pull a Key with IBK 1901

Rotoscopying 1906
To Draw a Garbage Matte 1906

Epilogue 1911

Tutorial 4: 3D Integration 1912
Introduction 1912

The Basic 3D System 1913
The 3D Viewer 1913
The Geometry or Scene Node 1914
The Camera Node 1914
The ScanlineRender Node 1914

Open the Tutorial Project File 1916
To Open the Project File 1916

Setting Up a 3D System 1917
To Set Up a 3D Node Tree 1917
To Position Objects in the Scene 1919

Making a Scene 1922
To Set Up a Scene 1922

Merging and Constraining Objects 1925
To Move the Merged Objects Together 1926
To Make Objects ‘Look’ at the Camera 1926
Animating a Scene
To Animate the Camera

Working with Geometry
To Add Primitive Objects to the Scene

Lighting and Surface Properties
To Define Surface Attributes to Objects
To Add Light Objects to a Scene

Epilogue

Appendices
Organization of the Section

Appendix A: Preferences
The Available Preference Settings
General
Project Defaults
Performance
Behaviors
Panels

Appendix B: Keyboard Shortcuts
Conventions
Global
Nuke Studio's Timeline Viewer
Nuke Studio's Timeline
2D Compositing Viewer
3D Compositing Viewer
Node Graph
Project/Tags/Versions Bin
Properties Panel
<table>
<thead>
<tr>
<th>Section</th>
<th>Year</th>
</tr>
</thead>
<tbody>
<tr>
<td>Curve Editor/Dope Sheet</td>
<td>2003</td>
</tr>
<tr>
<td>Script Editor</td>
<td>2004</td>
</tr>
<tr>
<td>Roto/RotoPaint</td>
<td>2005</td>
</tr>
<tr>
<td>Appendix C: Supported File Formats</td>
<td>2007</td>
</tr>
<tr>
<td>Notes</td>
<td>2007</td>
</tr>
<tr>
<td>Supported File Formats</td>
<td>2007</td>
</tr>
<tr>
<td>Supported Camera Formats</td>
<td>2016</td>
</tr>
<tr>
<td>Supported Audio Formats</td>
<td>2018</td>
</tr>
<tr>
<td>Appendix D: External Software</td>
<td>2019</td>
</tr>
<tr>
<td>Third-Party Contributions</td>
<td>2019</td>
</tr>
<tr>
<td>Third-Party Libraries and Fonts</td>
<td>2021</td>
</tr>
<tr>
<td>Third-Party Library Versions</td>
<td>2021</td>
</tr>
</tbody>
</table>
Getting Started

An overview of installation and licensing, the default Nuke environments and workspaces, and the Preferences dialog.

The Nuke Family

Learn about the different products and modes of Nuke as well as some key concepts, everything you should know before using Nuke products.

Installation & Licensing

Find out how to install and license Nuke on Windows, Mac, and Linux to the point where you have the application in front of you and are ready to start work.

Comp Environment

Discover the basics of Nuke's compositing environment, designed to streamline day-to-day workflow. Flexible, efficient and feature packed, this toolset delivers film-grade results, fast.

Timeline Environment

Explore Nuke Studio's timeline environment allowing you to conform, review, edit, and even create and render compositions from the timeline.
Meet the Nuke Product Family

These pages introduce the different products and modes of Nuke and provide links to some key concepts you should know before using Nuke products.

Nuke Products

The Nuke products have different purposes and different levels of functionality.

Nuke Studio is a product that combines the functionality of all the Nuke products listed below, which are then used as different modes in which to run Nuke Studio. It also includes its own specific functions that are only applicable to Nuke Studio.

All the Nuke products listed below can also be licensed as separate products, except for Nuke Assist, which is included with NukeX and Nuke Studio.

The Nuke Products include the following:

• **Nuke** - Foundry's award-winning compositing tool. Nuke provides a stable foundation from which to deliver shot-based VFX work.

• **NukeX** - includes all of Nuke's features with the addition of a range of specialized plug-in tools to eliminate the need to perform certain tasks in third-party software packages. To run it, you need both a Nuke and a NukeX license, or a Nuke license.


• **Nuke Studio** - the most powerful application in the Nuke product family, it includes all the functionality of Nuke and NukeX with the addition of timeline and conform workflows.

• **Nuke Assist** - a feature of NukeX intended for use as a workstation for artists performing painting, rotoscoping, and tracking. Two complementary licenses of Nuke Assist are included in every NukeX maintenance package. Nuke Assist only supports a limited subset of Nuke tools.

See About Nuke Assist for a run down of the functionality provided by Nuke Assist.

• **Nuke Non-commercial** - a special version of the commercial Nuke application that you can run without a license, intended for use as an educational tool.

See About Nuke Non-commercial for a run down of the restrictions placed on using the non-commercial version.
About Nuke

When you purchase Nuke, it contains all the Nuke products as different modes. For example, you can choose to run Nuke in NukeX mode, or Nuke Non-commercial mode.

Nuke modes include the following:

<table>
<thead>
<tr>
<th>Mode</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nuke Studio</td>
<td>Includes the Timeline environment and Compositing environment, and has some additional Nuke Studio specific features, such as the ability to create a Nuke Comp, add soft effects to the timeline, and the annotations menu. Nuke Studio combines the tools and abilities of Nuke, NukeX, and Hiero. For more information about Nuke Studio, see Nuke Studio Environments.</td>
</tr>
<tr>
<td>Nuke</td>
<td>A single shot compositor (what is referred to as the Compositing environment in Nuke Studio), offering a Node Graph, Viewer, animation tools, and so on.</td>
</tr>
<tr>
<td>NukeX</td>
<td>Just like Nuke, a single shot compositor (what is referred to as the Compositing environment in Nuke Studio), with additional tools such as CameraTracker, DepthGenerator, FurnaceCore, Kronos, and so on. See About NukeX for more information.</td>
</tr>
<tr>
<td>Nuke Assist</td>
<td>Provides basic tools including Bezier, Draw, Roto, and so on. The Nuke Assist mode is intended for use as a workstation for artists performing painting, rotoscoping, and tracking. See About Nuke Assist for more information.</td>
</tr>
<tr>
<td>Nuke Non-commercial</td>
<td>This mode is intended for personal and educational use. It includes all the features of the commercial version of Nuke, offering you a chance to explore and learn the application fully while using it from the comfort of your own home. See About Nuke Non-commercial for more information.</td>
</tr>
</tbody>
</table>
About NukeX

When using NukeX, you have all the features of Nuke in use, plus the following:

• CameraTracker
• DepthGenerator
• FurnaceCore
• Kronos
• LensDistortion
• MotionBlur
• ModelBuilder
• Particles
• PlanarTracker
• PointCloudGenerator
• PoissonMesh
• PrmanRender
• ProjectionSolver
• VectorGenerator

Nuke and NukeX are fully compatible, with Nuke capable of viewing and rendering - but not editing - NukeX features.
# About Nuke Assist

Nuke Assist is licensed as part of a NukeX or Nuke Studio maintenance package and is intended for use as a workstation for artists performing painting, rotooning, and tracking. Nuke Assist doesn't support any NukeX-or Nuke Studio-specific features apart from PlanarTracker, and has a limited subset of Nuke nodes and features available. Nuke Assist does not support any custom plug-ins, only the following nodes are supported:

<table>
<thead>
<tr>
<th>Node Type</th>
<th>Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Image</strong></td>
<td>Checkerboard, ColorBars, ColorWheel, Constant</td>
</tr>
<tr>
<td><strong>Read</strong></td>
<td>Viewer</td>
</tr>
<tr>
<td><strong>Draw</strong></td>
<td>Bezier, Radial, Ramp, Rectangle</td>
</tr>
<tr>
<td><strong>Roto</strong></td>
<td>RotoPaint</td>
</tr>
<tr>
<td><strong>Time</strong></td>
<td>FrameBlend, FrameHold, FrameRange, TimeEcho</td>
</tr>
<tr>
<td><strong>Channel</strong></td>
<td>Add, Copy, ChannelMerge, Remove, Shuffle, ShuffleCopy</td>
</tr>
<tr>
<td><strong>Color</strong></td>
<td>Grade, Invert, OCIO CDLTransform, OCIO Colorspace, OCIO Display, OCIO FileTransform, OCIO LogConvert</td>
</tr>
</tbody>
</table>

**Note:** Bezier is only available through the X menu. Press X in the Node Graph and then enter Bezier as a Tcl command to add the node.
### Nuke Assist Nodes

<table>
<thead>
<tr>
<th>Category</th>
<th>Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Filter</td>
<td></td>
</tr>
<tr>
<td>Blur</td>
<td></td>
</tr>
<tr>
<td>Keyer</td>
<td></td>
</tr>
<tr>
<td>Merge</td>
<td></td>
</tr>
<tr>
<td>AddMix</td>
<td>Dissolve, KeyMix, Merge</td>
</tr>
<tr>
<td>Premult</td>
<td>Switch, UnpremUlt</td>
</tr>
<tr>
<td>Transform</td>
<td></td>
</tr>
<tr>
<td>Crop</td>
<td>CornerPin, PlanarTracker, Reformat</td>
</tr>
<tr>
<td>Tracker</td>
<td>Transform, TransformMasked</td>
</tr>
<tr>
<td>Views</td>
<td></td>
</tr>
<tr>
<td>JoinViews</td>
<td>OneView, ShuffleView, Split and Join</td>
</tr>
<tr>
<td>Stereo Anaglyph</td>
<td>Stereo MixViews, Stereo ReConverge, Stereo SideBySide</td>
</tr>
<tr>
<td>Metadata</td>
<td></td>
</tr>
<tr>
<td>AddTimeCode</td>
<td>CompareMetadata, CopyMetadata, ModifyMetadata</td>
</tr>
<tr>
<td>ViewMetadata</td>
<td></td>
</tr>
<tr>
<td>Other</td>
<td></td>
</tr>
<tr>
<td>Backdrop</td>
<td>Dot, Group, Input</td>
</tr>
<tr>
<td>Output</td>
<td>PostageStamp, StickyNote</td>
</tr>
</tbody>
</table>

**Note:** You cannot render output using Write nodes in Nuke Assist.

You can load projects created in Nuke's other modes and work as normal, within the constraints of Nuke Assist. The Viewer renders the output of the node tree whether the components are supported or not. Any unsupported nodes and plug-ins in the Node Graph are outlined in red and their controls are grayed out. You cannot modify the output of unsupported nodes and plug-ins.
**Tip:** For more information on node trees and node-based compositing, see Understanding the Workflow.

**Using Gizmos, Groups, Precomps, Knobs, and Python**

Gizmos, Group, and Precomp nodes can be loaded as normal, but if they contain any nodes that are not supported by Nuke Assist they have a red outline around the node in the Node Graph and the control panel controls are grayed out.

**Note:** Nuke Assist allows the creation of custom knobs, but they can't be edited.

Python scripts work as usual for nodes that are supported by Nuke Assist, but any attempt to add unsupported nodes displays an error message in the Script Editor output section, or terminal (in -t mode).

For example, executing `nuke.createNode('Transform')` adds a Transform node to the Node Graph, but `nuke.createNode('Convolve')` prints the following error:

```python
# Result:
Traceback (most recent call last):
File "<string>", line 1, in <module>
RuntimeError: Convolve is not available in Nuke Assist
```
About Nuke Non-commercial

Nuke Non-commercial is a free version of Nuke that runs outside the regular licensing model. Nuke Non-commercial is meant for personal, educational, and other non-commercial use. It is aimed at students, industry professionals, and others interested in Nuke. It includes most of the features of the commercial version of Nuke, offering you a chance to explore and learn the application fully while using it from the comfort of your own home.

You can run Nuke, NukeX, and Nuke Studio in non-commercial mode using the --nc command line argument. For example, to launch Nuke Studio in non-commercial mode on Mac, enter:


Nuke Non-commercial is a fully functional version of Nuke, but as it’s designed for non-commercial use only, it does differ from the commercial version in some aspects. Here are the main differences:

• Certain nodes are disabled in Nuke Non-commercial, including BlinkScript, GenerateLUT, Primatte, Ultimatte, and WriteGeo.
• Rendered output is restricted to 1920x1080 HD and the MPEG4 and H.264 formats are disabled.
• command line renders are restricted to encrypted .nknc scripts.
• Frame Server slave rendering is disabled.
• Exporting EDL/XML sequences is disabled.
• Exporting LUTs from MatchGrade is disabled.
• Gizmos, clipboard contents, .hrox project files, and .nk scripts are all encrypted.
• Monitor Output is disabled.

In other respects, Nuke Non-commercial contains all the functionality of the commercial version of Nuke.
Key Concepts

Nuke products are a resolution-independent compositing system, with extensive channel support, powerful image manipulation tools, and a rich 3D compositing environment. This section explains concepts you should know before using Nuke products.

Understanding the Workflow

A Nuke project consists of a network of linked operators called nodes. Nuke’s Compositing environment utilizes a node-based workflow, where you connect a series of nodes to read, process, and manipulate images. Each node in the script, or comp, performs an operation and contributes to the output.

You can open a Nuke comp file in a text editor, and a series of sequential commands are displayed, which are interpreted and executed when you render the output.
In the image above, you see an example of a very simple Nuke script. Two Read nodes reference media on disk. Effect nodes extract a matte and blur an image. A Merge node set to over, composites the foreground image (input A) over the background image (input B). Finally, a Write node renders and outputs the completed composite to disk. There is also a Viewer node, which displays the output of any node in the script.

**Note:** Nuke Assist does not support Write nodes or render farms. See [Nuke Products](#) for more information.

## Working with Multiple Image Formats

Nuke products support multiple file formats, such as Cineon, TIFF, OpenEXR, HDRI, and RAW camera data (using the dcraw command line program), and allows you to mix them all within the same composite. By default, Nuke products convert all imported sequences to their native 32-bit linear RGB colorspace. You can, however, use the Colorspace node to force one of several color models, including sRGB, Cineon, rec709, gamma 1.80/2.20, HSV, or HSL. The Log2Lin node lets you convert between logarithmic and linear colorspace (and vice versa).
Note: Nuke Assist does not support Colorspace or Log2Lin nodes. See Nuke Products for more information.

There are no restrictions on image resolution - you can freely mix and scale elements of any resolution within the same script. You can, for example, use a 2k film plate as the background for video shot in PAL format, and then output the result in HD1080i. Nuke products automatically adjust their Viewers to accommodate the image you’re viewing.

Channel Operations

In Nuke products, you can assign the output of each node as new channels, and pass them to the next node in the script. When you need to re-use a particular channel (say, to apply a color correction to the hair), you simply choose the channel containing the matte from the downstream color-correction node.

Nuke products support up to 1023 channels of image data. This provides additional benefits when working with computer-generated (CG) elements, especially when such elements are rendered out in the OpenEXR format.

For example, your 3D department could render out multiple lighting passes for a particular CG element (beauty, fill, backlight, reflection, shadow, etc.) as an .exr sequence, which you could then read into a Nuke script, or comp. You would be able to access all of the render passes stored within the .exr sequence from any downstream node in your script.

You might choose to color correct only the CG element’s highlights by using the specular pass as a mask to a particular color correction operator. Such an approach again has the advantage of keeping the Nuke comp free of unnecessarily complex branching - virtually all render passes and mattes can be passed through a single pipe in the comp.
The **Channels** chapter explains how to take full advantage of the 1023-channel workflow.

### 8-, 16-, and 32-Bit Image Processing

Some digital compositing systems, especially those geared for video work, are optimized for processing exclusively 8-bit elements (that is, images with 256 intensity values per channel). Other systems allow for the mixing of 8, 16, and 32-bit elements.

For Nuke products, which began as a film effects tool, image quality is paramount. Thus, they support the processing of exclusively 32-bit-per channel elements (elements with lower bit depths are converted to 32 bits per channel upon import). Thirty-two bit support allows for a much richer palette of colors and floating point precision in all script calculations. In practice, this means that Nuke products carry out every operation - from an increase in gamma to a transform - with much greater accuracy than a lower-bit-depth system.

### Compositing in 3D

Some digital compositing systems support a strictly two-dimensional workflow. Nuke products, by contrast, offer a robust 3D workspace that lets you create and render complex scenes composed of polygonal models, cards (planes textured with images), cameras, lights, and textures.

This 3D workspace has countless uses, the simplest of which is generating pan-and-tile scenes. These are scenes with 2D image planes arranged into a curved shape, and then rendered out through an animated camera to give the illusion of a seamless environment.

The **3D Compositing** chapter explains how to make full use of Nuke's 3D workspace.
Render Farms and Frame Servers

Nuke products support virtually all third-party and proprietary render-queuing software. By integrating Nuke products with such a system, the render load can be distributed across all the Nuke- or NukeX-licensed machines on your network, whether Windows, Mac, or Linux-based. See Render Farms for more information.

In addition, Nuke Studio ships with an internal Frame Server, which also allows you to setup external slave machines to process renders faster. See Using the Frame Server on External Machines for more information.

**Note:** Nuke Assist does not support Write nodes, render farms, or the Frame Server. See Nuke Products for more information.
Installation and Licensing

We know the installation and licensing of a new application can be a boring task that you just want to be done with as soon as possible. To help you with that, this chapter guides you to the point where you have the application in front of you and are ready to start work.

Operating Systems

To see the installation and licensing instructions for your operating system, go to:

- **Windows**, 
- **Mac OS X and macOS**, or
- **Linux**.

**Note:** For more information on the differences between the Nuke applications, see Nuke Products.
Windows

These pages guide you through installation and licensing to the point where you have the application in front of you and are ready to start work. After installation, all applications are run from either desktop icons, the browser, or from the command line using arguments.

System Requirements

Qualified Operating Systems

- Windows 7 64-bit
- Windows 10 64-bit

Note: Other operating systems may work, but have not been fully tested.

Minimum Hardware Requirements

- x86-64 processor, such as Intel Pentium 4 or AMD Athlon, with SSE3 instruction set support (or newer).
- 5 GB disk space available for caching and temporary files.
- At least 8 GB of RAM.
- Display with at least 1280 x 1024 pixel resolution and 24-bit color.
- Graphics card with at least 512 MB of video memory and driver support for OpenGL 2.0 (minimum requirement).
  - To enable optional GPU acceleration of Viewer processing, you need OpenGL 2.0 with support for floating point textures and GLSL.
  - To enable Nuke to calculate certain nodes using the GPU, there are some additional requirements. For more information, see Requirements for GPU Acceleration.
- R3D Rocket cards require the Rocket Driver 1.4.19.0 and Firmware 1.1.16.5 or later.

Note: To avoid graphical problems, such as text disappearing in the Viewer and Node Graph, it is important to keep your graphics card drivers up-to-date. Driver updates can be
Requirements for GPU Acceleration

If you want to enable Nuke to calculate certain nodes using the GPU, there are some additional requirements.

**NVIDIA**

- An NVIDIA GPU with compute capability 2.0 (Fermi) or above. A list of the compute capabilities of NVIDIA GPUs is available at [www.nvidia.co.uk/object/cuda_gpus uk.html](http://www.nvidia.co.uk/object/cuda_gpus uk.html)

  **Note:** The compute capability is a property of the GPU hardware and can’t be altered by a software update.

- Graphics drivers capable of running CUDA 8.0 & 6.5 or above. These are bundled with the regular drivers for your NVIDIA GPU. Driver version r361 or above is required.


  **Note:** If your computer enters sleep mode, the CUDA drivers cannot recover and you must restart Nuke to use GPU acceleration.

Note: obtained from the websites of the graphics card manufacturers (for example, [www.nvidia.com](http://www.nvidia.com) and [support.amd.com](http://support.amd.com)).

**Note:** If you’re using R3D Rocket graphics card, note that using it with Foundry applications will most likely only be considerably faster when you’re reading in at full resolution. If you’re reading in at half resolution, for instance, disabling the R3D Rocket card may be faster. This is because the R3D Rocket graphics card is designed to be fast when reading in multiple frames at the same time. This is not how Nuke works internally, and therefore reads with the R3D Rocket card disabled may sometimes be faster when working in lower resolutions (< 4K widths). Note that the R3D Rocket card always produces better results than Nuke when downsampling. Also, the R3D Rocket card can only be used by one application at a time, so if you are viewing multiple Nuke comps at once, you may only be able to use the R3D Rocket card in one.
Tip: We recommend using the latest graphics drivers, where possible.

AMD

An AMD GPU and recommended driver from the following list:

<table>
<thead>
<tr>
<th>Card</th>
<th>Driver</th>
</tr>
</thead>
<tbody>
<tr>
<td>AMD FirePro W8100</td>
<td>17.Q2.1</td>
</tr>
<tr>
<td>AMD FirePro W9100</td>
<td>17.Q2.1</td>
</tr>
<tr>
<td>AMD Radeon R9 Fury X</td>
<td>17.4.3 - 17.6.2</td>
</tr>
<tr>
<td>AMD Radeon RX 480</td>
<td>17.Q2.1</td>
</tr>
<tr>
<td>AMD Radeon Pro WX 7100</td>
<td>17.4.3 - 17.6.2</td>
</tr>
</tbody>
</table>

Note: Other AMD GPUs may work, but have not been fully tested.

Multi-GPU Processing

Nuke's GPU support includes an Enable multi-GPU support option. When enabled in the preferences, GPU processing is shared between the available GPUs for extra processing speed.

Note: Multi-GPU processing is only available for identical GPUs in the same machine. For example, two NVIDIA GeForce GTX 1080s or two AMD FirePro W9100s.
Installing on Windows

The installation bundle installs the entire Nuke family, including Hiero and HieroPlayer, and icons for the various components appear in your installation folder.

**Note:** Some modern anti-virus software may wrongly assume that certain files in the Nuke installer are suspicious. Examples of these files include `libnuke-12.0.0.so` and `geolib-runtime-prof.so`. If you have trouble installing Nuke on your machine, try disabling your anti-virus software before installation. Don't forget to restart your anti-virus software after installation.

Installing with the User Interface (UI)

1. Download the correct `.zip` file from our website at [wwwFOUNDRY.com](http://www.foundry.com).
2. Unzip the installer from the `.zip` file and double-click on the `.exe` file to install Nuke.
4. The Nuke plug-ins page on our website opens, giving you easy access to a large selection of plug-ins to use with Nuke.
5. Proceed to Licensing on Windows.

**Note:** On Windows, if you install Nuke to a network drive to run from multiple computers, please ensure that the correct Microsoft run time libraries are installed on each machine that runs Nuke. To do this, run `vcredist_x64.exe` on each machine. The appropriate one of these files can be found in the `VCRedist` subdirectory in the folder where Nuke is installed.

Running Nuke without installing the libraries on your machine may work correctly, particularly as many systems (such as Windows Vista by default) already have them. If the libraries are not present, Nuke can still run correctly, but some plug-ins may fail to load with error messages such as “This application has failed to start because the application configuration is incorrect. Reinstalling the application may fix this problem.”

Please note that these libraries are set up automatically on the machine that runs the Nuke installer, so users installing on their local machine do not need to worry about this issue.
Installing from the Command Line

To install the Nuke bundle from the command line, do the following:

1. Download the correct .exe installation file from our website at www.foundry.com.
2. To open a command prompt window, select Start > All Programs > Accessories > Command Prompt.
3. Use the cd (change directory) command to move to the directory where you saved the installation file. For example, if you saved the installation file in C:\Temp, use the following command and press Return:
   cd \Temp
4. To install Nuke, do one of the following:
   - To install Nuke to the current directory and display the installation dialog, type the name of the install file without the file extension and press Return:
     Nuke11.3v6-win-x86-release-64
   - To install Nuke to a specified directory and display the installation dialog, use the /dir install option:
     Nuke11.3v6-win-x86-release-64 /dir="C:\Nuke"
   - To install Nuke silently so that the installer does not prompt you for anything but displays a progress bar, enter /silent after the installation command:
     Nuke11.3v6-win-x86-release-64 /silent
   - To install Nuke silently so that nothing is displayed, enter /verysilent after the installation command:
     Nuke11.3v6-win-x86-release-64 /verysilent

   **Tip:** You can add /MERGETASKS="!desktopicon" to the /verysilent flag to avoid creating Desktop icons for all the flavors of Nuke.

   • You can also use a combination of install options:
     Nuke11.3v6-win-x86-release-64 /silent /dir="C:\Nuke"

   **Note:** By using the silent or verysilent install options, you agree to the terms of the Nuke End User Licensing Agreement. To see this agreement, please refer to Foundry's [End User License Agreement](#) or run the installer in standard, non-silent mode.
Launching on Windows

To launch the application on Windows, do one of the following:

- Double-click the required icon on the Desktop.
- Navigate to Start > All Programs > The Foundry > Nuke11.3v6 and select the required application.
- Using a command prompt, navigate to the Nuke application directory (by default, %ProgramFiles%\Nuke11.3v6) and enter:
  - `Nuke11.3.exe --nukex` to launch NukeX.
  - `Nuke11.3.exe --hiero` to launch Hiero.
  - `Nuke11.3.exe --player` to launch HieroPlayer.

**Note:** Nuke Assist licenses are only available as part of the NukeX or Nuke Studio package, and cannot be purchased separately. For more information, see About Nuke Assist.

**Tip:** For more information on other command prompt options, such as safe mode, see Command Line Operations.

If you already have a valid license, the graphical interface appears, and a command line window opens. If you don't have a license or haven't installed one yet, proceed to Licensing on Windows.

Nuke Analytics

In an effort to further improve quality and reliability, we ask you to allow us to collect usage statistics from the machines on which you license Nuke, NukeX, Nuke Studio, Hiero, and HieroPlayer. This usage information also assists our Support team to resolve issues more quickly.

**Note:** The port number used to communicate with Foundry is 443, the same one used for uploading crash reports.
The first time you start an application, and on every major release, a dialog displays asking for permission for us to collect this information. You can enable or disable collection at any time in the Preferences under Behaviors > Startup.

Note: This information is only collected for interactive sessions. Running applications in terminal mode or under render licenses does not upload data to Foundry.

The following list shows the information we'll collect, if you give us permission to do so:

- Unique session ID
- Application name
- If the session exited cleanly
- Operating system
- CPU Name
- Amount of GPU RAM
- Amount of RAM
- Anonymous user key
- Session start time (GMT)
- Peak memory usage
- System OS version
- CPU Cores
- OpenGL driver version
- Application version string
- Session duration (in seconds)
- Model
- MAC address
- GPU model name
- GPU driver version

Nuke Non-commercial

If you want to try out or learn Nuke, you can run Nuke Non-commercial. This version allows you to explore most of Nuke's features, but prevents the commercial use of the application. For more information, see About Nuke Non-commercial.

To launch the application on Windows, do one of the following:

- Navigate to Start > All Programs > The Foundry > Nuke11.3v6 and select Nuke11.3v6NC, NukeX11.3v6NC, or NukeStudio11.3v6NC.
- Using a command prompt, navigate to the Nuke application directory and enter:
  
  - Nuke11.3.exe --nc --nukex to launch NukeX.

If you have already activated Nuke Non-commercial on the current device, the graphical interface appears, and a command line window opens. If you haven't activated the device yet, proceed to Licensing Nuke Non-commercial on Windows.
Licensing on Windows

The following licensing methods are available:

- **Activation Keys** - activation keys allow you to activate and generate your actual product license key, at a later point after purchase, on the machine for which you require the license.

They are provided for both node locked and floating licenses, and generate the appropriate license type once installed using the product's Licensing dialog or online using the Activate a Product page: https://www.foundry.com/licensing/activate-product

- **Node Locked Licenses** - these can be used to license an application on a single machine. They do not work on different machines and if you need them to, you'll have to transfer your license.

Node locked licenses, sometimes called uncounted licenses, do not require additional licensing software to be installed.

- **Floating Licenses** - also known as counted licenses, enable applications to work on any networked client machine. The floating license is put on the server and is locked to a unique number on that server.

Floating licenses on a server requires additional software to be installed on the server to manage the licenses and give them out to the client stations that want them. This software is called the Foundry Licensing Tools (FLT) and can be downloaded at no extra cost from our website.

- **Subscription Licenses** - subscription licensing differs from traditional node locked or floating licenses in that a single license, or entitlement, is valid on any authorized device up to the entitlement's maximum number of activations.

For more information on Subscription Licensing, see Licensing Nuke Non-commercial on Windows.

The following instructions run through the basic options for the first two licensing methods, but you can find a more detailed description in the Foundry Licensing Tools (FLT) User Guide available on our website: https://www.foundry.com/licensing

Obtaining Licenses

To obtain a license, you'll need your machine's System ID (sometimes called Host ID or rlmhostid). Just so you know what a System ID number looks like, here's an example: 000ea641d7a1.
Note: Bear in mind that, for floating licenses, you'll need the System ID of the license server, not the machines on which you intend to run the application.

There are a number of ways you can find out your machine's System ID:

- Launch the application without a license, click **Status**, and then scroll down the error report until you see your System ID.
- Download the Foundry License Utility (FLU) from [https://www.foundry.com/licensing](https://www.foundry.com/licensing) and run it. Your System ID is displayed.
- Download the Foundry Licensing Tools (FLT) free of charge from [https://www.foundry.com/licensing](https://www.foundry.com/licensing) and then run `C:\Program Files\TheFoundry\LicensingTools7.0\Foundry License Utility.exe`

When you know your System ID, you can request a license for Foundry products:

- from Foundry's Sales Department at [sales@foundry.com](mailto:sales@foundry.com)
- from the product pages on our website, such as [https://www.foundry.com/products/nuke](https://www.foundry.com/products/nuke)
- by launching the application without a license and selecting:
  - **Buy Nuke** or **Buy Hiero** - opens a web browser directly to our website to purchase a license.
  - **Try Nuke** or **Try Hiero** - displays the 15-day trial license download screen. Enter your Foundry account details or create a new account and follow the on-screen instructions to receive a trial license.

Note: By default, if you have installed a temporary license, the application displays a dialog at start-up alerting you to the number of days remaining. If you want to disable this behavior, you can set the FN_DISABLE_LICENSE_DIALOG environment variable to 1 to suppress the warning message about imminent license expiration. See [Environment Variables](#) for more information.

## Installing Licenses

When you start the application before installing a license, a Licensing dialog displays an error informing you that no license was available. The installation process is dependent on what type of license you requested:

- **License file** - if you requested a license file, typically `foundry.lic`, this option allows you to browse to the file location and install it automatically. See [To Install a License from Disk](#) for more information.
• **Activation Key or license text** - if you requested an Activation Key or license by email, this option allows you to paste the key or license text into the Licensing dialog, which then installs the license in the correct directory. See To Install an Activation Key or License Text for more information.

• **A floating license** - if you requested a floating license to supply licenses to multiple client machines, this option allows you to enter the server address that supplies the client licenses.

![Note: You must install a floating license and additional software on the license server to use this option.](image)

See To Install a Floating License for more information.

### To Install a License from Disk

1. Save the license file to a known location on disk.
2. Launch the application.
   
   The **Licensing** dialog displays.
3. Click **Install License** to display the available license installation options.
4. Click **Install from Disk**.
5. Browse to the location of the license file.
6. Click **Open** to install the license automatically in the correct directory.

### To Install an Activation Key or License Text

1. Launch the application.
   
   The **Licensing** dialog displays.
2. Click **Install License** to display the available license installation options.
3. Click **Activation Key / License Text** and then either:
   
   • Enter the Activation Key string in place of Insert Activation Key Here. A license key typically looks something like this:
     
     nuke-0101-77d3-99bd-a977-93e9-8035

   OR

   • Copy the license text and paste it over the Copy/Paste license text here string. License text typically looks something like this:
     
     LICENSE foundry nuke_i 2015.0929 29-sep-2015 uncounted
     
     hostid=000a957bfde5 share=h min_timeout=30 start=29-sep-2015 issued=29-sep-2015 disable=VM _ck=da32d7372f sig="60P0450MJRP97E3DPB42C99Y5UAPREMJQ39PG22H4WGH3WFK2KPTXFWJTYR0GYASJBC0PU8"

4. Click **Install**.
The license is automatically installed on your machine in the correct directory.

**Note:** Activation Keys require an internet connection. If you access the internet through a proxy server and cannot connect to the activation server, you may get an error dialog prompting you to either:

Click **Use Proxy** to enter the proxy server name, port number, username, and password. This enables the application to connect to the activation server and obtain a license. The license is then installed automatically, or

Click on the web link in the dialog and use the System ID (also known as hostid) provided to manually activate and install a license.

### To Install a Floating License

If you requested a floating license from Foundry, you will receive your license key (foundry.lic) in an email or internet download. You should also receive the Foundry License Utility (FLU) application to help you install the license key on the license server machine. The server manages licenses for the client machines on your network.

**Note:** The FLU is also available to download from https://www.foundry.com/licensing

1. Make sure you have saved both the license key (foundry.lic) and the FLU application in the same directory.
2. Run the FLU application.
   - The license key automatically appears in the FLU window if the FLU and **foundry.lic** are in the same directory.

**Tip:** If they are not in the same directory, you can either copy and paste the contents of the license key or drag-and-drop the file into the FLU window.

3. Click **Install**.
   - This checks the license file and, provided that the license is valid, installs it into the correct directory.
4. In order for the floating license to work, you will need to install the Foundry Licensing Tools (FLT) on the license server machine.
For more information on how to install floating licenses, refer to the *FLT User Guide*, which you can download from our website [https://www.foundry.com/licensing](https://www.foundry.com/licensing)

5. Once your license server is up and running, launch the application on the client machine. The **Licensing** dialog displays.

6. Click **Install License** to display the available install methods.

7. Click **Use Server** and enter the server address in the field provided. The format for the server name is: `<port>@<servername>`, for example, `30001@red`.

**Note:** You must perform steps 5 through 7 on each client machine that requires a license from the server.

### Further Reading

There is a lot to learn about licenses, much of which is beyond the scope of this manual. For more information on licensing, displaying the System ID number, setting up a floating license server, adding new license keys and managing license usage across a network, you should read the Foundry Licensing Tools (FLT) User Guide, which can be downloaded from our website, [https://www.foundry.com/licensing](https://www.foundry.com/licensing)
Licensing Nuke Non-commercial on Windows

Subscription licensing differs from traditional node locked or floating licenses in that a single license, or entitlement, is valid on any authorized device up to the entitlement’s maximum number of activations.

• An **Entitlement** represents the right to run a Foundry product for a set amount of time on a set number of devices.

• An **Authorized Device** is a recognized device, such as a desktop computer, on which entitlements can be activated.

For example, if an **Entitlement** for Nuke has five activations, you can use Nuke on five separate **Authorized Devices** simultaneously. If you want to activate another device, you have to deactivate an existing one, but you can activate and deactivate devices as often as you like.

To get started with Nuke Non-commercial, follow these steps:

1. Create a Foundry account using a valid email address on our website, [https://www.foundry.com/user/register](https://www.foundry.com/user/register)
2. Launch Nuke in non-commercial mode as described under **Launching on Windows**. A **Licensing** dialog displays, similar to regular licensing. Nuke Non-commercial is free, but your entitlement only contains two activations.
3. Click **Authorise Device**.
4. Enter your account email address and password and then click **Authorise Device**.
5. A subscription license is created in your home directory: `C:\Users\<username>\FoundryLicensing\<SystemID>`

**Note:** Replace `<username>` and `<SystemID>` with the current user and the MAC address of the device, respectively.

The license looks something like this: `c58edf7e-17ab-435b-8d8a-b3a9b347ab11.lic`

6. Once the license is installed, click **Launch** to start using Nuke.

**Note:** On Windows, there is a known issue with user names containing non-ASCII characters causing licensing to fail. If a licensing error similar to the following displays:

```plaintext
Unable to create subscription license directory: C:\Users\Zoë
```
Hernández\FoundryLicensing\n
Try changing the license directory to an alternate location using the FN_SUBSCRIPTION_LICENSE_DIR environment variable. See Environment Variables for more information.

7. If you need to deactivate an entitlement or deauthorize a device, navigate to Help > License and, click:
   - **Deactivate Nuke** to reclaim one of your entitlements,
   - **Deauthorize Device** to reclaim your existing Foundry entitlements on this device and stop additional ones running, or
   - **Deauthorize All Devices** to reclaim your existing Foundry entitlements on all devices associated with your account, and stop additional ones running.
Uninstalling Apps on Windows

To uninstall Nuke or Hiero on Windows, there are a few things you need to do:

2. Click Yes to uninstall.
3. Delete, rename, or move your .nuke or .hiero directories, if they exist.
   The .nuke and .hiero directories are usually found under the directory pointed to by the HOME environment variable. If this variable is not set (which is common), the directories are under the directory specified by the USERPROFILE environment variable, which is generally one of the following:
   
   drive letter:\Documents and Settings\login name\  
   drive letter:\Users\login name\  

   To find out if the HOME and USERPROFILE environment variables are set and where they are pointing at, enter %HOME% or %USERPROFILE% into the address bar in Windows Explorer. If the environment variable is set, the folder it’s pointing at is opened. If it’s not set, you get an error.
4. Delete, rename, or move your cached files, which reside in the following directory by default:
   ~\AppData\Local\Temp\nuke\  

   Where ~ is equal to %HOME% or %USERPROFILE% as detailed above.

Note: If you specified an alternate directory using the NUKE_TEMP_DIR environment variable, purge those files as well as the default location. See Nuke Environment Variables for more information.
Mac OS X and macOS

These pages guide you through installation and licensing to the point where you have the application in front of you and are ready to start work. After installation, all applications are run from either desktop icons, the browser, or from the command line using arguments.

System Requirements

Qualified Operating Systems

• macOS 10.13 (High Sierra)
• macOS 10.14 (Mojave)

Note: Other operating systems may work, but have not been fully tested.

Minimum Hardware Requirements

• x86-64 processor, such as Intel Core 2 Duo or later.
• 5 GB of disk space available for caching and temporary files.
• At least 8 GB of RAM.
• Display with at least 1280 x 1024 pixel resolution and 24-bit color.
• Graphics card with at least 512 MB of video memory and driver support for OpenGL 2.0 (minimum requirement).
  • To enable optional GPU acceleration of Viewer processing, you need OpenGL 2.0 with support for floating point textures and GLSL.
  • To enable Nuke to calculate certain nodes using the GPU, there are some additional requirements. For more information, see Requirements for GPU Acceleration.
• R3D Rocket cards require the Rocket Driver 1.4.19.0 and Firmware 1.1.16.5 or later.

Note: To avoid graphical problems, such as text disappearing in the Viewer and Node Graph, it is important to keep your graphics card drivers up-to-date. Driver updates can be
obtained from the websites of the graphics card manufacturers (for example, www.nvidia.com and support.amd.com).

**Note:** If you’re using R3D Rocket graphics card, note that using it in Nuke will most likely only be considerably faster when you’re reading in at full resolution. If you’re reading in at half resolution, for instance, using Nuke without the R3D Rocket card enabled may be faster. This is because the R3D Rocket graphics card is designed to be fast when reading in multiple frames at the same time. This is not how Nuke works internally, and therefore reads with the R3D Rocket card disabled may sometimes be faster when working in lower resolutions (< 4K widths). Note that the R3D Rocket card always produces better results than Nuke when downsampling. Also, the R3D Rocket card can only be used by one application at a time, so if you are viewing multiple Nuke scripts at once, you may only be able to use the R3D Rocket card in one.

**Requirements for GPU Acceleration**

If you want to enable Nuke to calculate certain nodes using the GPU, there are some additional requirements.

**NVIDIA**

An NVIDIA GPU with compute capability 2.0 (Fermi) or above. A list of the compute capabilities of NVIDIA GPUs is available at: www.nvidia.co.uk/object/cuda_gpus_uk.html.

**Note:** The compute capability is a property of the GPU hardware and can't be altered by a software update.

Graphics drivers capable of running CUDA 8.0 & 6.5 or above. On Mac, the CUDA driver is separate from the NVIDIA graphics driver and will need to be installed, if you don't have it already. The minimum requirement is driver version r361 which can be downloaded from www.nvidia.com/drivers.
**Note:** If your computer enters sleep mode, the CUDA drivers cannot recover and you must restart Nuke to use GPU acceleration.

**Tip:** We recommend using the latest graphics drivers, where possible.

**AMD**

On Mac, AMD GPUs are supported on any Mac Pro running Mac OS X Mavericks (10.9.3), mid 2015 MacBook Pros onward, and late 2017 iMac Pros. Bit-wise equality between GPU and CPU holds in most cases, but for some operations there are limitations to the accuracy possible with this configuration.

**Warning:** Although AMD GPUs are enabled on Mac Pros manufactured prior to the late 2013 model, they are not officially supported and used at your own risk.

**Note:** To ensure you get the best performance from OpenCL GPUs, we recommend updating Mavericks to 10.9.5, or above for full functionality. However:
If you're running an earlier version of Mac OS X than 10.9.5 and processing images greater than 4 mega pixels resolution, VectorGenerator, Kronos, and MotionBlur do not support GPU acceleration.
If you're running an earlier version of Mac OS X than 10.9.4, Kronos and MotionBlur do not support GPU acceleration.

**Multi-GPU Processing**

Nuke's GPU support includes an **Enable multi-GPU support** option. When enabled in the preferences, GPU processing is shared between the available GPUs for extra processing speed.

**Note:** Multi-GPU processing is only available for identical GPUs in the same machine. For example, two NVIDIA GeForce GTX 1080s or two AMD FirePro W9100s.
Installing on Mac

The installation bundle installs the entire Nuke family, including Hiero and HieroPlayer, and icons for the various components appear in your installation folder.

**Note:** Some modern anti-virus software may wrongly assume that certain files in the Nuke installer are suspicious. Examples of these files include *libnuke-12.0.0.so* and *geolib-runtime-prof.so*. If you have trouble installing Nuke on your machine, try disabling your anti-virus software before installation. Don't forget to restart your anti-virus software after installation.

Installing with the User Interface (UI)

1. Download the correct `.dmg` installation file from our website at [www.foundry.com](http://www.foundry.com).
2. Double-click on the Nuke11.3v6-mac-x86-release-64.dmg archive to extract the `.pkg` installer: A `.pkg` file is created.
5. The Nuke plug-ins page on our website opens, giving you easy access to a large selection of plug-ins to use with Nuke.
6. Proceed to [Launching on Mac](#).

Installing from the Terminal

1. Download the correct `.dmg` installation file from our website at [www.foundry.com](http://www.foundry.com).
2. Launch a Terminal window.
3. To mount the `.dmg` installation file, use the `hdiutil attach` command with the directory where you saved the installation file. For example, if you saved the installation file in **Builds/Nuke**, use the following command:
   ```bash
   hdiutil attach /Builds/Nuke/Nuke11.3v6-mac-x86-release-64.dmg
   ```
4. Enter the following command:
   ```bash
   pushd /Volumes/Nuke11.3v6/
   ```
   This stores the directory path in memory, so it can be returned to later.
5. To install Nuke, use the following command:
sudo installer -pkg Nuke11.3v6-mac-x86-release-64.pkg -target "/"
You are prompted for a password.

6. Enter the following command:
   popd
   This changes to the directory stored by the pushd command.

7. Finally, use the following command to eject the mounted disk image:
   hdiutil detach /Volumes/Nuke11.3v6

**Note:** By running a silent install of Nuke, you agree to the terms of the End User Licensing Agreement. To see this agreement, please refer to Foundry's End User License Agreement or run the installer in standard, non-silent mode.
Launching on Mac

To launch the application on Mac, do one of the following:

• Double-click the required icon on the Desktop.
• Open the Nuke application directory (/Applications/Nuke11.3v6/), and double-click the required icon.
• Using the terminal, navigate to the Nuke application directory (/Applications/Nuke11.3v6/Nuke11.3v6.app/Contents/MacOS/) and enter:
  • ./Nuke11.3v6 --studio to launch Nuke Studio.
  • ./Nuke11.3v6 --nukex to launch NukeX.
  • ./Nuke11.3v6 to launch Nuke.
  • ./Nuke11.3v6 --hiero to launch Hiero.
  • ./Nuke11.3v6 --player to launch HieroPlayer.
  • ./Nuke11.3v6 --nukeassist to launch Nuke Assist.

Note: Nuke Assist licenses are only available as part of the NukeX or Nuke Studio package, and cannot be purchased separately. For more information, see About Nuke Assist.

Tip: For more information on other terminal options, such as safe mode, see Command Line Operations.

If you already have a valid license, the graphical interface appears, and a command line window opens. If you don't have a license or haven't installed one yet, proceed to Licensing on Mac.

Nuke Analytics

In an effort to further improve quality and reliability, we ask you to allow us to collect usage statistics from the machines on which you license Nuke, NukeX, Nuke Studio, Hiero, and HieroPlayer. This usage information also assists our Support team to resolve issues more quickly.

Note: The port number used to communicate with Foundry is 443, the same one used for uploading crash reports.
The first time you start an application, and on every major release, a dialog displays asking for permission for us to collect this information. You can enable or disable collection at any time in the Preferences under Behaviors > Startup.

**Note:** This information is only collected for interactive sessions. Running applications in terminal mode or under render licenses does not upload data to Foundry.

The following list shows the information we'll collect, if you give us permission to do so:

- Unique session ID
- Application name
- If the session exited cleanly
- Operating system
- CPU Name
- Amount of GPU RAM
- Amount of RAM
- Anonymous user key
- Session start time (GMT)
- Peak memory usage
- System OS version
- CPU Cores
- OpenGL driver version
- Application version string
- Session duration (in seconds)
- Model
- MAC address
- GPU model name
- GPU driver version

## Nuke Non-commercial

If you want to try out or learn Nuke, you can run Nuke Non-commercial. This version allows you to explore most of Nuke's features, but prevents the commercial use of the application. For more information, see About Nuke Non-commercial.

To launch the application on Mac, do one of the following:

- Double-click the NukeNC, NukeXNC, or Nuke StudioNC dock icon.
- Open the Nuke application directory and double-click the NukeNC, NukeXNC, or Nuke StudioNC icon.
- Using the terminal, navigate to the Nuke application directory and enter:
  
  - */Nuke11.3v6 --nc --nukex* to launch NukeX.

If you have already activated Nuke Non-commercial on the current device, the graphical interface appears, and a command line window opens. If you haven't activated the device yet, proceed to Licensing Nuke Non-commercial on Mac.
Licensing on Mac

The following licensing methods are available:

- **Activation Keys** - activation keys allow you to activate and generate your actual product license key, at a later point after purchase, on the machine for which you require the license.

They are provided for both node locked and floating licenses, and generate the appropriate license type once installed using the product's Licensing dialog or online using the Activate a Product page: https://www.foundry.com/licensing/activate-product

- **Node Locked Licenses** - these can be used to license an application on a single machine. They do not work on different machines and if you need them to, you’ll have to transfer your license.

Node locked licenses, sometimes called uncounted licenses, do not require additional licensing software to be installed.

- **Floating Licenses** - also known as counted licenses, enable applications to work on any networked client machine. The floating license is put on the server and is locked to a unique number on that server.

Floating licenses on a server requires additional software to be installed on the server to manage the licenses and give them out to the client stations that want them. This software is called the Foundry Licensing Tools (FLT) and can be downloaded at no extra cost from our website.

- **Subscription Licenses** - subscription licensing differs from traditional node locked or floating licenses in that a single license, or entitlement, is valid on any authorized device up to the entitlement's maximum number of activations.

For more information on Subscription Licensing, see Licensing Nuke Non-commercial on Mac.

The following instructions run through the basic options for the first two licensing methods, but you can find a more detailed description in the Foundry Licensing Tools (FLT) User Guide available on our website: https://www.foundry.com/licensing

Obtaining Licenses

To obtain a license, you'll need your machine's System ID (sometimes called Host ID or rlmhostid). Just so you know what a System ID number looks like, here's an example: 000ea641d7a1.
Note: Bear in mind that, for floating licenses, you'll need the System ID of the license server, not the machines on which you intend to run the application.

There are a number of ways you can find out your machine’s System ID:

- Launch the application without a license, click Status, and then scroll down the error report until you see your System ID.
- Download the Foundry License Utility (FLU) from https://www.foundry.com/licensing and run it. Your System ID is displayed.
- Download the Foundry Licensing Tools (FLT) free of charge from https://www.foundry.com/licensing and then run /Applications/TheFoundry/LicensingTools7.0/Foundry Licence Utility.app

When you know your System ID, you can request a license for Foundry products:

- from Foundry's Sales Department at sales@foundry.com
- from the product pages on our website, such as https://www.foundry.com/products/nuke
- by launching the application without a license and selecting:
  - Buy Nuke or Buy Hiero - opens a web browser directly to our website to purchase a license.
  - Try Nuke or Try Hiero - displays the 15-day trial license download screen. Enter your Foundry account details or create a new account and follow the on-screen instructions to receive a trial license.

Note: By default, if you have installed a temporary license, Nuke displays a dialog at start-up alerting you to the number of days remaining. If you want to disable this behavior, you can set the FN_DISABLE_LICENSE_DIALOG environment variable to 1 to suppress the warning message about imminent license expiration. See Environment Variables for more information.

Installing Licenses

When you start the application before installing a license, a Licensing dialog displays an error informing you that no license was available. The installation process is dependent on what type of license you requested:

- License file - if you requested a license file, typically foundry.lic, this option allows you to browse to the file location and install it automatically. See To Install a License from Disk for more information.
• **Activation Key or license text** - if you requested an Activation Key or license by email, this option allows you to paste the key or license text into the Licensing dialog, which then installs the license in the correct directory. See To Install an Activation Key or License Text for more information.

• **A floating license** - if you requested a floating license to supply licenses to multiple client machines, this option allows you to enter the server address that supplies the client licenses.

   **Note:** You must install a floating license and additional software on the license server to use this option.

See To Install a Floating License for more information.

### To Install a License from Disk

1. Save the license file to a known location on disk.
2. Launch the application.
   - The Licensing dialog displays.
3. Click Install License to display the available license installation options.
4. Click Install from Disk.
5. Browse to the location of the license file.
6. Click **Open** to install the license automatically in the correct directory.

### To Install an Activation Key or License Text

1. Launch the application.
   - The Licensing dialog displays.
2. Click Install License to display the available license installation options.
3. Click **Activation Key / License Text** and then either:
   • Enter the Activation Key string in place of Insert Activation Key Here. A license key typically looks something like this:
     
     nuke-0101-77d3-99bd-a977-93e9-8035

     OR

   • Copy the license text and paste it over the Copy/Paste license text here string. License text typically looks something like this:
     
     LICENSE foundry nuke_i 2015.0929 29-sep-2015 uncounted
     hostid=000a957bfde5 share=h min_timeout=30 start=29-sep-2015 issued=29-sep-2015 disable=VM ck=da32d7372f sig="60P0450MJRP97E3DPB42C99Y5UAPREMNGNQ39PG22H4WGH3WFK2KPTXFWJTYR0GYASJBC0PU8"

4. Click Install.
The license is automatically installed on your machine in the correct directory.

**Note:** Activation Keys require an internet connection. If you access the internet through a proxy server and cannot connect to the activation server, you may get an error dialog prompting you to either:

Click **Use Proxy** to enter the proxy server name, port number, username, and password. This enables the application to connect to the activation server and obtain a license. The license is then installed automatically, or

Click on the web link in the dialog and use the System ID (also known as hostid) provided to manually activate and install a license.

**To Install a Floating License**

If you requested a floating license from Foundry, you will receive your license key (foundry.lic) in an email or internet download. You should also receive the Foundry License Utility (FLU) application to help you install the license key on the license server machine. The server manages licenses for the client machines on your network.

**Note:** The FLU is also available to download from [https://www.foundry.com/licensing](https://www.foundry.com/licensing)

1. Make sure you have saved both the license key (foundry.lic) and the FLU application in the same directory.
2. Run the FLU application.
   - The license key automatically appears in the FLU window if the FLU and foundry.lic are in the same directory.

**Tip:** If they are not in the same directory, you can either copy and paste the contents of the license key or drag-and-drop the file into the FLU window.

3. Click **Install**.
   - This checks the license file and, provided that the license is valid, installs it into the correct directory.
4. In order for the floating license to work, you will need to install the Foundry Licensing Tools (FLT) on the license server machine.
For more information on how to install floating licenses, refer to the FLT User Guide, which you can download from our website https://www.foundry.com/licensing

5. Once your license server is up and running, launch the application on the client machine. The Licensing dialog displays.

6. Click Install License to display the available install methods.

7. Click Use Server and enter the server address in the field provided. The format for the server name is: <port>@<servername>, for example, 30001@red.

Note: You must perform steps 5 through 7 on each client machine that requires a license from the server.

Further Reading

There is a lot to learn about licenses, much of which is beyond the scope of this manual. For more information on licensing Nuke, displaying the System ID number, setting up a floating license server, adding new license keys and managing license usage across a network, you should read the Foundry Licensing Tools (FLT) User Guide, which can be downloaded from our website, https://www.foundry.com/licensing
Licensing Nuke Non-commercial on Mac

Subscription licensing differs from traditional node locked or floating licenses in that a single license, or entitlement, is valid on any authorized device up to the entitlement’s maximum number of activations.

- An **Entitlement** represents the right to run a Foundry product for a set amount of time on a set number of devices.
- An **Authorized Device** is a recognized device, such as a desktop computer, on which entitlements can be activated.

For example, if an **Entitlement** for Nuke has five activations, you can use Nuke on five separate **Authorized Devices** simultaneously. If you want to activate another device, you have to deactivate an existing one, but you can activate and deactivate devices as often as you like.

To get started with Nuke Non-commercial, follow these steps:

1. Create a Foundry account using a valid email address on our website, [https://www.foundry.com/user/register](https://www.foundry.com/user/register)
2. Launch Nuke in non-commercial mode as described under Launching on Mac.

   A **Licensing** dialog displays, similar to regular licensing. Nuke Non-commercial is free, but your entitlement only contains two activations.
3. Click **Authorise Device**.
4. Enter your account email address and password and then click **Authorise Device**.
5. A subscription license is created in your home directory:

   `/Users/<username>/FoundryLicensing/<SystemID>`

   **Note:** Replace `<username>` and `<SystemId>` with the current user and the MAC address of the device, respectively.

   The license looks something like this: `c58edf7e-17ab-435b-8d8a-b3a9b347ab11.lic`

6. Once the license is installed, click **Launch** to start using Nuke.
7. If you need to deactivate an entitlement or deauthorize a device, navigate to **Help > License** and, click:
   - **Deactivate Nuke** to reclaim one of your entitlements,
• **Deauthorize Device** to reclaim your existing Foundry entitlements on this device and stop additional ones running, or

• **Deauthorize All Devices** to reclaim your existing Foundry entitlements on all devices associated with your account, and stop additional ones running.
Uninstalling Apps on Mac

To uninstall Nuke or Hiero on Mac, there are a few things you need to do:

1. Navigate to Applications and delete the Nuke 11.3v6 directory.
2. Delete, rename, or move your .nuke or .hiero directories, if they exist.
   The directories are found in your home directory, by default:
   /Users/<login name>/.nuke

   **Note:** The directories may be a hidden directory on your machine. To allow your Mac to display hidden files and directories, type the following command in the Terminal application, press Return, and then relaunch the Finder application:
   defaults write com.apple.finder AppleShowAllFiles YES

3. Delete, rename, or move your cached files, which reside in the following directory by default:
   /var/tmp/nuke

   **Note:** If you specified an alternate directory using the NUKE_TEMP_DIR environment variable, purge those files as well as the default location. See Nuke Environment Variables for more information.
Linux

These pages guide you through installation and licensing to the point where you have the application in front of you and are ready to start work. After installation, all applications are run from either desktop icons, the browser, or from the command line using arguments.

System Requirements

Qualified Operating Systems

• CentOS 6 (64-bit)
• CentOS 7 (64-bit)

Note: Other operating systems may work, but have not been fully tested.

Minimum Hardware Requirements

• x86-64 processor, such as Intel Pentium 4 or AMD Athlon, with SSE3 instruction set support (or newer).
• 5 GB disk space available for caching and temporary files.
• At least 8 GB of RAM.
• Display with at least 1280 x 1024 pixel resolution and 24-bit color.
• Graphics card with at least 512 MB of video memory and driver support for OpenGL 2.0 (minimum requirement).
  • To enable optional GPU acceleration of Viewer processing, you need OpenGL 2.0 with support for floating point textures and GLSL.
  • To enable Nuke to calculate certain nodes using the GPU, there are some additional requirements. For more information, see Requirements for GPU Acceleration.
• R3D Rocket cards require the Rocket Driver 1.4.19.0 and Firmware 1.1.16.5 or later.

Note: To avoid graphical problems, such as text disappearing in the Viewer and Node Graph, it is important to keep your graphics card drivers up-to-date. Driver updates can be
obtained from the websites of the graphics card manufacturers (for example, www.nvidia.com and support.amd.com).

**Note:** If you’re using R3D Rocket graphics card, note that using it in Nuke will most likely only be considerably faster when you’re reading in at full resolution. If you’re reading in at half resolution, for instance, using Nuke without the R3D Rocket card enabled may be faster. This is because the R3D Rocket graphics card is designed to be fast when reading in multiple frames at the same time. This is not how Nuke works internally, and therefore reads with the R3D Rocket card disabled may sometimes be faster when working in lower resolutions (< 4K widths). Note that the R3D Rocket card always produces better results than Nuke when downsampling. Also, the R3D Rocket card can only be used by one application at a time, so if you are viewing multiple Nuke scripts at once, you may only be able to use the R3D Rocket card in one.

## Requirements for GPU Acceleration

If you want to enable Nuke to calculate certain nodes using the GPU, there are some additional requirements.

### NVIDIA

- An NVIDIA GPU with compute capability 2.0 (Fermi) or above. A list of the compute capabilities of NVIDIA GPUs is available at:  
  [www.nvidia.co.uk/object/cuda_gpus_uk.html](http://www.nvidia.co.uk/object/cuda_gpus_uk.html)

  **Note:** The compute capability is a property of the GPU hardware and can’t be altered by a software update.

- Graphics drivers capable of running CUDA 8.0 & 6.5 or above. These are bundled with the regular drivers for your NVIDIA GPU. Driver version r361 or above is required.


  **Note:** If your computer enters sleep mode, the CUDA drivers cannot recover and you must restart Nuke to use GPU acceleration.
Tip: We recommend using the latest graphics drivers, where possible.

**AMD**

An AMD GPU and recommended driver from the following list:

<table>
<thead>
<tr>
<th>Card</th>
<th>Driver</th>
</tr>
</thead>
<tbody>
<tr>
<td>AMD FirePro W8100</td>
<td>17.Q2.1</td>
</tr>
<tr>
<td>AMD FirePro W9100</td>
<td>17.Q2.1</td>
</tr>
<tr>
<td>AMD Radeon R9 Fury X</td>
<td>17.10</td>
</tr>
<tr>
<td>AMD Radeon RX 480</td>
<td>17.Q2.1</td>
</tr>
<tr>
<td>AMD Radeon Pro WX 7100</td>
<td>17.10</td>
</tr>
</tbody>
</table>

Note: Other AMD GPUs may work, but have not been fully tested.

**Multi-GPU Processing**

Nuke's GPU support includes an **Enable multi-GPU support** option. When enabled in the preferences, GPU processing is shared between the available GPUs for extra processing speed.

Note: Multi-GPU processing is only available for identical GPUs in the same machine. For example, two NVIDIA GeForce GTX 1080s or two AMD FirePro W9100s.
Installing on Linux

The installation bundle installs the entire Nuke family, including Hiero and HieroPlayer, and icons for the various components appear in your installation folder.

Note: Some modern anti-virus software may wrongly assume that certain files in the Nuke installer are suspicious. Examples of these files include libnuke-12.0.0.so and geolib-runtime-prof.so. If you have trouble installing Nuke on your machine, try disabling your anti-virus software before installation. Don't forget to restart your anti-virus software after installation.

Installing from the Terminal

1. Download the correct .tgz installation file from our website at www.foundry.com.
2. Extract the installer from the .tgz archive with the following terminal command:
   ```bash
tar xvzf Nuke11.3v6-linux-x86-release-64.tgz
   
   This gives you an installer file.
3. Run the installer.
   ```bash
   sudo ./Nuke11.3v6-linux-x86-release-64-installer
   ```
4. Follow the on-screen instructions. By default, Nuke is installed to /usr/local/Nuke11.3v6
5. The Nuke plug-ins page on our website opens, giving you easy access to a large selection of plug-ins to use with Nuke.
6. If you didn't add a license key during the installation, do that now. Proceed to Licensing on Linux.

Tip: To install Nuke silently, you can simply unzip the installer file. This creates the properly formed Nuke directory tree in the current directory.

Note: By installing Nuke silently, you agree to the terms of the End User Licensing Agreement. To see this agreement, please refer to Foundry's End User License Agreement or run the installer in standard, non-silent mode.
Installing Remotely from the Terminal

If you need to install Nuke on render machines using the terminal, do the following:

1. Download the correct .tgz installation file from our website at www.foundry.com.
2. Extract the installer from the .tgz archive with the following terminal command:
   ```bash
tar xvzf Nuke11.3v6-linux-x86-release-64.tgz
   ```
   This gives you an installer file.
3. Use the following terminal command to log in to your render machine as root:
   ```bash
   ssh root@render_machine
   ```
   Replace render_machine with the name of your render node.
4. Make a directory to install Nuke to:
   ```bash
   mkdir /usr/local/Nuke11.3v6
   ```
5. Copy the installer file from the machine that you downloaded it on to your render machine with a command like:
   ```bash
   scp root@download_machine:/tmp/Nuke11.3v6-linux-x86-release-64-installer root@render_machine:/usr/local/Nuke11.3v6/
   ```
   Replace download_machine with the name of the machine you downloaded the installer file to, and render_machine with the name of your render node.
6. Unzip the installer file to unpack its contents into your Nuke directory:
   ```bash
   cd /usr/local/Nuke11.3v6
   unzip Nuke11.3v6-linux-x86-release-64-installer
   ```
7. Repeat steps 3-6 for each render machine.
Launching on Linux

To launch the application on Linux, do one of the following:

• Double-click the required icon on the Desktop.
• Open the Nuke application directory (by default, /usr/local/Nuke11.3v6) and double-click the required icon.
• Using a terminal, navigate to the Nuke application directory and enter:
  • ./Nuke11.3 --studio to launch Nuke Studio.
  • ./Nuke11.3 --nukex to launch NukeX.
  • ./Nuke11.3 to launch Nuke.
  • ./Nuke11.3 --hiero to launch Hiero.
  • ./Nuke11.3 --player to launch HieroPlayer.
  • ./Nuke11.3 --nukeassist to launch Nuke Assist.

Note: Nuke Assist licenses are only available as part of the NukeX or Nuke Studio package, and cannot be purchased separately. For more information, see About Nuke Assist.

Tip: For more information on other terminal options, such as safe mode, see Command Line Operations.

If you already have a valid license, the graphical interface appears. If you don't have a license or haven't installed one yet, proceed to Licensing on Linux.

Nuke Analytics

In an effort to further improve quality and reliability, we ask you to allow us to collect usage statistics from the machines on which you license license Nuke, NukeX, Nuke Studio, Hiero, and HieroPlayer. This usage information also assists our Support team to resolve issues more quickly.

Note: The port number used to communicate with Foundry is 443, the same one used for uploading crash reports.
The first time you start an application, and on every major release, a dialog displays asking for permission for us to collect this information. You can enable or disable collection at any time in the Preferences under Behaviors > Startup.

Note: This information is only collected for interactive sessions. Running applications in terminal mode or under render licenses does not upload data to Foundry.

The following list shows the information we'll collect, if you give us permission to do so:

- Unique session ID
- Application name
- If the session exited cleanly
- Operating system
- CPU Name
- Amount of GPU RAM
- Amount of RAM
- Anonymous user key
- Session start time (GMT)
- Peak memory usage
- System OS version
- CPU Cores
- OpenGL driver version
- Application version string
- Session duration (in seconds)
- Model
- MAC address
- GPU model name
- GPU driver version

Nuke Non-commercial

If you want to try out or learn Nuke, you can run Nuke Non-commercial. This version allows you to explore most of Nuke's features, but prevents the commercial use of the application. For more information, see About Nuke Non-commercial.

To launch the application on Linux, do one of the following:

- Open the Nuke application directory and double-click the NukeNC, NukeXNC, or Nuke StudioNC icon.
- Using a terminal, navigate to the Nuke application directory and enter:
  - ./Nuke11.3 --nc --nukex to launch NukeX.
  - ./Nuke11.3 --nc to launch Nuke.

If you have already activated Nuke Non-commercial on the current device, the graphical interface appears, and a command line window opens. If you haven't activated the device yet, proceed to Licensing Nuke Non-commercial on Linux.
Licensing on Linux

The following licensing methods are available:

- **Activation Keys** - activation keys allow you to activate and generate your actual product license key, at a later point after purchase, on the machine for which you require the license.

  They are provided for both node locked and floating licenses, and generate the appropriate license type once installed using the product's Licensing dialog or online using the Activate a Product page: https://www.foundry.com/licensing/activate-product

- **Node Locked Licenses** - these can be used to license an application on a single machine. They do not work on different machines and if you need them to, you’ll have to transfer your license.

  Node locked licenses, sometimes called uncounted licenses, do not require additional licensing software to be installed.

- **Floating Licenses** - also known as counted licenses, enable applications to work on any networked client machine. The floating license is put on the server and is locked to a unique number on that server.

  Floating licenses on a server requires additional software to be installed on the server to manage the licenses and give them out to the client stations that want them. This software is called the Foundry Licensing Tools (FLT) and can be downloaded at no extra cost from our website.

- **Subscription Licenses** - subscription licensing differs from traditional node locked or floating licenses in that a single license, or entitlement, is valid on any authorized device up to the entitlement's maximum number of activations.

  For more information on Subscription Licensing, see Licensing Nuke Non-commercial on Linux.

The following instructions run through the basic options for the first two licensing methods, but you can find a more detailed description in the Foundry Licensing Tools (FLT) User Guide available on our website: https://www.foundry.com/licensing

Obtaining Licenses

To obtain a license, you'll need your machine's System ID (sometimes called Host ID or rlmhostid). Just so you know what a System ID number looks like, here's an example: 000ea641d7a1.
Note: Bear in mind that, for floating licenses, you'll need the System ID of the license server, not the machines on which you intend to run the application.

There are a number of ways you can find out your machine’s System ID:

- Launch the application without a license, click Status, and then scroll down the error report until you see your System ID.
- Download the Foundry Licensing Tools (FLT) free of charge from https://www.foundry.com/licensing and then run the following command in a terminal shell:

  ```bash
  /usr/local/foundry/LicensingTools7.0/bin/systemid
  ```

When you know your System ID, you can request a license for Foundry products:

- from Foundry's Sales Department at sales@foundry.com
- from the product pages on our website, such as https://www.foundry.com/products/nuke
- by launching the application without a license and selecting:
  - **Buy Nuke** or **Buy Hiero** - opens a web browser directly to our website to purchase a license.
  - **Try Nuke** or **Try Hiero** - displays the 15-day trial license download screen. Enter your Foundry account details or create a new account and follow the on-screen instructions to receive a trial license.

Note: By default, if you have installed a temporary license, the application displays a dialog at start-up alerting you to the number of days remaining. If you want to disable this behavior, you can set the FN_DISABLE_LICENSE_DIALOG environment variable to 1 to suppress the warning message about imminent license expiration. See [Environment Variables](#) for more information.

Installing Licenses

When you start the application before installing a license, a Licensing dialog displays an error informing you that no license was available. The installation process is dependent on what type of license you requested:

- **License file** - if you requested a license file, typically `foundry.lic`, this option allows you to browse to the file location and install it automatically. See [To Install a License from Disk](#) for more information.
- **Activation Key or license text** - if you requested an Activation Key or license by email, this option allows you to paste the key or license text into the Licensing dialog, which then installs the license in the correct directory. See [To Install an Activation Key or License Text](#) for more information.
• **A floating license** - if you requested a floating license to supply licenses to multiple client machines, this option allows you to enter the server address that supplies the client licenses.

  ![Note: You must install a floating license and additional software on the license server to use this option.]

  See [To Install a Floating License](#) for more information.

### To Install a License from Disk

1. Save the license file to a known location on disk.
2. Launch the application.
   - The **Licensing** dialog displays.
3. Click **Install License** to display the available license installation options.
4. Click **Install from Disk**.
5. Browse to the location of the license file.
6. Click **Open** to install the license automatically in the correct directory.

### To Install an Activation Key or License Text

1. Launch the application.
   - The **Licensing** dialog displays.
2. Click **Install License** to display the available license installation options.
3. Click **Activation Key / License Text** and then either:
   - Enter the Activation Key string in place of Insert Activation Key Here. A license key typically looks something like this:  
     ```
     nuke-0101-77d3-99bd-a977-93e9-8035
     ```
   OR
   - Copy the license text and paste it over the Copy/Paste license text here string. License text typically looks something like this:
     ```
     LICENSE foundry nuke_i 2015.0929 29-sep-2015 uncounted
     hostid=000a957bfde5 share=h min_timeout=30 start=29-sep-2015 issued=29-sep-2015 disable=VM _ck=da32d7372f sig="60P0450MJRP97E3DPB42C99Y5UAPRMEMGNQ39PG22H4WGH3WFK2KPTXFWJTYR0GYASJBC0PU8"
     ```
4. Click **Install**.
   - The license is automatically installed on your machine in the correct directory.
Note: Activation Keys require an internet connection. If you access the internet through a proxy server and cannot connect to the activation server, you may get an error dialog prompting you to either:

Click Use Proxy to enter the proxy server name, port number, username, and password. This enables the application to connect to the activation server and obtain a license. The license is then installed automatically, or

Click on the web link in the dialog and use the System ID (also known as hostid) provided to manually activate and install a license.

To Install a Floating License

If you requested a floating license from Foundry, you will receive your license key (foundry.lic) in an email or internet download. You should also receive the Foundry License Utility (FLU) application to help you install the license key on the license server machine. The server manages licenses for the client machines on your network.

Note: The FLU is also available to download from https://www.foundry.com/licensing

1. Make sure you have saved both the license key (foundry.lic) and the FLU application in the same directory.
2. Run the FLU application.
   The license key automatically appears in the FLU window if the FLU and foundry.lic are in the same directory.

Tip: If they are not in the same directory, you can either copy and paste the contents of the license key or drag-and-drop the file into the FLU window.

3. Click Install.
   This checks the license file and, provided that the license is valid, installs it into the correct directory.
4. In order for the floating license to work, you will need to install the Foundry Licensing Tools (FLT) on the license server machine.
   For more information on how to install floating licenses, refer to the FLT User Guide, which you can download from our website https://www.foundry.com/licensing
5. Once your license server is up and running, launch Nuke on the client machine. The Licensing dialog displays.

6. Click Install License to display the available install methods.

7. Click Use Server and enter the server address in the field provided. The format for the server name is: <port>@<servername>, for example, 30001@red.

**Note:** You must perform steps 5 through 7 on each client machine that requires a license from the server.

**Further Reading**

There is a lot to learn about licenses, much of which is beyond the scope of this manual. For more information on licensing Nuke, displaying the System ID number, setting up a floating license server, adding new license keys and managing license usage across a network, you should read the Foundry Licensing Tools (FLT) User Guide, which can be downloaded from our website, https://www.foundry.com/licensing
Licensing Nuke Non-commercial on Linux

Subscription licensing differs from traditional node locked or floating licenses in that a single license, or entitlement, is valid on any authorized device up to the entitlement's maximum number of activations.

- An **Entitlement** represents the right to run a Foundry product for a set amount of time on a set number of devices.
- An **Authorized Device** is a recognized device, such as a desktop computer, on which entitlements can be activated.

For example, if an **Entitlement** for Nuke has five activations, you can use Nuke on five separate **Authorized Devices** simultaneously. If you want to activate another device, you have to deactivate an existing one, but you can activate and deactivate devices as often as you like.

To get started with Nuke Non-commercial, follow these steps:

1. Create a Foundry account using a valid email address on our website, https://www.foundry.com/user/register
2. Launch Nuke in non-commercial mode as described under Launching on Linux. A **Licensing** dialog displays, similar to regular licensing. Nuke Non-commercial is free, but your entitlement only contains two activations.
3. Click **Authorise Device**.
4. Enter your account email address and password and then click **Authorise Device**.
5. A subscription license is created in your home directory: 
   /home/<username>/FoundryLicensing/<SystemID>

**Note:** Replace `<username>` and `<SystemID>` with the current user and the MAC address of the device, respectively.

The license looks something like this: c58edf7e-17ab-435b-8d8a-b3a9b347ab11.lic

6. Once the license is installed, click **Launch** to start using Nuke.
7. If you need to deactivate an entitlement or deauthorize a device, navigate to **Help > License** and, click:
   - **Deactivate Nuke** to reclaim one of your entitlements,
• **Deauthorize Device** to reclaim your existing Foundry entitlements on this device and stop additional ones running, or

• **Deauthorize All Devices** to reclaim your existing Foundry entitlements on all devices associated with your account, and stop additional ones running.
Uninstalling Apps on Linux

To uninstall Nuke or Hiero on Linux, there are a few things you need to do:

1. Navigate to `/usr/local/` and delete the **Nuke 11.3v6** directory.
2. Delete, rename, or move your `.nuke` or `.hiero` directories, if they exist.
   - The directories are found in your home directory, by default:
     `/home/<login name>/.nuke`
3. Delete, rename, or move your cached files, which reside in the following directory by default:
   `/var/tmp/nuke`

**Note:** If you specified an alternate directory using the NUKE_TEMP_DIR environment variable, purge those files as well as the default location. See **Nuke Environment Variables** for more information.
Nuke Studio Environments

The Compositing Environment

You can use the Compositing environment to perform node-based compositing with a choice of different VFX tools, manage color grading, review your script, and render out your script. See Using the Compositing Environment for more information.

By default, there is a Node Graph panel in the lower-left corner, a Viewer panel in the top-left corner, and a Properties panel on the right.

Node Graph
The Node Graph is where you add nodes and build your node tree.
Properties Panel | When you add a node to the Node Graph, its properties appear in the Properties panel on the right.

Viewer | To check the result, you can view the output in a Viewer.

The Timeline Environment

The Nuke Studio Timeline environment allows you to conform, create Nuke Comps, add soft effects, perform timeline-based editing, export your project, and view and edit metadata and properties. See Using Nuke Studio’s Timeline Environment for more information.

Project Tab | You can manage all aspects of your projects and bins in the Project tab.

Menu Bar | Use the Menu bar to access Nuke Studio's dropdown menus.

Bin View | The bin displays the contents of any selected Project tab.

Viewer | You can display and review your media in the Viewer.

Timeline Info | The timeline info displays the current timeline’s sequence, media, and...
metadata information.

<table>
<thead>
<tr>
<th>Editing Tools</th>
<th>There is a comprehensive set of editing tools provided in Nuke Studio. See Timeline Editing Tools for more information.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spreadsheet Tab</td>
<td>Use the Spreadsheet tab to display the contents of the timeline in spreadsheet form. Note that the spreadsheet and timeline are linked, mirroring any selections made.</td>
</tr>
<tr>
<td>Timeline</td>
<td>The timeline displays the current track, including all shots and any effects that have been added.</td>
</tr>
</tbody>
</table>

How the Panels Link

In Nuke Studio, some panels are linked together so that the linked panel tabs are automatically displayed at the front of a panel, when one of linked group is selected. This makes it quick and easy to switch between working in the Timeline and Compositing environments.

The Timeline linked group includes the spreadsheet, the Timeline environment Viewer, and the timeline. The Compositing linked group includes the Node Graph, the Compositing environment Viewer, and the node Toolbar.

For example, when you are in the Compositing environment and you select a spreadsheet from the Properties content menu. The spread sheet appears as a new tab in the Properties pane, and automatically opens the timeline, and the timeline Viewer.
Shared panels take precedence over unshared panels. Shared panels include Curve Editor, Dope Sheet, Pixel Analyzer, Scopes, and Progress bars. The Properties pane and Script Editor are also shared panels, but differ slightly in that they show workspace-specific content. For example, the Properties panels shows soft effect keyframes and Node Graph keyframes depending on the currently selected workspace.

**Panel Focus and Keyboard Shortcuts**

Nuke Studio deals with panel focus in two ways: click focus and mouse-over focus. Click focus defines the main panel where keyboard events are registered and mouse-over focus allows you to temporarily override that focus. If the mouse-over focus is centered on a panel that doesn't recognize a particular keyboard event, the event falls back to the click focus panel.

A good example of click focus versus mouse-over focus is between Viewers and the Node Graph:

- If a Compositing Viewer has click focus and you press `B`, only the blue color channel is displayed in the Viewer. If the mouse-over focus resides on the Node Graph and you press `B`, the click focus is overridden and a Blur node is added to the Node Graph.
- If a Compositing Viewer has click focus and you press `, (comma), the gain is reduced in the Viewer. If the mouse-over focus resides on the Node Graph and you press `, (comma), the gain is still reduced in the Viewer because the Node Graph has no `, (comma) equivalent keyboard shortcut.

Some areas of the interface retain focus no matter where the mouse-over focus resides, such as the message control in the Text node's Properties panel and the Filter field in the Project panel.

Click focus is shown in the interface with a bright orange highlight in the panel name and mouse-over focus is shown as a muted orange highlight.
Default Workspaces

There are six default workspaces in Nuke/NukeX and Nuke Studio. By default, Nuke/NukeX opens in the **Compositing** workspace and Nuke Studio opens in the **Finishing** workspace. To change the workspace, you can do either of the following:

- Select **Workspace** from the top menu bar and then select the required workspace.

  OR

- Use the keyboard shortcuts to open the required workspace. Press **Shift** and the required workspace keyboard shortcut, depending on mode:

<table>
<thead>
<tr>
<th>Keyboard Shortcut</th>
<th>Nuke/NukeX</th>
<th>Nuke Studio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shift+F1</td>
<td>Compositing</td>
<td>Conforming</td>
</tr>
<tr>
<td>Shift+F2</td>
<td>LargeNodeGraph</td>
<td>Editing</td>
</tr>
</tbody>
</table>
Customizing Workspaces

Customizing Panes

You can resize and split panes to make more room for different elements on the screen. To resize a pane, drag the divider line of the pane into a new location.

You can split a pane by clicking on the content menu button in the top-left corner of the pane, and then selecting Split Vertical or Split Horizontal from the menu that opens.

Note: Pressing and holding the **spacebar** brings up the right-click menu for that pane, where available.
Moving the Toolbar

You can move the Toolbar into a new position by adding a new panel for it, hiding the panel name and controls, and resizing the panel. For more information on how to do this, see Adding Tabs, and Hiding Tab Names and Controls.

Adding Tabs

When you can’t fit more elements into your display, you can use tabs to save space. You can also use tabs to move the Toolbar into a new location.

You can add a tab by clicking on the content menu button in the top-left corner of the pane, and then selecting the type of tab you want to add. For example, you can add Node Toolbar, Node Graph, New Viewer, or Script Editor. The new tab is added on top of the existing tabs.

To move tabs, click on the name of the tab and drag it to a new position inside the same pane or in another pane.

You can close tabs again by clicking the X in the top-right corner of the tab you want to close.

Note: Closing a linked tab closes all associated tabs. If you hold Alt while closing a linked tab, it only closes that tab.

Soloing Tabs

You can choose to solo a tab by either right-clicking on the tab name or clicking the content menu, and then selecting Solo Tab. This automatically closes any other open tabs in the same pane, except for the one you have chosen to solo.

Floating Windows

You can turn tabs and panes into floating windows and vice versa. To turn a tab or pane into a floating window, click the content menu button in the top-left corner in the tab or pane you want to float, and then select Float Tab or Float Pane. You can also float tabs by either clicking Ctrl/Cmd+click on the tab name, or right-clicking on the tab name and select Float Tab.
To change a floating window into a tab or pane, click on the tab or pane name in the floating window and drag it to where you want it to dock. You can close floating windows by clicking the X button in the top-right corner of the tab or pane.

Maximizing Windows

To make a window fullscreen, first ensure the window you want to make fullscreen is active, and then press Alt+S. This could be the main application window or a floating Viewer. Making it fullscreen removes the window borders. You can also maximize tabs and panels by pressing spacebar.

Hiding Tab Names and Controls

You can hide the names and control buttons of tabs, as you may not need them with all panels, such as the Toolbar panel. To hide the names and controls on tabs, click the content menu button in the top-left corner of the tab, and disable Show Tabs.

You can show the names and controls on tabs again by moving the cursor over the top of the pane area until the top edge of the pane highlights, right-click to open the content menu, and select Show Tabs.

Saving and Loading Workspaces

After you have customized a workspace and you are happy with it, you can save it by selecting Workspace > Save Workspace... You are then asked to name it. After saving it, your custom workspace appears in the Workspace dropdown under the existing default workspaces. Select it from the dropdown to load it.

Workspaces are saved in your .nuke file under Workspaces > Nuke or NukeStudio, depending on which Nuke mode you're currently using.

**Note:** The location of the .nuke file varies by platform. See Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts for more detailed information.

Setting the Startup Workspace

When you launch Nuke, it opens in the workspace set in the Preferences. You can change the startup workspace to any other default workspace or a custom workspace, by doing the following:
1. Open the **Preferences** dialog by pressing **Shift+S**.
2. In the **Behaviors** section, select **Startup**.
3. Use the **startup workspace** dropdown to select the workspace you want Nuke to load on startup.

**Setting Preferences**

Open the **Preferences** dialog by pressing **Shift+S**. The preferences available depend on which mode Nuke is launched in.

**Changing Preferences**

The function of each preference is described under **Appendix A: Preferences**.

When you make a change to a preference, in most cases, the interface registers the change immediately (for example, an interface element displays in the new color). Some preference changes, such as **Performance > Hardware > default blink device**, require you to restart Nuke Studio, for the changes to take effect.
Saving Preferences

Nuke stores your preference settings in a file called preferences11.3.nk, which resides in your .nuke directory. The location of this is dependent on your operating system.

- Linux: /home/login name/.nuke
- Mac: /Users/login name/.nuke
- Windows: drive letter:\Users\login name\.nuke

**Note:** On Windows, .nuke resides in the directory pointed to by the HOME environment variable. If this variable is not set (which is common), the .nuke directory is under the folder specified by the USERPROFILE environment variable.

Each Nuke user can maintain his or her own unique settings. After making a change in the Preferences dialog, you can simply click OK to save and close your preferences. If you click Cancel, any changes that you made are not saved.

To save your preferences

Make the desired changes inside the Preferences dialog, then click OK. Nuke writes the new settings to preferences11.3.nk file, which you can find in the .nuke directory:

- **On Windows:** The .nuke directory can be found under the directory pointed to by the HOME environment variable. If this variable is not set (which is common), the .nuke directory is under the folder specified by the USERPROFILE environment variable - which is generally of the form drive_letter:\Documents and Settings\login name\ (Windows XP) or drive_letter:\Users\login name\ (Windows Vista).

  To find out if the HOME and USERPROFILE environment variables are set and where they are pointing at, enter %HOME% or %USERPROFILE% into the address bar in Windows Explorer. If the environment variable is set, the folder it’s pointing at is opened. If it’s not set, you get an error.

- **On Mac:** /Users/login name/.nuke
- **On Linux:** /users/login name/.nuke

Your new preferences remain in effect for the current and all subsequent sessions.

Resetting Preferences

To reset any changes you made simply click Restore Defaults in the bottom-left of the Preferences dialog. You can reset the preferences to default by deleting the preferences11.3.nk file. After doing
this, the next time you launch Nuke, it rebuilds the file with the default preferences.
Using the Compositing Environment

These pages are designed to help you learn how to use the Compositing environment including working with nodes, using the Toolbar, and using the Properties panel.

Toolbar, Menu Bar, and Content Menus

The Toolbar is located on the left-hand side of the Viewer in the Compositing environment. It consists of a number of menu icons. The different nodes are grouped under these icons based on their functions. You use the Toolbar to add nodes to the Node Graph.

The menu bar is located on top of the Nuke window (in all workspaces). Its menus, such as the File or Edit menu, let you perform more general actions related to the project or script, the Viewers, or editing, rather than certain individual nodes.

In addition to the Toolbar and the menu bar, you should also familiarize yourself with the content menus. They are the gray checkered boxes in the top-left corner of each pane. If you click on the box, a menu opens as shown in the image below. You can use the options in the menu to customize the workspace.

Finally, to work faster, you can right-click on the different panels to display a menu with options related to that particular panel.
Using the Toolbar

The Compositing environment’s Toolbar includes the following icons:

<table>
<thead>
<tr>
<th>Icon</th>
<th>Functions</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image.png" alt="Image Icon" /></td>
<td>Image Read and Write nodes, built-in Nuke elements, and Viewer nodes.</td>
</tr>
<tr>
<td><img src="draw.png" alt="Draw Icon" /></td>
<td>Roto shapes, paint tools, film grain, fills, lens flares, sparkles, and other vector-based image tools.</td>
</tr>
<tr>
<td><img src="time.png" alt="Time Icon" /></td>
<td>Retiming image sequences.</td>
</tr>
<tr>
<td><img src="channel.png" alt="Channel Icon" /></td>
<td>Channel management.</td>
</tr>
<tr>
<td><img src="color.png" alt="Color Icon" /></td>
<td>Applying color correction effects.</td>
</tr>
<tr>
<td><img src="filter.png" alt="Filter Icon" /></td>
<td>Applying convolve filters, such as blur, sharpen, edge detect, and erode.</td>
</tr>
<tr>
<td><img src="keyer.png" alt="Keyer Icon" /></td>
<td>Extracting procedural mattes.</td>
</tr>
<tr>
<td><img src="merge.png" alt="Merge Icon" /></td>
<td>Layering background and foreground elements.</td>
</tr>
<tr>
<td><img src="transform.png" alt="Transform Icon" /></td>
<td>Translating, scaling, tracking, and stabilizing elements.</td>
</tr>
<tr>
<td><img src="3d.png" alt="3D Icon" /></td>
<td>3D compositing nodes and tools.</td>
</tr>
<tr>
<td><img src="particles.png" alt="Particles Icon" /></td>
<td>Creating, spawning, and editing particles.</td>
</tr>
<tr>
<td><img src="deep.png" alt="Deep Icon" /></td>
<td>Creating, merging, and editing deep images.</td>
</tr>
</tbody>
</table>
### Icon | Functions
--- | ---
Eye | Views
   - Nodes for working with views and stereoscopic or multi-view material.

File | Metadata
   - Viewing, editing, and comparing image metadata.

Tool | ToolSets
   - Creating, deleting, and managing tool sets.

Folder | Other
   - Additional operators for script and Viewer management.

Info | Any installed plug-ins and custom menus that do not have their own icon.

**Note:** Nuke Assist does not support all the toolbars available in Nuke. See [Nuke Products](#) for more information.

To display a tool tip that explains the icon's function, move the cursor over the icon.

To make selections from the Toolbar, click on an icon and select an option from the menu.
Working with Nodes

Nodes are the basic building blocks of any composite. You can create a new compositing script by inserting and connecting nodes to form a network of operations. These operations concatenate and allow you to manipulate your images.

Note: Not all nodes are supported in all versions of products. For a run down of supported nodes, see Nuke Products.

Adding Nodes

You can add nodes using the Toolbar, the Tab menu, or the right-click menu. When you add a node, Nuke automatically connects it to the currently selected node.

Using the Toolbar

1. Select the existing node that you want the new node to follow by clicking on it.
2. Click an icon on the Toolbar and select a node from the menu that appears. For example, if you want to add a Blur node, click the Filter icon and select Blur.

Note: You can press the middle-mouse button on a menu icon to repeat the last item used from that menu. For example, if you first select a Blur node from the Filter menu, you can then add another Blur node by simply pressing the middle-mouse button on the Filter icon.

Using the Tab Menu

Tip: See Using the Tab Menu for more detailed information.

1. Select the existing node that you want the new node to follow by clicking on it.
2. Press the Tab key and start typing the name of the node you want to create.
   This opens a prompt displaying a list of matches.
3. To select the node you want to add from the list, you can either click on it, or scroll to it with the **Up** and **Down** arrow keys and press **Return**.

**Note:** To add the last node created using this method, simply press **Tab** and then **Return**.

Using the Right-Click Menu

To add a node using the right-click menu, do the following:
1. Right-click on an existing node that you want the new node to follow.
2. From the menu that opens, select the node you want to add.

**Note:** You can also add nodes using keyboard shortcuts. Most menus in the Toolbar include a note of the relevant keyboard shortcut next to the item in question.

Adding a New Branch of the Node Tree

To add a node in a new branch of the Node Tree, do the following:
1. Select the existing node that you want the new node to follow by clicking on it.

![Diagram](image)

2. Hold down **Shift** and create the node using the Toolbar, Tab menu, or right-click menu. To add a node in a new branch with the Tab menu, press the **Tab** key first, then hold down **Shift** when selecting the new node.

![Diagram](image)

The node is added after the selected node in a new branch of the node tree.

Selecting Nodes

Nuke offers a number of options for selecting nodes. Selected nodes are highlighted in a color that is defined in the Preferences dialog. See Appendix A: Preferences for more information.
<table>
<thead>
<tr>
<th>Type of Selection</th>
<th>How to Make the Selection</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select a single node</td>
<td>To select a single node, simply click on it.</td>
</tr>
<tr>
<td>Select multiple nodes</td>
<td>To select multiple nodes, you can either press Shift while clicking on each node you want to select, or click and drag in the workspace to draw a marquee around the nodes you want to select.</td>
</tr>
<tr>
<td>Select all upstream nodes</td>
<td>Press Ctrl/Cmd while dragging a node. Nuke selects all nodes that feed data to the selected node. You can also Ctrl/Cmd+Shift+click to select more nodes without clearing the current selection.</td>
</tr>
<tr>
<td>Select all nodes in a node tree</td>
<td>Click on a node in the Node Graph and select Edit &gt; Select Connected Nodes (or press Ctrl/Cmd+Alt+A). This selects all nodes in the node tree, whether they are upstream or downstream from the current node. Nodes in any other node trees are not selected.</td>
</tr>
<tr>
<td>Select all nodes in a script</td>
<td>Select Edit &gt; Select all (or press Ctrl/Cmd+A).</td>
</tr>
<tr>
<td>Select nodes by name</td>
<td>1. Choose Edit &gt; Search, or press the forward slash (/). A dialog is displayed.</td>
</tr>
<tr>
<td></td>
<td>2. Type an alphanumeric string that is included in the names of the nodes you wish to select. Click OK.</td>
</tr>
<tr>
<td>Invert a selection</td>
<td>Select Edit &gt; Invert Selection.</td>
</tr>
</tbody>
</table>

### Replacing Nodes

To replace a node in Nuke, simply hold down Ctrl/Cmd and create a new node. See below for more details.

---

**Tip**: When typing the above alphanumeric search string, you can use asterisks (*) and question marks (?) as wild cards. An asterisk stands for multiple alphanumeric characters. A question mark represents just one character.
To Replace One Node with Another

1. In the Node Graph, select the node you want to replace by clicking on it.
2. Hold down **Ctrl/Cmd** and create a new node using the Toolbar, a right-click menu, or the Tab menu. To replace a node with the Tab menu, press the **Tab** key first, then hold down **Ctrl/Cmd** when selecting the new node.

The new node replaces the selected node in the Node Graph.

**Note:** Note that you cannot replace nodes in this manner if you are using a keyboard shortcut (such as **B** for the Blur node) to create the new node.

Renaming Nodes

There are a couple of ways to rename a node in Nuke.

To rename a node, you can either:

1. Double-click on the node to open its properties panel.
2. In the title field on top of the **Properties** panel, you should see the current name of the node. Delete that name and enter a new name in its place,

OR

1. Click on the node in the Node Graph to select it.
2. Press **N**.
3. Enter a new name for the node in the rename field that appears on top of the node.
Editing Nodes

To copy, paste, and perform other editing functions in the Node Graph, you can use the standard editing keys (for example, Ctrl/Cmd+C to copy, and Ctrl/Cmd+V to paste). You can copy nodes to files or memory. Copied nodes inherit the values of their parent, but these values, unlike those in cloned nodes, are not actively linked. Copied nodes allow you to assign different values to the original and the copy.

When you paste nodes, Nuke automatically connects them to the node that is selected before the paste operation. If you don't want to connect anything, click on a blank area of the workspace to deselect any selected nodes before pasting.

<table>
<thead>
<tr>
<th>Type of Edit</th>
<th>How to Preform It</th>
</tr>
</thead>
</table>
| Copy nodes to the clipboard | To copy nodes to the clipboard:  
1. Select the node or nodes you want to copy.  
2. Choose Edit > Copy (or press Ctrl/Cmd+C). |
| Copy nodes to files       | To copy nodes to files:  
1. Select the node or nodes you want to copy.  
2. Choose File > Export Comp Nodes...  
3. Navigate to the directory where you want to store the node(s) as a script.  
4. Type a name for the script at the end of the pathway, followed by the extension .nk. |
| Cut Nodes                 | To cut nodes:  
1. Select the node or nodes you want to cut.  
2. Choose Edit > Cut (or press Ctrl/Cmd+X).  
   Nuke removes the node(s) from the script and writes the node(s) to the clipboard. |
| Paste nodes from the clipboard | To paste nodes from the clipboard:  
1. Select the node that you want the pasted node(s) to follow.  
2. Choose Edit > Paste (or press Ctrl/Cmd+V).  
   Nuke adds the nodes to the script, connecting them to the selected node. |
| Load nodes from files     | To load nodes from files:  
1. Select the node that you want the loaded node to follow. |
### Cloning Nodes

You can clone nodes directly into the Node Graph or copy clone nodes in preparation for pasting them elsewhere in a script. Cloned nodes inherit the values of their parent, but unlike copied nodes, they also maintain an active link with their parents' values. If you alter the values of a node that has been cloned, the clone automatically inherits these changes.

Clones are helpful for maintaining consistent setups across multiple elements. For example, you might use clones to apply an identical film grain setup to a series of elements shot on the same stock. If you need to make changes to the setup, these changes would automatically ripple throughout the script.

**Note:** You cannot clone gizmos. This applies to both gizmos created in your facility and the nodes in the Nuke default Toolbar that are, in fact, gizmos, such as the LightWrap node, the Grain node, and the IBK nodes. For more information on gizmos, see the Configuring Nuke chapter.

### Cloning a Node

To clone nodes, do the following:
1. Select the node or nodes you want to clone.
2. Click **Edit > Clone**.

   Nuke clones the node(s), while maintaining an active link to the parental node(s). The clone has the same name as the original node and is indicated with an orange line connecting the clone to its parent node.

### Copying a Node as a Clone

You can copy a node as a clone and paste it in a different location, by doing the following:
1. Select the node or nodes you want to clone.
2. Click **Edit > Copy as Clones** (or press **Ctrl+K**).

Decloning a Node

To declone nodes, do the following:
1. Select the node or nodes you want to declone.
2. Click **Edit > Declone** (or press **Alt+Shift+K**).
   Nuke removes the clone status of the selected nodes.

Disabling and Deleting Nodes

Nuke allows you to disable, re-enable, and delete nodes in your Node Graph.

<table>
<thead>
<tr>
<th>Action</th>
<th>Instructions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Disable node</td>
<td>To disable nodes:</td>
</tr>
<tr>
<td></td>
<td>1. Select the node or nodes you want to disable.</td>
</tr>
<tr>
<td></td>
<td>2. Select <strong>Edit &gt; Node &gt; Disable/Enable</strong> (or press <strong>D</strong>).</td>
</tr>
<tr>
<td></td>
<td>Nuke disables the node's effect on the data stream.</td>
</tr>
<tr>
<td>Re-enable node</td>
<td>To re-enable nodes:</td>
</tr>
<tr>
<td></td>
<td>1. Select the node or nodes you want to re-enable.</td>
</tr>
<tr>
<td></td>
<td>2. Select <strong>Edit &gt; Node &gt; Disable/Enable</strong> (or press <strong>D</strong>).</td>
</tr>
<tr>
<td>Delete nodes</td>
<td>To delete nodes:</td>
</tr>
<tr>
<td></td>
<td>1. Select the node or nodes you want to delete.</td>
</tr>
<tr>
<td></td>
<td>2. Select <strong>Edit &gt; Delete</strong> (or press <strong>Delete</strong>).</td>
</tr>
<tr>
<td></td>
<td>Nuke removes the node(s) from the script.</td>
</tr>
</tbody>
</table>

Connecting Nodes

When you add or paste nodes into a script, Nuke automatically generates pipes between the currently selected node and the new nodes. As you build up a script, you'll need to move these pipes, or run new pipes between nodes. In Nuke, you make such modifications by dragging on the back end of the pipe (the end without the arrowhead).
Disconnecting Nodes

You can disconnect nodes by either dragging the head or tail of the connecting arrow to an empty area of the workspace, or selecting the lower node in the tree and pressing Ctrl/Cmd+D.

Reconnecting Nodes

You can reconnect a node by dragging the head or tail of the connecting arrow and drop it over the center of the node that want to connect.

Note: Nuke distinguishes the dual inputs that may run into a Merge node with the labels A and B. A refers to the foreground element, and B to the background element. Nuke always copies from the A input to the B. This means that if you later decide to disable the node associated with an A input, the data stream keeps flowing because, by default, it uses the B input.
Duplicating a Connecting Arrow

To duplicate a connecting arrow, hold **Shift** and drag the connecting arrow on top of the node you want to create a connection to. Nuke duplicates the connecting arrow, leaving the original connection untouched.

![Connecting Arrow Duplicating](image)

Adding Nodes Between Nodes

To add a node between two connected nodes, drag the node into the space between the already connected nodes. As you do so, you see the link between these two nodes become active. When that happens, simply release the node you are dragging and it is automatically placed and connected between the two nodes.

**Note:** Dragging multiple nodes into a node tree only connects the first node in the selection to the existing node tree.

Bending Connecting Arrows

To bend connecting arrows, do the following:

1. Select the node before the connector you want to bend.
2. From the Toolbar, select **Other > Dot.**
   
   A dot appears after the selected node, causing a bend in the connector.
3. Drag the dot as necessary to reposition the bend.

**Tip:** You can also add a dot to an existing connection by pressing **Ctrl/Cmd** and clicking on the yellow dot that appears on the connecting arrow.
Node Indicators

There are several indicators that can appear on the nodes in the Node Graph, depending on what you are doing. The following table describes what each indicator means.

<table>
<thead>
<tr>
<th>Indicator</th>
<th>Where it appears</th>
<th>What it means</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>The wide rectangles indicate the channels the node processes.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The thin rectangles indicate the channels that are passed through the node untouched.</td>
</tr>
<tr>
<td>M</td>
<td></td>
<td>The node’s effect is limited by a mask from either the node’s primary input or output.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The node has been disabled by pressing D or by selecting Edit &gt; Node &gt; Disable/Enable.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The node has been disabled using an expression.</td>
</tr>
<tr>
<td>G</td>
<td></td>
<td>The node has been cloned. The indicator appears on both the parent and the child node.</td>
</tr>
<tr>
<td>Indicator</td>
<td>Where it appears</td>
<td>What it means</td>
</tr>
<tr>
<td>-----------</td>
<td>------------------</td>
<td>---------------</td>
</tr>
<tr>
<td><img src="image" alt="A" /></td>
<td><img src="image" alt="Blur2" /></td>
<td>One or more of the node parameters are animated over time.</td>
</tr>
<tr>
<td><img src="image" alt="E" /></td>
<td><img src="image" alt="Blur4" /></td>
<td>One or more of the node parameters are being driven by an expression.</td>
</tr>
<tr>
<td><img src="image" alt="V" /></td>
<td><img src="image" alt="Blur3" /></td>
<td>You are working with a multi-view project and have split off one or more views in the node’s controls.</td>
</tr>
<tr>
<td><img src="image" alt=" " /></td>
<td><img src="image" alt="Blur1" /></td>
<td>You are working with a multi-view project and have split off one or more views in the node’s controls, dots also appear on the node to indicate which views have been split off. For example, if you are using red for the left view and split off that view, a red dot appears on the node.</td>
</tr>
<tr>
<td><img src="image" alt="X" /></td>
<td><img src="image" alt="Blur1" /></td>
<td>The full effect of the node is not in use, because you have adjusted the <strong>mix</strong> slider in the node’s controls.</td>
</tr>
</tbody>
</table>

**Displaying Node Information**

You can obtain more detailed information from any node by selecting that node and then pressing the I key. This displays an information window associated with that node, particularly useful when troubleshooting.
Customizing the Node Display

You can modify the color, name, and notes that a particular node displays. Doing so can make it easier for other artists to decipher the intent of your script. For example, you might color all nodes green that relate to keying.

Modifying Node Display Characteristics

To modify a node's display characteristics, do the following:
1. Double-click on the node in the Node Graph to display its properties in the Properties pane.
2. Click the Node tab at the top of the dialog in the Properties pane. Its attributes are displayed:
In the node's properties, you can also do the following:

- Select the **hide input** checkbox to conceal the node's incoming pipe. This can enhance the readability of large scripts.
- Select the **postage stamp** checkbox to display a thumbnail render of the node's output on its surface. You can also press Alt+P on the Properties pane or on the node, to toggle postage stamps.

If you want the postage stamp to display a fixed frame (rather than update to match the current frame), enter the frame to display in the **static frame** field. For this to work, you also need to press Shift+S to open the Preferences dialog, go to Panels > Node Graph, and ensure the **postage stamp mode** dropdown is set to Static frame. Note that if the frame number you use is outside the frame range for the node, it is clamped to the first or last frame in the range.

**Tip:** You can also have Nuke automatically color code nodes for you based on their function. See Appendix A: Preferences.
Creating Node Tool Sets

If you find yourself creating the same set of nodes repeatedly, you can create a custom tool set in the Nuke toolbar. This allows you to quickly and easily create the set of nodes from the ToolSets menu.

Tool sets can be shared between artists if they are using a centralized .nuke folder. This needs to be accessed through a NUKE_PATH environment variable that you can set up (see Configuring Nuke for more information).

To create a tool set do the following:

1. In the Node Graph, select any number of nodes (or just one node). The nodes don't have to be connected to each other.

2. Click (ToolSets) on the toolbar and select Create. You can also right-click on the Node Graph and select ToolSets > Create. The CreateToolSet dialog appears. You can also access your tool sets by pressing Tab and searching for them in the search field.

3. In the ToolSets menu dropdown, select the menu where you'd like to place your new tool set. Then give it a name in the Menu item field and click Create.

   By default, your new tool set goes under the ToolSets menu, but if you'd like to create a subfolder for the tool set, you can do that by specifying the folder name before the tool set name, separated by a forward slash, in the Menu item field. For example, entering Roto/BasicRoto would create a subfolder called Roto in the ToolSets menu, and place a new tool set by the name of BasicRoto in it.

   There are some tool sets for creating particles, for instance, that are built in to Nuke. These can also be found in the ToolSets menu. To prevent confusion between built-in tool sets and tool sets that you've created, if you happen to create a tool set folder with the same name as a built-in one, Nuke adds [user] in front of the folder name for clarity.

4. To delete a tool set you've created, you can click ToolSets > Delete in the toolbar and select the tool set you want to remove.
Using the Tab Menu

The Node Graph’s Tab menu, opened by pressing Tab in the Node Graph, enables you to search for and add nodes easily using partial names. Commonly used nodes are weighted so that they appear higher up the list of choices and you can favorite nodes, pinning them to the top of the list. Weights and favorites can be enabled, disabled, and cleared in the Preferences under Behaviors > Nodes.

Adding Nodes by Sub-string

Typing characters into the Tab menu displays a list of matches containing those characters. The more characters you enter, the more the list of applicable nodes is refined. To display the menu, press Tab when the Node Graph has either click or mouse-over focus.

Tip: See Panel Focus and Keyboard Shortcuts for more information about focus.

Typing in the Tab menu narrows the selection. For example, entering the string dea displays all entries beginning with dea. In this case, you can see that there are no nodes beginning dea, so Nuke looks instead for node beginning de with an a in the name.
The Tab menu parses capitalization as well. For example, if you want a DeepMerge node, start your string DM rather than dm to refine the results.

You can also search by toolbar menu name. For example, if you know that you're searching for a color correction node you could enter [Color].
Nuke’s Bookmark feature also allows you to use sub-strings to locate the bookmark you need. See Bookmarking Nodes for more information.

Adding and Removing Favorites

Favorites allow you to mark nodes that you use most often so that they are listed first in the Tab menu. You can add and remove favorites by clicking the star next to their entry in the node list.

You can enable, disable, and clear all favorites in the Preferences under Behaviors > Nodes.
Managing Weighting

Weighting describes how often you use a node from the Tab menu. Commonly added nodes carry more weight and are listed higher up in the node list.

**Note:** Adding nodes using a keyboard shortcut, such as pressing B to add a Blur node, does not affect node weighting.

The more often you add a node form the Tab menu, the larger the green dot next to the node name.

Weighting can produce unexpected results when you enter strings in the Tab menu. For example, if you use Backdrop nodes often, you may find that weighting causes Backdrop to appear above nodes that start with the first letter of the search string.
You can enable, disable, and clear weighting information in the Preferences under Behaviors > Nodes.
Navigating Inside the Node Graph

As scripts grow in complexity, you need to be able to pan to a particular cluster of nodes quickly. The Node Graph offers a couple of methods for doing so.

Panning

You can pan using the mouse or the navigator map that appears in the lower right corner of the Node Graph.

Panning with the Mouse

To pan with the mouse, press the middle mouse button and drag the mouse pointer over the workspace (you can also use Alt+drag). The script moves with your pointer.

**Note:** In many Linux windows managers, the Alt key is used by default as a mouse modifier key. This can cause problems in 3D applications where Alt is used for camera navigation in 3D environments.

You can use key mapping to assign the mouse modifier to another key, such as the (Super or Windows logo) key, but the method changes depending on which flavor of Linux you're using. Please refer to the documentation on key mapping for your particular Linux distribution for more information.

Panning with the Map

If your script is larger than the visible workspace, a navigator map automatically appears in the bottom-right corner.
The map shows you a miniature view of the entire script and the pink rectangle shows the portion of the script that you see within the workspace borders.

To pan with the map, drag the pink rectangle to pan to a different view of the script.

When the whole script is contained within the window border, then the map automatically disappears.

**Tip:** The navigation map is resizeable. Drag on its upper-left corner to make it as large or small as you like.

## Zooming, Fitting in the Node Graph

<table>
<thead>
<tr>
<th>Action</th>
<th>How to perform it</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Zooming in</strong></td>
<td>To zoom in, you can either:</td>
</tr>
<tr>
<td></td>
<td>• Move the cursor over the area you want to zoom in on, and press the addition key (+) repeatedly until the workspace displays the script at the desired scale.</td>
</tr>
<tr>
<td></td>
<td>OR</td>
</tr>
<tr>
<td></td>
<td>• Press <strong>Alt</strong> and drag right while holding down the middle-mouse button.</td>
</tr>
<tr>
<td><strong>Zooming out</strong></td>
<td>You can zoom out by doing one of the following:</td>
</tr>
<tr>
<td></td>
<td>• Move your mouse pointer over the area you want to zoom out from, and press the subtraction key (-) repeatedly until the workspace displays the script at the desired scale.</td>
</tr>
<tr>
<td></td>
<td>OR</td>
</tr>
<tr>
<td></td>
<td>• Press <strong>Alt</strong> and drag left while holding down the middle-mouse button.</td>
</tr>
</tbody>
</table>

**Note:** In many Linux windows managers, the **Alt** key is used by default as a mouse modifier key. This can cause problems in 3D applications where **Alt** is used for camera navigation in 3D environments. You can use key mapping to assign the mouse modifier to another key, such as the (Super or Windows logo) key, but the method changes depending on which flavor of Linux you're using. Please refer to the documentation on key mapping for your particular Linux.
### Action | How to perform it
--- | ---
distribution for more information. | 
Fitting the selected nodes within the visible workspace | To fit selected nodes in the Node Graph, click the middle-mouse button or press **F**.
Fitting the node tree within the visible workspace | To fit the entire node tree in the Node Graph, click on the Node Graph to make sure no nodes are selected and click the middle-mouse button or press **F**.

### Bookmarking the Pan and Zoom Level

To bookmark the pan and zoom level:

1. Pan and/or zoom the script as necessary.
2. To save the current pan and zoom level, navigate to **Edit > Bookmark > Save Location 1** (or press **Ctrl/Cmd+F7**).
   
   You can save three more locations in a similar manner, using **Save Location 2**, **Save Location 3**, and **Save Location 4** (or by pressing **Ctrl/Cmd+F8/F9/F10**).
   
   The saves are temporary and not saved to the script.
3. To restore a saved location later, select **Edit > Bookmark > Restore Location1, Restore Location 2, Restore Location 3, or Restore Location 4** (or press **Shift+F7/F8/F9/F10**).

**Note:** Restoring a saved location doesn't work across different Node Graphs - you can't save a location in one Node Graph and then restore it in another.

### Bookmarking Nodes

One extremely useful function of Nuke nodes, is their ability to act as jump-to points throughout a project.

1. Double-click on the node you want to bookmark to open its properties.
2. Go to the **Node** tab and select the **bookmark** checkbox (or press **Ctrl/Cmd+Shift+B**).
   
   This adds the node to the bookmark list.
By default, Backdrop nodes have their bookmark checkbox enabled. All other nodes have it disabled.

3. Select Edit > Bookmark > Jump to Bookmarked Node (or press J on the Node Graph) to bring up the bookmarks jump to menu.

4. Start typing the name of the node you wish to navigate to.
   Nuke shows all matching nodes that have the bookmark flag enabled. They are listed according to the node name or, if a label is present, the label followed by the node name in square brackets.

5. To select the node to navigate to, you can either click on it, or scroll to it with the Up and Down arrow keys, and press Return.

Searching for Nodes

Nuke’s Search feature allows you to search for nodes in your script and select any matches found. As a search string, you can enter all or part of a node name. For example, you can search for all Blur nodes in your script by entering bl as the search string.

You can also do more complex searches using regular expressions, such as searching for all the Read and Write nodes in a script.

To Search for Nodes

1. Select Edit > Search (or press /) to bring up the search dialog.
2. In the search field, enter the string you want to search for.
   - If you want to search for all nodes in the script, enter * (an asterisk).
   - If you want to search for all Read nodes in the script, enter Read.
   - If you want to search for all the Read and Write nodes, enter the following expression: (*Read* | *Write*)
3. Click OK.

Nuke searches for the nodes in the script and selects all matches it finds and focuses the Node Graph the first of them. If you perform the same search again, the Node Graph view focuses on the next node in order. For instance, if you have a ColorBars node, a ColorWheel node, and a ColorCorrect node in your Node Graph, searching for “Color” three times in a row selects all of these nodes and focuses on each of them in alphabetical order.

Note: When you enter expressions in the search dialog, remember that the search field only takes regular expressions. Any characters that have specific meanings in regular
expressions, such as square brackets ([ and ]), need to be preceded by the backslash character (\).

Cleaning up the Node Graph

Sometimes you may find that your node tree becomes a bit disorganized as you're creating and removing nodes. There's a quick fix you can use to tidy up your tree:

1. Select all the nodes you want to rearrange in the Node Graph.
2. Click **Edit > Node > Autoplace**. Alternatively, press **L**.
   
   The Autoplace function automatically arranges the selected nodes in a neat tree formation. It determines a sensible position for the nodes by ensuring they don't overlap with other nodes and by keeping the input connections horizontal.
Properties Panels

When you insert a node, its properties automatically appear in the Properties panel with options to define the node’s output. You can also open the properties panel later by doing any of the following:

- Double-click on the node in the Node Graph.
- Ctrl/Cmd+click on the node in the Node Graph.
- Select the node in the Node Graph and press Return.

**Tip:** To open a properties panel in a floating window, Ctrl/Cmd+double-click or Ctrl/Cmd+Alt+click on the node.

Managing the Properties Panel

You can limit the number of node panels that can be open in the Properties panel. To do so, enter the maximum number of nodes in the field on the Properties panel.

To lock the Properties panel and have all new panels appear in floating windows, click the lock button on the Properties panel.

To empty the Properties panel and close all the node panels in it, click the remove all panels button.

**Tip:** You can also close all the nodes in the Properties panel by Alt+clicking on the close (X) button of one of the nodes.
### Properties Panel Controls

These are the standard controls of every properties panel:

<table>
<thead>
<tr>
<th>Control</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>▼ ▶</td>
<td>Hide or show the node’s tabbed pages.</td>
</tr>
<tr>
<td>❌</td>
<td>Centers the node in the Node Graph.</td>
</tr>
<tr>
<td>❌</td>
<td>Centers one of the node’s inputs in the Node Graph. Select the input from the dropdown menu that appears.</td>
</tr>
<tr>
<td>❌</td>
<td>You can save, load, and manage node presets here.</td>
</tr>
<tr>
<td>❌</td>
<td>You can enter a new name for the node here.</td>
</tr>
<tr>
<td>□</td>
<td>Changes the color of the node. You can drag and drop this button on top of another color button to copy the color. To revert to the default color defined in your Preferences, right-click on the button and select <strong>Set color to default</strong>. An X on the button indicates the color is unset, and the color defined in the Preferences is used.</td>
</tr>
<tr>
<td>□</td>
<td>Changes the color used for the node’s controls in the Viewer. You can drag and drop this button on top of another color button to copy the color. To revert to the default color defined in your Preferences, right-click on the button and select <strong>Set color to default</strong>. An X on the button indicates the color is unset, and the color defined in the Preferences is used.</td>
</tr>
<tr>
<td>❌</td>
<td>Undoes the last change made to the node.</td>
</tr>
<tr>
<td>❌</td>
<td>Redoes the last change undone.</td>
</tr>
</tbody>
</table>
**Control** | **Function**
---|---
![Revert](image) | Reverts any changes made after the current node panel was opened.

Displays Help related to the node and its controls.

**Note:** If the Online Help is unavailable, a tooltip displays information about the node.

![Float](image) | Floats the panel. Clicking this button again docks the panel back into the pane if the originating pane is open.

![Close](image) | Closes the node panel. **Alt**+click this to close all the panels in the pane. **Ctrl/Cmd**+click to close all panels except the one clicked on.

Floating control panels also include the following buttons:

<table>
<thead>
<tr>
<th>Control</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Revert" /></td>
<td>Reverts any changes made after the panel was opened.</td>
</tr>
<tr>
<td><img src="image" alt="Cancel" /></td>
<td>Reverts any changes made after the panel was opened and closes the panel. Hitting this button right after a node was created also deletes the node from the Node Graph.</td>
</tr>
<tr>
<td><img src="image" alt="Close" /></td>
<td>Closes the panel.</td>
</tr>
</tbody>
</table>

Many node panels also contain several tabbed pages.
On the **Node** tab, you can usually adjust the following controls:

<table>
<thead>
<tr>
<th>Control</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>label</td>
<td>Lets you add comments to the node. The comments are displayed on the node's surface. If you like, you can use HTML in the label field. For example, to have your comments appear in bold, you can enter <code>&lt;b&gt;My Comment&lt;/b&gt;</code>. To add an icon called <code>MyIcon.png</code> to the node, you can use <code>&lt;img src=&quot;MyIcon.png&quot;/&gt;</code>. Save the icon in your plug-in path directory. (For more information on plug-in path directories, see <a href="#">Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts</a>. Most common image formats work, but we recommend using <code>.png</code>. Note that the HTML has been changed to a slightly non-standard form where newlines are significant. If there is a newline character in your data, a new line is displayed in the label.</td>
</tr>
<tr>
<td>font</td>
<td>Lets you change the font for any text displayed on the node. Bolds any text displayed on the node. Emphasizes any text displayed on the node. Lets you change the font size of any text displayed on the node. Lets you change the color of any text displayed on the node.</td>
</tr>
<tr>
<td>hide input</td>
<td>Check this to hide the node's incoming pipe. This control does not appear on all nodes.</td>
</tr>
<tr>
<td>cached</td>
<td>Check this to keep the data upstream from the node in memory, so that it can be read quickly. When this is checked, a yellow line displays under the node in the Node Graph.</td>
</tr>
<tr>
<td>disable</td>
<td>Check this to disable the node. Uncheck to re-enable. (You can also disable or re-enable a node by selecting it in the Node Graph and pressing D.)</td>
</tr>
<tr>
<td>dope sheet</td>
<td>Check this to force the node to always display in the Dope Sheet. You can also press <strong>Alt+D</strong> to toggle this on and off.</td>
</tr>
</tbody>
</table>
### Control		Function

<table>
<thead>
<tr>
<th>Control</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>bookmark</td>
<td>Check this to show the node in the bookmark list. This allows you to quickly navigate to the node. For more information, see <a href="#">Navigating Inside the Node Graph</a>.</td>
</tr>
<tr>
<td>postage stamp</td>
<td>Check this to display a thumbnail render of the node's output on its surface. You can also press Alt+P on the Properties Bin to toggle postage stamps.</td>
</tr>
<tr>
<td>static frame</td>
<td>If you want the postage stamp to display a fixed frame (rather than update to match the current frame), enter the frame to display here. For this to work, you also need to press Shift+S to open the Preferences dialog, go to the Node Graph tab, and make sure postage stamp mode is set to Static frame. Note that if the frame number you use is outside the frame range for the node, it is clamped to the first or last frame in the range.</td>
</tr>
<tr>
<td>lifetime range</td>
<td>Enter a start and end frame to control the frame range when this node is active. You can quickly toggle the lifetime on and off with use lifetime.</td>
</tr>
</tbody>
</table>

---

### Displaying Controls

To display a node's controls, double-click the node. Its properties panel appears.

The image below shows the controls available for editing parameters. Note that the presence of each control varies according to the parameter's function.
Using Input Fields

You can key values directly into a field, press the arrow keys to increment and decrement values, or use the middle-mouse button to activate virtual sliders.

Changing Field Value

To key in field values:
1. Double-click in the field to select the whole value.
2. Type the value you want to replace the selection.

**Tip:** You can also enter expressions (programmatic instructions for generating values) into fields. You always start an expression by typing =. See Expressions for information about how to format expressions.

**Tip:** Nuke also allows you to enter formulas into fields, making it easy to do quick calculations. For example, if you wanted to halve a value of 378, you could simply type 378/2 into a field and press Enter to get 189.

You can increment or decrement field values by hundreds, tens, tenths, hundredths, and so on. The magnitude of change depends on the initial position of your cursor. For example if you wanted to increment the initial value of 20.51 by ones, you would insert your cursor before the 0.

To increment or decrement a field value:
1. Click to insert the cursor just prior to the digit you want to increment or decrement.
2. Press the **up arrow** to increment by one unit, or the **down arrow** to decrement by one unit.

**Tip:** You can also increment and decrement values using the mouse wheel (if available) or by pressing **Alt** while dragging on the value. The latter method is particularly useful for tablet users.

Using Virtual Sliders

To use a virtual slider, you can hover the cursor over a numeric control, click the middle-mouse button, and drag the field to adjust the associated control.
Tip: Hold Shift to increase slider sensitivity.
Using the Color Controls

In the Properties panels, there are usually four extra controls to the right of a control including color swatch, color picker, channel selector and the animation menu. The first three are color controls. For more information about using the animation menu, see Animating Parameters.

You can use the color swatch button to activate the eye dropper tool. You can copy a color from one swatch to another by dragging and dropping the color swatch with the color you want to use over the color swatch you want to replace.

The color picker button can be used to display and make color selections. Depending on the settings in the Preferences dialog, you may use an in-panel color picker, or a floating color-picker window. (See Appendix A: Preferences for more information.)

You can use the channel selector to toggle between the slider and manually enter values for each of the channels.

Using the Color Swatch

You can use the color swatch to sample a color from the Viewer, by doing the following:
1. Click the color swatch to activate the eye dropper tool.
2. Move the cursor over the Viewer, over the color you want to sample.
   You can zoom in and pan as necessary until the area you want to sample is clear.
3. Ctrl/Cmd+click to sample a color value, or Ctrl/Cmd+Alt+click to sample a color from the node's input while viewing its output.
   You can sample a region instead of just a single pixel by also pressing Shift+drag when you're sampling the color. When sampling a region, Nuke calculates the average color of the region.
   If you are not happy with the pixel or region you've sample, you can simply repeat the sample procedure until you are happy with the selected pixel or region.
4. When you are happy with the color sample, click the color swatch again. This closes the eye dropper tool and now displays your selected sample.

Using the In-Panel Color Picker

If your Preferences > Panels > Control Panels > Color Panel > color picker button opens dropdown is set to in-panel color picker, you can display the color sliders and wheel within the Properties panel by clicking the color picker button 🖌️. You can have multiple in-pane color pickers open simultaneously.

![Color Picker](image)

Tip: Holding Ctrl/Cmd and clicking the color picker button opens the alternate color picker to the one specified in the Preferences dialog.

Using the In-Panel Color Wheel

You can adjust the hue by dragging the marker on the edge of the color wheel (or the marker inside the wheel) around to the required hue. You can adjust the saturation by dragging the marker inside the wheel in or out to the required saturation.

![Color Wheel](image)
Tip: You can also Ctrl/Cmd+click on the color wheel to only affect the hue, and Shift+click to only affect the saturation.

To adjust the value (brightness) of the color, hold down Ctrl/Cmd+Shift and drag right or left.

By default, the color wheel uses relative cursor positioning. This means the cursor position can be offset from the marker position, which allows for very fine adjustments. You can disable this behavior and use absolute cursor positioning by holding Alt while moving the markers.

Note: If any red, green, or blue values are higher than 1, the marker inside the wheel turns red. If the red, green, and blue channels all have a value of less than 0.1, the marker turns blue. If at least one channel is negative and at least one is positive, the hue indicator turns red.

Using the In-Panel Color Sliders

To increment the value by 0.01, left-click on the arrow button. To decrement the value by 0.01, right-click on the arrow button. Use Shift+click for 0.1, and Alt+click for 0.001.

You can also click and drag right or left on the button to scrub the value up or down. Use Shift+drag to scrub quickly, or Alt+drag to scrub slowly.

Using the Floating Color Picker Window

If your Preferences > Panels > Control Panels > Color Panel > color picker button opens dropdown is set to floating color picker, you can display color sliders and wheel in a floating window by clicking the color picker button.

Tip: Holding Ctrl/Cmd and clicking the color picker button opens the alternate color picker to the one specified in the Preferences dialog.

You can change the floating window to be horizontal by dragging on one of the corners of the window to resize it. When it is wide enough, the sliders automatically become horizontal.
• From the **TMI**, **HSV**, and **RGB** buttons, you can select which slider you want to display.
• There are three states of the color wheel: hide color wheel, show color wheel, and show color wheel with square. You can cycle through these states by simply clicking on the **color wheel** button multiple times.
• You can choose to show the color swatches by clicking the **color swatch** button.
• If you want the background of the sliders to show what the value of the color would be if the sliders were set to the current position, click the **Dyn** button.

Using the Color Wheel in the Floating Window

Using the color wheel in the floating window is exactly the same as using the in-panel color wheel; drag the marker on the edge of the wheel to adjust the hue, and drag the marker inside the wheel to adjust the saturation.

However, you can also pan and zoom on the color wheel in the floating window:
• To pan, press **Alt** and drag the cursor over the color wheel.
• To zoom, press **Alt** and drag left or right with the middle-mouse button.
• You can reset the zoom and/or pan by simply middle-clicking on the color wheel.

Using the Color Sliders in the Floating Window

Using the sliders in the floating color window is exactly the same as using the in-panel sliders.

Using the Color Swatches in the Floating Window

When you're happy with a color, you can save it by right-clicking on the swatch where you want to save it. This replaces the original swatch that was there. You can also drag and drop a color you want to save on any of the swatches to replace it.
The rectangle above the sliders shows the original color on the right, the currently selected color on the left. When you hover the cursor over this, an arrow appears allowing you to copy one to the other by clicking once on the rectangle.

Opening Another Floating Color Picker Window

You can open multiple color picker floating windows, by pressing Ctrl/Cmd+Alt+click on another parameter's color picker button.

Using the Channel Selector

By default, many parameters in Nuke automatically group channels for you. For example, if you drag on the gain slider in the ColorCorrect node, you simultaneously affect the R, G, and B channels (assuming you're processing the RGB layer.) You can use the channel selector button to display and edit the individual channel values. The number displayed on the channel selector button represents the number of available channels for editing.

You can edit an individual channel's value by doing the following:

1. Click the channel selector button.
   The available channels appear with value fields for each. For example, when you press the gain - channel selector button, four channel value fields appear (R,G,B, and A) as the channel selector shows that there are four available channels for editing.
   
   ![Channel Selector](image)

2. Enter the required values in the channel value fields.
   After you are happy with the values, simply click the channel selector button again to close and save your settings.
## Color Slider Functions

### RGBA Sliders

<table>
<thead>
<tr>
<th>Slider</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>The red slider (R)</td>
<td>This allows you to control the red channel's value (or the first channel in a layer if you are processing another layer besides RGBA).</td>
</tr>
<tr>
<td>The green slider (G)</td>
<td>This allows you to control the green channel's value (or the second channel in a layer if you are processing another layer besides RGBA).</td>
</tr>
<tr>
<td>The blue slider (B)</td>
<td>This allows you to control the blue channel's value (or the third channel in a layer if you are processing another layer besides RGBA).</td>
</tr>
<tr>
<td>The alpha slider (A)</td>
<td>This allows you to control the alpha channel's value (or the fourth channel in a layer if you are processing another layer besides RGBA).</td>
</tr>
</tbody>
</table>

### TMI Sliders

<table>
<thead>
<tr>
<th>Slider</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>The temperature slider (T)</td>
<td>This allows you to control apparent color temperature by inversely affecting red and blue values (assuming you are processing the RGBA layer).</td>
</tr>
<tr>
<td>The magenta/green slider (M)</td>
<td>This allows you to control the mix of green and magenta hues. To add more magenta (increase the red and blue channels' values, while decreasing the green channel's), drag up. To add more green (increase the green channel's value, while decreasing the red and blue channels'), drag down.</td>
</tr>
<tr>
<td>The intensity slider (I)</td>
<td>This allows you to simultaneously control the red, green, and blue channel values. To increase the value of all channels by the same amount, drag up. To decrease the value of all channels by the same amount, drag down.</td>
</tr>
</tbody>
</table>
HSV Sliders

<table>
<thead>
<tr>
<th>Slider</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>The hue slider (H)</td>
<td>This allows you to control the color’s location on the traditional color wheel (for example, whether the color is red, yellow, or violet).</td>
</tr>
<tr>
<td>The saturation slider (S)</td>
<td>This allows you to control the intensity or purity of the color.</td>
</tr>
<tr>
<td>The value slider (V)</td>
<td>This allows you to control the brightness of the color (the maximum of red, green, and blue values).</td>
</tr>
</tbody>
</table>

**Note:** The HSV sliders are only available in the floating color picker window. In the in-panel color picker, you can use the color wheel to adjust hue, saturation, and value.
Customizing a Node’s Defaults with Node Presets

Sometimes the default control values on a node aren’t the ones you use most often. If you find yourself frequently adjusting many control defaults, you may want to save your customized default values as a node preset.

To Create a Node Preset:
1. Set the controls in a node’s control panel to the values you want.
2. Click the load node presets button on the control panel title bar and select Save as Preset.
3. Give your preset a name and click Create.
   Your new preset now appears as an option when you click the load node presets button and select Apply Preset.

Loading and Deleting a Preset
1. To load a preset on a node, click the load node presets button and select your preset under Apply Preset. User presets are your own personal presets, and shared presets can also be used by other users.
2. To delete a preset, click the load node presets button and select Delete Preset. In the dialog that displays, select the preset you want to delete from the dropdown menu, and click Delete.
Animating Parameters

Animating a parameter refers to changing its value over time. You do so by setting keyframes (frames at which you explicitly define a value) and allowing Nuke to interpolate the values in between. You can animate most of Nuke's parameters in this manner.

Working with Animated Parameters

The Animation menu allows you to set key frames, delete keys, and perform other editing operations on the curves for animated parameters.

Setting Keyframes

To set keyframes, do the following:

1. Use a Viewer to navigate to a frame where you want to place a key.
2. Click the animation button next to the parameter you want to animate.
3. Select Set key from the dropdown menu. The parameter’s input field turns blue, indicating that a keyframe has been inserted. Nuke switches to autokey mode: when you change the parameters value at another frame, it automatically inserts another keyframe for you.
4. You can also set a key for all the controls in a node. To do so, right-click on a node's properties panel and select Set key on all knobs.
5. Navigate to the next frame where you want to place a key.
5. Edit the parameter’s value using the input field, regular slider, or color slider. The moment you change the value, Nuke creates a keyframe.
6. Continue adding key frames as necessary.
7. Use the Viewer’s scrubber to preview the result.

Deleting Keyframes

To delete a single keyframe:
1. Use the Viewer’s next edit/keyframe and previous edit/keyframe buttons to navigate to the keyframe that you want to remove. Notice that the scrub bar indicates key frames with a blue mark.
2. Click the animation button.
3. Select Delete key from the dropdown menu.
   Nuke removes the keyframe.

To delete all key frames from a parameter:
1. Click the animation button.
2. Select No animation from the dropdown menu.
   A confirmation dialog appears.
3. Click Yes.
   Nuke removes all key frames from the parameter, and sets the static value to match that of the current frame.

Linking Animated Parameters with Tracker

You can link any controls with others by using expressions (see Expressions), but you can also quickly and easily link with the Tracker node using the Link to option in the Animation menu. For example, to link the translate control of the RotoPaint node with a Tracker node, do the following:
1. Create the Tracker node you want to link to.
2. On the Transform tab of the RotoPaint node’s control panel, click on the translate animation menu.
3. Select Link to > Tracker linking dialog...
4. Select the Tracker node you want to use in the tracker node dropdown and in the link to dropdown, select whether you want to link to the position of the track or the translate values of it.
5. Select which tracks you want to use by checking the track boxes.
   The expression fields update with the appropriate expression syntax.
6. Then click OK, and your linking is done.
Your Bezier shape's translate value now changes when the Tracker value is changed.

Animated Parameters in the Dope Sheet and the Curve Editor

As you add keyframes to a parameter, Nuke automatically both adds markers to the Dope Sheet for them and plots a curve on its Curve Editor panel.

The Dope Sheet provides an easy way to edit keyframes as well as reposition, trim, and slip clips.

The Curve Editor allows you to adjust keyframes and the interpolation between them.

You can add key frames, delete key frames, and even adjust the interpolation between key frames without ever looking at the Dope Sheet or Curve Editor. However, as the animation grows more complex, you may find it easier to edit the animation by manipulating its keyframes directly. For more information on how to do so, see Editing Clips in the Dope Sheet and Using the Curve Editor.
Using the Dope Sheet

The Dope Sheet gives you access to all the keyframes you've created and provides an easy way of editing them. You can also use it to reposition, trim, and slip Read nodes and certain Time nodes.

Also see Using the Curve Editor.

Using the Dope Sheet Interface

To show the Dope Sheet, click the Dope Sheet tab. If you can't see the tab, right-click the Node Graph title bar and select Windows > Dope Sheet.

In the Dope Sheet, you can see the following:

<table>
<thead>
<tr>
<th>Hierarchy view</th>
<th>Click through the hierarchy view to see your nodes and a hierarchical list of animated controls. If you can't see a node in the list, try opening its properties panel. To make a node appear in the Dope Sheet without having its properties panel open, select the dope sheet checkbox in the bottom of the Node tab of the node's properties panel. You can display the Read and TimeClip nodes in the Dope Sheet when their properties are closed, by clicking the middle button in the bottom-left of the Dope Sheet.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Read nodes</td>
<td>Any Read nodes that have their properties panels open display as gray bars in the Dope Sheet.</td>
</tr>
</tbody>
</table>
### Time nodes

Any AppendClip, FrameRange, Retime, TimeOffset, TimeWarp, and TimeClip nodes that have their properties panels open display as green bars.

### Keyframes

Your keyframes display as gray markers in the keyframe grid. If you go down to the lowest level of hierarchy, you can also view the keyframes set for different views in stereoscopic or multi-view projects.

### Current frame indicator

The current frame indicator displays as an orange line.

### First and last frame in the project

The first and last frame in the project's frame range (as defined in the Project Settings) display as white/gray lines.

You can also use the following controls at the bottom of the Dope Sheet:

<table>
<thead>
<tr>
<th>Control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="lock.png" alt="Lock Icon" /></td>
<td>Synchronize the frame range of your project between the Dope Sheet and the Curve Editor.</td>
</tr>
<tr>
<td><img src="folder.png" alt="Folder Icon" /></td>
<td>Display all Read and TimeClip nodes in the Dope Sheet. By default, only the nodes that have their properties panels open show up on the Dope Sheet. If you click this button, all Read and TimeClip nodes in your script are displayed in the Dope Sheet. If you click the button again, the default view is restored.</td>
</tr>
<tr>
<td><img src="move.png" alt="Move Icon" /></td>
<td>Move the selected keyframes by the number of frames specified. You can enter negative values to move the keyframes backwards. <strong>Note:</strong> You cannot move keyframes past other keyframes. If you don't enter a number in the <strong>Move</strong> field, it shows the amount of your previous move.</td>
</tr>
<tr>
<td><img src="range.png" alt="Range Icon" /></td>
<td>Set the frame range to display in the Dope Sheet.</td>
</tr>
</tbody>
</table>

### Viewing Keys in the Dope Sheet

To view keys in the Dope Sheet, you need to create some Read nodes, Time nodes, or keyframes first. The Dope Sheet displays:
• All the Read nodes that have their properties panels open.
• Any AppendClip, FrameRange, Retime, TimeOffset, TimeWarp, and TimeClip nodes that have their properties panels open.
• Any nodes that have keyframes set on them and have their properties panels open.

Tip: To force the above nodes to display in the Dope Sheet even when their properties aren’t open, check the dope sheet box on the Node tab of the node’s properties panel or select the node in the Node Graph and press Alt+D.

Tip: If you want to view an AudioRead node under your keyframes, you can display it by right-clicking and selecting View > Audio > Draw Style > Below. For more information on the audio options, see Audio in Nuke.

The Dope Sheet and Time Nodes

If you have an AppendClip, FrameRange, Retime, TimeOffset, TimeWarp, or TimeClip node downstream of the node being animated, the Dope Sheet shows your keyframes in the context of the currently active Viewer. Whenever you change the Viewer input, the keys in the Dope Sheet move accordingly to reflect where their effect is actually seen in the Viewer:
• If you view keys before a node that moves them in time, you see them at their original time.
• If you view keys after the time manipulation, you see them shifted by the time operation.

For example, in the below script, the Transform node has been animated on frames 0, 10, 20, and 30. When the Viewer is connected to the Transform node, the Dope Sheet shows keys on those frames.

There is also a TimeOffset node in the script, set to move the input clip 5 frames forward in time. When the Viewer is connected to the TimeOffset node, the Dope Sheet shows the keys shifted by 5 frames - on frames 5, 15, 25, and 35.
Multiple Branches of Nodes in the Dope Sheet

If there are multiple paths from an animated node to the currently active Viewer, the keyframes for the animated node appear in the Dope Sheet multiple times. For example, in the below script, there are two paths from the animated Blur node to the Viewer:

- Blur > Merge > Viewer
- Blur > TimeOffset > Merge > Viewer

As a result, the keyframes for the Blur node appear in the Dope Sheet twice. Adjusting the keys in one location also updates them in the other.

Selecting Keyframes

There are several ways to select and edit different keyframes:

<table>
<thead>
<tr>
<th>Select multiple frames</th>
<th>Drag a marquee over your keyframes in the keyframe view.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add or remove keyframes from a marquee selection</td>
<td>Hold <strong>Shift</strong> and drag another marquee around some of the selected keyframes.</td>
</tr>
<tr>
<td>Select a single keyframe</td>
<td>Click it in the keyframe view. Its parent keyframe and all other keyframes depending on it are selected too. The frame number of the keyframe is displayed next to it.</td>
</tr>
<tr>
<td>Select all keyframes in one node</td>
<td>Click the node name in the hierarchy view. All the child keyframes under that node are selected.</td>
</tr>
</tbody>
</table>

If there are multiple paths from an animated node to the currently active Viewer, the keyframes for the animated node appear in the Dope Sheet multiple times. When you select them in one location, they turn orange, while the corresponding keys in other locations turn white.
**Note:** When selecting keys that appear in the Dope Sheet multiple times (using multiple paths), marquee selection is constrained to one path only.

**Copying and Pasting**

You can copy and paste keyframes in the Dope Sheet. After selecting the keyframe(s) you want to copy, use the Ctrl/Cmd+C keyboard shortcut to copy the keyframe(s), and Ctrl/Cmd+V to paste them in another location of the Dope Sheet.

When using the Ctrl/Cmd+V keyboard shortcut, pasting keyframes is relative to the location of the current frame indicator. In other words, if you have keyframes at 1, 10, and 20 and you copy this selection before pasting the selection at frame 21, new keyframes are set at frames 21, 30, and 40.

To paste keyframes on the same frame as the source keys, select the control you want to receive the keys, right-click on the Dope Sheet, and select **Edit > Paste > Paste Absolute**.

**Deleting and Adding Keyframes**

You can remove keyframes in the Dope Sheet. To delete a keyframe, just select it and press **Delete/Backspace** button or right-click on the keyframe and select **Edit > Erase**.

You can also add keyframes in the Dope Sheet to add animation where there isn't any at the moment. Add keyframes by Ctrl/Cmd+Alt+clicking on an empty spot in the Dope sheet. A new keyframe is created and you can adjust it as necessary.

**Editing Clips in the Dope Sheet**

The Dope Sheet allows you to easily reposition, trim, and slip your clips as well as set a frame range for them:

- Repositioning clips refers to changing the position, but not the content or the duration, of the clip.
- In the Dope Sheet, you can reposition clips using the Read, TimeOffset, and TimeClip nodes.
• Trimming clips refers to removing unwanted frames from the head or tail of the clip. In the Dope Sheet, you can trim clips using the Read and TimeClip nodes.

• Slipping clips refers to changing the content of the clip that is seen, but not the position or duration. In the Dope Sheet, you can slip clips using the Read and TimeClip nodes.

• Setting a frame range for a clip controls which frames a downstream AppendClip node uses from the input, which frames are displayed in the Viewer when the timeline range dropdown menu is set to Input, and which frames are sent to the flipbook. In the Dope Sheet, you can set the frame range for a clip using the FrameRange node.

Repositioning, Trimming, and Slipping Clips Using Read and TimeClip Nodes

• To reposition a clip, place the cursor over the Read or TimeClip bar in the Dope Sheet and drag left or right.

![Dope Sheet](image)

The new start and end frame numbers are displayed on either side of the bar, and the frameoffset control in the Read or TimeClip properties panel is adjusted automatically.

**Tip:** You can also reposition a clip by selecting the bar, double-clicking on the first and last frame numbers next to it, and adjusting them in the x fields that pop up.

• To trim a clip, place the cursor in the beginning or end of the bar and drag left or right.

![Dope Sheet](image)

The new start or end frame is displayed next to the bar, and an orange line appears to represent the original frame range. The framerange control in the Read or TimeClip properties panel is adjusted automatically.
Tip: You can also trim a clip by selecting the bar, double-clicking on the first or last frame number next to it, and adjusting the number in the x field that pops up.

Tip: The number of frames you can trim from either end of a clip is constrained by the clip's original frame range. However, if necessary, you can trim beyond the original range by holding Ctrl/Cmd while trimming.

If necessary, you can change how the clip displays inside the clip handles (the unused frames beyond the start or end of the clip). Adjust the before and after dropdown menus next to the frame range control in the Read or TimeClip properties to change what happens before and after the frame range limits:

- hold - select to show a still picture of the first/last frame of the frame range.
- loop - select to start over and keep looping the span of the frame range outside the first/last frame of the frame range.
- bounce - select to play the span of the frame range backwards and forwards between the frame range limits.
- black - select to display a black frame outside of the first/last frame.

• To slip a clip, first trim the clip to create clip handles. Then, place the cursor over the bottom half of the bar and drag left or right.

The clip handles move to indicate what portion of the clip is visible. The frame range and frame offset controls in the Read or TimeClip properties panel are adjusted automatically.

Tip: The number of frames you can slip a clip by is constrained by the clip's original frame range. However, if necessary, you can slip beyond the original range by holding Ctrl/Cmd while slipping.

Repositioning Clips Using TimeOffset Nodes

To reposition a clip, place the cursor over the TimeOffset bar in the Dope Sheet and drag left or right.
The new start and end frame numbers are displayed on either side of the bar, and the **time offset (frames)** control in the TimeOffset properties panel is adjusted automatically.

**Tip:** You can also reposition a clip by selecting the bar, double-clicking on the first and last frame numbers next to it, and adjusting them in the x fields that pop up.

**To Set the Frame Range Using FrameRange Nodes**

- To set both the first and the last frame for a clip, place the cursor over the FrameRange bar in the Dope Sheet and drag left or right.

The new start and end frame numbers are displayed on either side of the bar, and the **frame range** control in the FrameRange properties panel is adjusted automatically.

- To set only the first or the last frame for a clip, place the cursor in the beginning or end of the bar and drag left or right.

The new start or end frame is displayed next to the bar, and the **frame range** control in the FrameRange properties panel is adjusted automatically.
Synchronizing the Frame Range

You can synchronize the frame range of your project between the Dope Sheet and the Curve Editor. This is useful when you’re using both of them to edit your animation curves and keyframes at the same time. To synchronize the frame range:

1. Right-click anywhere on the Dope Sheet.
2. Select View > Synchronize frame range. Any changes you now make to the frame range in the Dope Sheet are applied in the Curve Editor too, and any changes you make in the Curve Editor are also reflected in the Dope Sheet. Zooming in and out in either also zooms the other view.

**Tip:** You can also do this by clicking the synchronize button at the bottom of the Dope Sheet.

Using the Curve Editor

The Curve Editor enables you to edit curves without physically entering information in the Properties pane.

Displaying an Animation Curve

To reveal an animation curve, do the following:

1. Click the animation button next to the parameter whose curve you wish to view.
2. Select Curve Editor. The Curve Editor panel appears with a focus on the selected parameter’s curve.
   - The vertical, or y axis, denotes the value of the parameter.
   - The horizontal, or x axis, denotes time (in frame units).
Displaying Curves in the Editor

To display curves in the editor, do the following:

1. In the parameter tree on the left, click the + and - signs to expand and collapse the hierarchy as necessary.
2. Click a parameter’s name to make its curve the focus of the editor. To focus on multiple curves at the same time, either:
   - Hold Shift and click on curves to create a consecutive list,
   - OR
   - Hold Ctrl/Cmd and click on individual curves to add them to the selection.
3. To display separate curves for each channel, separate the channels for the relevant control in the node’s properties panel.

The parameter tree on the left allows you to focus on any curve in the script.

**Tip:** If you want to view an AudioRead node under your animation curves, you can display it by right-clicking anywhere in the curve editor and selecting View > Audio > Draw Style > Below. For more information on the audio options, see Audio in Nuke.

Enabling/Disable a Curve in the Editor

To enable or disable a curve in the Editor, in the parameter tree on the left, click the + and - signs to expand and collapse the hierarchy as necessary.

Zooming In or Out in the Editor

To zoom in or out in the Editor, do the following:
1. Click on the area you want to zoom in on or out of.
2. Press the + button to zoom in, or the - button to zoom out,
   OR
   Scroll up with the mouse wheel to zoom in, or down to zoom out.

**Tip:** Depending on your mouse preferences, you can zoom to a custom area in the Curve Editor, middle-click on the Editor and drag to select an area with a marquee. When you release the mouse button, the Editor zooms to fit the selected area in the Editor.

### Panning in the Editor

To pan in the Editor, hold the middle-mouse button and drag over the Editor. You can also use `Alt+drag`.

### Resetting Zoom and Panning

To reset the zoom or panning:

1. Right-click on the Curve Editor.
2. From the menu that opens, select **View > Frame All**.
   OR
   Press `A` on the Editor.
   OR
   Click the middle-mouse button (dependent on your mouse preferences).
   Nuke centers the curve in the Editor, resetting the zoom.

### Centering a Portion of the Curve

To center a point of the curve in the Editor:

1. Select the points you want to center in the editor.
2. Right-click on the Editor, and select **View > Frame Selected** (or press `F` on the Editor).
   Nuke centers the selected portion of the curve in the editor. If no points are selected, Nuke centers the selected curve, or all curves.
Editing Curves

You edit curves by moving the points on the curve to new locations. If necessary, you can add more points to the curve. You can also sketch curves freely, use the usual editing functions such as copy and paste, smooth curves with filtering, interpolate curves, loop, reverse or negate curves, and use expressions to modify curves.

Adding Points to a Curve

To add points to a curve, do the following:
1. Click on the curve you want to edit. The curve turns yellow to indicate it's selected.
2. Ctrl/Cmd+Alt+click on the part of the Curve Editor you want to add a point to. You can add points both on the curve and outside the curve,
   OR
1. Right-click on the Editor and select Edit > Generate. The Generate keys dialog opens.
2. In the Start at field, enter the first frame you want to use as a keyframe.
3. In the End at field, enter the last frame you want to use as a keyframe.
4. In the Increment field, enter the frame increment you want to use between the first and the last keyframe. For example, if you want every tenth frame to be a keyframe, enter 10.
5. In the last field, enter the value you want to use for y. If you do not enter a value here, the key frames are added to the current curve without modifying the curve shape.
6. Click OK.

Selecting Points on a Curve

You can select points in the following ways:
• To select individual points, click on the point you want to select.
• To select multiple points, Shift+click on the points, or drag a marquee around them.

A box is drawn around the points to indicate they have been selected.
• To select all points, press Ctrl+A (Mac users press Cmd+A).

A box is drawn around the points to indicate they have been selected.

Moving Points on a Curve

• To move a point along either the x or y axis only, drag the point to a new location.
• To move a point in any direction, Ctrl+drag (Mac users Cmd+drag) the point to a new location. You can also nudge points using the numeric keypad arrows.

• To adjust the values of a point numerically, select the point and double-click on the x or y value that appears next to it.

• By default, when you move a point, its position on the x axis is rounded to the nearest integer. To disable this, you can right-click on the Curve Editor and select Edit > Frame Snap. You can also momentarily disable the snapping by pressing Shift while moving a point.

• To move several points at the same time, select them and drag the selection box to a new location.

To add or remove points to or from the selection box, Shift+click on the points.

To resize and scale the selection box, drag its edges. If the selection box is very narrow, you can press Ctrl/Cmd when resizing it. This allows you to resize the box in one dimension only. For example, if you have a box that's wide on the x axis but flat on the y axis, you can resize it in this way along the x axis.

To avoid accidentally moving a point inside the selection box, press Ctrl/Cmd+Shift when dragging the box to hide the points inside the box.

Adjusting the Slope Around the Points

To adjust the slope around the points, do the following:

1. Select a point on the curve. Red tangent handles appear on both sides of the point.
2. Drag the tangent handles to a new location. The curve follows the handles.

Sketching a Curve

You can sketch a curve freely by doing the following:

Press Alt+Ctrl+Shift (Mac users press Alt+Cmd+Shift) while drawing a curve on the editor. Nuke sketches a curve that follows your mouse movements.

Cutting, Copying, and Pasting

You can cut, copy, or paste any selected points, expressions, or curves, by doing the following:
1. Right-click on the Curve Editor.
2. From the menu that opens, select Edit and the editing function you want to use on the entire curve, for example:
   • Edit > Copy > Copy Selected Keys to only copy the points you have currently selected.
   • Edit > Copy > Copy Curves to copy an entire curve.
   • Edit > Copy > Copy Expressions to copy the expression that creates the curve.
• **Edit > Copy > Copy Links** to copy a curve and keep its values linked to the original curve, so that if you change the original, your changes also affect the copied curve.

Move Selected Points on the Curve

To move selected points on the curve by a fixed value:
1. Select all the points you want to move.
2. Right-click on the editor and select **Edit > Move**. The **Move Animation Keys** dialog opens.
3. In the **x** and **y** fields, define how you want to move the points along the x and y axes. For example, to shift the selected points to the right by a value of 10, enter **x+10** in the x field.

4. In the **slope** and **left slope** fields, define how you want to move the points’ tangent handles.

Smoothing the Curve

To smooth the curve with filtering, do the following:
1. Select the portion of the curve that needs smoothing.
2. Right-click on the editor and select **Edit > Filter**. The **Filter Multiple** dialog opens.
3. In the **No. of times to filter** field, specify how many times you want to filter the curve. Filtering sets new values on each point based on the average values of their neighboring points. The more filtering, the smoother the curve.

Interpolating Parts of a Curve

You can interpolate parts of a curve, by doing the following:
1. Select the point(s) between or around which you want to interpolate the curve.
2. Right-click on the Editor. Select **Interpolation** and the type of interpolation you want to use. Select:

- **Constant** to force a constant value after each selected point.

- **Linear** to use linear interpolation. This produces sharp changes at key frames and straight lines between them.

- **Smooth** to set the tangents' slopes equal to the slope between the keyframe to the left and the keyframe to the right if the selected point is between these two key frames along the y axis. If the selected point is not between these key frames and has a larger or smaller value than both key frames, the tangents' slopes are made horizontal. This ensures the resulting curve never exceeds the keyframe value.

- **Catmull-Rom** to set the tangents' slope equal to the slope between the keyframe to the left and the keyframe to the right regardless of where the selected point is located. The resulting curve can exceed the keyframe values.
• **Cubic** to set the slope so that the second derivative is continuous. This smooths the curve.

• **Horizontal** to make the tangents horizontal, setting the slope around the selected points to zero.

• **Break** to adjust the two tangents of a selected point independent of each other.

• **Before > Constant** or **Linear** to interpolate the parts of the curve that are on the left side of the first point. This option only works if you have selected the first point on the curve.

• **After > Constant** or **Linear** to only interpolate the parts of the curve that are on the right side of the last point. This option only works if you have selected the last point on the curve.
Repeating a Part of the Curve

To repeat a portion of the curve, throughout the curve, do the following:

1. Right-click on the editor and select **Predefined > Loop**. The **Loop** dialogue opens.
2. In the **First frame of loop** field, enter first frame of the portion you want to repeat throughout the curve.
3. In the **Last frame of loop** field, enter the last frame of the portion you want to repeat.
4. Click **OK**.

The shape of the curve between these frames is repeated throughout the rest of the curve. The solid line represents the actual curve, and the dotted line the original curve with the key frames.

Reversing a Curve

You can reverse a curve by right-clicking on the editor and select **Predefined > Reverse**.

This makes the curve go backward in time. Both the new curve and the original curve are displayed. The solid line represents the actual curve, and the dotted line contains the key frames that you can modify.
Negating a Curve

You can negate a curve by right-clicking on the editor and select **Predefined > Negate**.

The curve becomes the negative of the key frames. For example, a value of 5 turns into -5. Both the new curve and the original curve are displayed. The solid line represents the actual curve, and the dotted line contains the key frames that you can modify.

![Example of negating a curve](image)

Modifying a Curve with an Expression

You can modify a curve using an expression by doing the following:

1. Enter the expression in the expression field at the bottom of the Curve Editor,

   ![Expression field](image)

2. Click the **Revert** button to set the field back to previous valid expression.

   **OR**

   1. Right-click on the Editor, and select **Edit > Edit expression**.
   2. In the dialog that opens, type the expression you want to use for the curve, for example, \( \sin(x)/x \).

   ![Expression editor](image)

3. Click **OK**.
Compositing Viewers

A Viewer node displays the render output of any connected process nodes in the Viewer panel and does not edit any data. Viewer nodes allow you to quickly assign the right values to parameters as they allow you to edit in context - that is, edit a given node's parameters upstream in a script while viewing the effect of those changes downstream.

Nuke's Viewer output is labeled with the bounding box size and the format. The bounding box is the area the Nuke treats as containing image data and the format is the resolution of the input image. These two are the same by default so that the output is the same as the resolution.

See Reformatting Elements and Adjusting the Bounding Box for more information on format and bounding boxes.

You can place as many Viewer nodes in a script as you wish, which allows you to simultaneously view multiple outputs. You can also pipe the output from up to ten process nodes into a single Viewer node, and then cycle through the various displays. This allows you to easily compare an image before and after processing by a given effect.

Note: The maximum image size the Viewer can display is 64k x 4k (or the equivalent number of total pixels at other resolutions). Make sure though, that you have sufficient RAM memory available if you want to use the maximum image size.
Adding Viewer Nodes

Viewers have corresponding nodes that appear in the Node Graph. These nodes do not produce output for rendering; they generate display data only. You can connect Viewer nodes as described in Working with Nodes. In practice, it is faster to use the Viewer keyboard shortcuts described below.

To add a Viewer node:
1. Select the node that you want to view the output of.
2. Do one of the following:
   • Using the menu bar, choose Viewer > Create New Viewer.
   • Using the Toolbar, choose Image > Viewer.
   • Using a keyboard shortcut, press Ctrl/Cmd+I.

Nuke connects a Viewer node to the node you selected in step 1, and displays the output of the node in the Viewer panel. You can also insert a Viewer node and set up its first connection by simply pressing 1 over the Node Graph.

Connecting Viewer Nodes

After you add a Viewer node to the script, you can quickly pipe any process node's output to it simply by selecting the process node then pressing any number key. Doing so, pipes the output to one of the ten input ports available on every Viewer node (the 0 key represents the tenth slot).
Toggling Views

If a Viewer node has multiple inputs, like the one depicted above, you can press the up or down arrow keys to quickly cycle through the views (your cursor needs to be in the Viewer window). To view a particular node press the number key \((1, 2, 3...0)\) corresponding to the pipe number whose contents you wish to view.

Panning and Zooming the Viewer Window

Nuke offers several ways, including some useful keyboard shortcuts, to focus on the area of the Viewer you need to see.

Panning the Frame

To pan the frame, hold the middle-mouse button and drag on the display (you can also use Alt+drag). The frame follows the mouse pointer.

**Note:** In many Linux windows managers, the Alt key is used by default as a mouse modifier key. This can cause problems in 3D applications where Alt is used for camera navigation in 3D environments. You can use key mapping to assign the mouse modifier to another key, such as the (Super or Windows logo) key, but the method changes depending on which flavor of Linux you're using. Please refer to the documentation on key mapping for your particular Linux distribution for more information.

Re-centering the Frame

You can re-center the frame by clicking the middle-mouse button or pressing F.

Zooming In and Out

To zoom in on the frame:
1. Move your pointer over the area of the display on which you want to zoom.
2. Drag the mouse while pressing the middle mouse button and the left mouse button.
   OR
   Press the plus button (\(+\)) repeatedly until the frame attains the desired scale.
OR
Select **zoom in** from the zoom dropdown menu in the top right corner.

To zoom out from the frame:
1. Move your pointer over the area of the display from which you want to zoom.
2. Drag the mouse while pressing the middle mouse button and the left mouse button.
   OR
   Press the minus button (-) repeatedly until the frame displays at the desired scale.
   OR
   Select **zoom out** from the zoom dropdown menu in the top right corner.

**Restoring the Zoom to 100%**

You can restore the zoom to 100% by pressing **Ctrl/Cmd+1**.

**Showing / Hiding Floating Viewers**

You can hide and show floating Viewers using a handy keyboard shortcut.

To hide or show a floating Viewer, press ‘ (the backtick key).

**Showing / Hiding Viewer Toolbars**

To hide or show the top toolbar, press `{`.

To hide or show the bottom toolbar, press `}`.

**Locking the Viewer Zoom Level**

You can choose to lock the Viewer zoom level, so that it doesn’t change when you switch between inputs of different sizes. Right-click on the Viewer, and select **Prevent Auto Zoom** to toggle between maintaining the same zoom level for all inputs (on) and changing it according to on-screen image dimensions (off). Alternatively, you can also press **Alt+Z** on Viewer, or toggle **prevent auto zoom** in the Viewer settings.
Viewing Overscan in the Viewer

If you want the Viewer to show overscan, that is any pixels extending beyond the left/right or top/bottom of the frame, you can check the `showoverscan` box in the Viewer settings or check the option in the Viewer right-click menu. Viewing pixels beyond the edges of the frame can be useful if some of your nodes need to have access outside the frame. You can also control the amount of overscan shown by specifying the pixel amount in the `overscan` field.
Using the Viewer Controls

A Viewer’s on-screen controls let you navigate the timeline, display channels, zoom, choose cameras (3D mode), and create display wipes and composites.

Compositing Viewer Controls

There are many useful tools at the top of the Viewer, some of which allow you to select layers and channels, adjust gain and gamma, and zoom and scale down the image in the Viewer.

The tools at the bottom of the Viewer allow you to adjust the playback settings, including setting the frame range, selecting the playback mode, and locking the Viewer playback range.

Drag the orange marker along the timeline to quickly cue to a specific frame. The number of the current frame appears below the center of the timeline. You can also cue to a frame by typing its number directly into this field.

**Tip:** The current frame and in an out point fields also accept simple mathematical functions, such as +/-20 to jump forward or backward 20 frames.
By default, Nuke automatically adjusts the timeline of every Viewer window to show the frame range defined in your Project Settings. If no frame range is defined, the frame range of the first image you read in is used as the global frame range.

Viewer timeline controls also have a frame range source dropdown menu that you can use to define where the timeline gets its frame range from. You can set this menu to Global, Input, or Custom. Global is the default setting described above.

The playback rate field (frames-per-second) displays the project’s playback speed. Nuke attempts to maintain this speed throughout playback, although this adjusts depending on the resolution of the imagery and your hardware configuration.

**Note:** The asterisk (*) denotes the playback speed selected using the Project Settings > Root > fps control.

To have the Viewer adjust the timeline to show the “in” and “out” frames of the current input clip, select Input from the frame range source dropdown menu. The number of the first frame in the clip is shown in the left end of the timeline and the number of the last frame in the right end. If you change the input of the Viewer, the frame range on the timeline is adjusted accordingly.

To manually adjust the frame range for the current Viewer window, pan and zoom on the timeline until you see the desired frame range and Custom becomes selected in the frame range source dropdown menu. Alt+drag to pan, and MMB+drag to zoom in. You can also zoom in on or out of the timeline using the mouse wheel. To reset the zoom, press the middle mouse button over the timeline.

To adjust the playback range for the current Viewer window, click+drag the red playback range markers on the timeline to a new “in” and “out” frames as shown below, or enter a new playback range in the playback range field and press the frame range lock button.

To toggle between the new playback range and the visible timeline range, click the lock frame button again.

The fps field (frames-per-second) initially displays the project’s playback speed. Nuke attempts to maintain this speed throughout playback, although this adjusts depending on the resolution of the imagery and your hardware configuration.

The following table lists the functions of the playback buttons:
<table>
<thead>
<tr>
<th>Buttons</th>
<th>Functions</th>
</tr>
</thead>
<tbody>
<tr>
<td>🔄 🔄</td>
<td>The <strong>Play backward</strong> and <strong>Play forward</strong> buttons play the sequence backward or forward at the script's frame rate. When you press a play buttons, it toggles to a stop a button.</td>
</tr>
<tr>
<td>🔄 🔄</td>
<td>The <strong>Back 1 Frame</strong> and <strong>Forward 1 Frame</strong> buttons cue the sequence to the previous or next frame.</td>
</tr>
<tr>
<td>🔄 🔄</td>
<td>The <strong>Previous keyframe</strong> and <strong>Next keyframe</strong> buttons cue the sequence to the script's previous or next keyframe.</td>
</tr>
<tr>
<td>🔄 🔄</td>
<td>The <strong>First frame</strong> and <strong>Last frame</strong> buttons cue the sequence to the first and last frame.</td>
</tr>
<tr>
<td>🔄 🔄</td>
<td>The <strong>Frame Increment</strong> field allow you to specify the number of frames by which the Previous increment/Next increment buttons cue the sequence. This is set to 10 frames by default.</td>
</tr>
</tbody>
</table>

The **Playback Mode** button lets you control how many times and in what direction the Viewer plays back the sequence. Click the button to toggle between the following modes:

<table>
<thead>
<tr>
<th>Button</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>🔄 Repeat</td>
<td>Repeatedly plays the sequence in a loop.</td>
</tr>
<tr>
<td>🔄 Bounce</td>
<td>Repeatedly plays the image back and forth from head to tail.</td>
</tr>
<tr>
<td>🔄 Stop</td>
<td>Plays once through the section between in and out point and stops at the out point. If these are not marked, then it plays from the beginning to the end of the sequence.</td>
</tr>
<tr>
<td>🔄 Continue</td>
<td>Plays once from the beginning to the end of the sequence, ignoring any in and out points.</td>
</tr>
</tbody>
</table>

**Jumping to a Specific Frame**

You can move quickly to a specific frame on the timeline by choosing **Viewer > Go to frame** (or by pressing **Alt+G**), entering a frame number in the dialog that appears, and clicking **OK**.
Synchronizing Viewer Playback

The 'Lock All Viewer Playback Ranges' button allows you to toggle synchronized playback of Viewer windows. By default, all Viewers are locked - that is, if you cue to a frame in one Viewer, all other Viewers follow suit.

When the lock icon changes from a closed lock to an open lock, that Viewer’s playback becomes independent of other Viewers, and not cued to the other Viewers.

Pausing the Display

The 'Pause Viewer Update' button stops the Viewer from updating and holds the last frame rendered. To reactivate display rendering for all frames, press the button again.

You can click the 'Refresh Viewer' button (or press U) to manually update the display while keeping Viewer paused.

Displaying a Single Channel

You can press the R, G, B, and A keys on your keyboard to display the red, green, blue, and alpha channels respectively. Or, you can also select a channel from the RGB dropdown menu in the top-left corner.
Press one of the channel keys again to toggle back and display all channels.

**Tip:** If you press *Shift* while selecting the channel, your selection only affects the currently active input of the Viewer node. This way, you can display different channels from the Viewer's different inputs. For example, when keying it can be useful to view the RGB channels from one input and the alpha channel from another, and toggle between the two.

To achieve this, do the following:

1. Create a Viewer with several inputs.
2. Activate one of the inputs by pressing its number (for example 1) on the Viewer.
3. Press *Shift* and select RGB from the channel dropdown menu.
4. Activate another input (for example, input 2) by pressing its number on the Viewer.
5. Press *Shift* and select A from the channel dropdown menu.
6. Toggle between the inputs by pressing their numbers or the up and down arrow keys.

### Layer and Channel Dropdown Menus

The layer dropdown menu lets you choose a set of color channels to display in the Viewer. By default, this is set to display the rgba layer, but you can choose any layer in the data stream.
The channel dropdown controls which channel appears when you view the “alpha” channel. The default setting displays the alpha channel when you press the A key, but you can change this by selecting any channel in the data stream, as shown below.

Displaying Alpha Mattes

When you've read in an image that has an alpha channel, you can display the alpha channel as a red overlay on top of the image's red, green, and blue channels.

To Display an Image's Alpha Channel on its RGB Channels

1. Select **Image > Read** to read in an image.
2. Connect a Viewer node to the Read node.
   By default, Nuke displays the red, green, and blue channels in the Viewer.
3. Click on the Viewer to make sure it's the currently active panel.

4. Press M.

   Nuke displays the image's alpha channel as a red overlay on top of the RGB channels.

5. To return to the RGB display, press M again.

**Viewer Info Bar**

The info bar displays information about the image format, bounding box, and pixel underlying the pointer, a sampled pixel or region of pixels. From left to right, the indicator displays the following about the current image and pixel or sample: image resolution and the size of the bounding box, the pixel's x and y position and its Red, Green, Blue, and Alpha values, and other values depending on the color type you have selected from the color type menu on the right.

<table>
<thead>
<tr>
<th>Color Type</th>
<th>Description</th>
<th>Values for RGB 0.1, 0.1, 0.1, 1.0 (Dark Gray)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spotmeter</td>
<td>Colorspace values measured in f-stops at 1/48th of a</td>
<td>f/5.9 @48th sec</td>
</tr>
<tr>
<td>Color Type</td>
<td>Description</td>
<td>Values for RGB 0.1, 0.1, 0.1, 1.0 (Dark Gray)</td>
</tr>
<tr>
<td>------------</td>
<td>-------------------------------------------</td>
<td>-----------------------------------------------</td>
</tr>
<tr>
<td></td>
<td>second shutter speed.</td>
<td></td>
</tr>
<tr>
<td>8-bit</td>
<td>RGB colorspace values.</td>
<td>25, 25, 25, 255</td>
</tr>
<tr>
<td>8-bit Hex</td>
<td>The same colorspace values as 8-bit, but displayed in hexadecimal.</td>
<td>1A 1A 1A FF</td>
</tr>
<tr>
<td>log</td>
<td>Logarithmic colorspace values.</td>
<td>397, 397, 397</td>
</tr>
<tr>
<td>HSVL</td>
<td>The default setting, shows colorspace values as Hue, Saturation, Value, and Level.</td>
<td>0, 0.00, 0.10, 0.10000</td>
</tr>
</tbody>
</table>

You can sample a single pixel from the Viewer by pressing Ctrl/Cmd while clicking, a region from the Viewer by pressing Ctrl/Cmd+Shift while dragging, a single pixel from the node’s input by pressing Ctrl/Cmd+Alt while clicking, and a region from the node’s input by pressing Ctrl/Cmd+Alt+Shift while dragging.

To cancel a pixel selection, hold Ctrl/Cmd and right-click in the Viewer.

**Adjust Display Gain and Gamma**

The gain and gamma sliders let you adjust the displayed image, without affecting your final output. These controls are useful for tasks like spotting holes in mattes. You can boost or reduce gain by entering a multiplier (exposure value), dragging on the slider, or using the F-Stop arrows. Boost or reduce gamma by entering a gamma level or dragging the gamma slider.

The gain and gamma toggle buttons let you switch between the default values of 1 (no change) and the last gain and gamma adjustments you made in the Viewer.

Press the ‘Clipping Warning’ button to apply stripes to all pixels outside the range 0.0 to 1.0.
2D / 3D Toggle and Camera Controls

The 2D / 3D dropdown menu lets you toggle between 2D and 3D display modes in the current Viewer. This menu also lets you choose between different orthographic (non-perspective) views when working in the 3D mode.

The camera dropdown menu on the right lets you choose which camera to look through when multiple cameras exist in your 3D scene. For more information on these controls, see 3D Compositing.

Selection Modes

Nuke features three selection modes, Rectangle, Ellipse, and Lasso. Selection modes work the same in all Viewer contexts, whether you're in 2D selecting Roto splines or 3D selecting vertices, faces, or objects. The selection mode dropdown is located above the Viewer at the top-right.
Each mode uses the same selection modifiers:

- **Shift** - additive selection. Holding **Shift** and selecting vertices, faces, or objects adds the selection to any existing selection.

- **Alt + Shift** - subtractive selection. Holding **Alt + Shift** and selecting vertices, faces, or objects removes the selection from any existing selection.
Monitor Output Toggle

The monitor output button – which automatically appears when you connect an external monitor – allows you to preview the current Viewer image on an external broadcast video monitor.

**Note:** For more information, see *Previewing on an External Broadcast Video Monitor.*
Controlling Zoom and Resolution

Nuke includes a dedicated set of controls to determine what is rendered in the Viewer and at what resolution, allowing you to work on specific areas of the image at the required quality level while keeping the interface responsive.

Using the Zoom Menu

The Zoom dropdown menu lets you select the magnification factor by which the current image is displayed. This menu also shows the keyboard shortcuts to press to quickly switch between the different zoom settings.

Proxy Mode

Nuke can generate low-res proxies for displayed frames as needed. Press Ctrl/Cmd+P or click the proxy mode toggle button on the Viewer to activate the proxy display mode.

By default, the proxy scale is set to 0.5. You can change the proxy scale in the Projects Settings, which you can open by selecting Edit > Project Settings (or pressing S).
Proxy display resolution defined on the Project Settings properties panel.

You can also read in rendered proxies using the Read nodes’ controls. The proxy file does not need to match the proxy resolution in use. Depending on your settings, either the full-res or proxy file is scaled to the required proxy size. For more information, see Read Nodes and Proxy Files.

**Lowering the Display Resolution of Individual Viewers**

Viewers also have a dropdown menu that allows you to easily switch to lower display resolutions, regardless of whether you have activated proxy mode or not. Using this multiplier setting, you can, for example, change the display resolution of an individual Viewer to 50% of the current (be it full-size or proxy) resolution. This is useful if you want to have Nuke display your images more quickly without having to touch the project settings. It also comes in handy if you have just a few very large plates in your script, as you can choose to use lower resolutions when viewing just these plates.

To Lower the Display Resolution of Individual Viewers:

From the Viewer’s down-rez dropdown menu, choose the factor by which you want to lower the display resolution.

For example, if you have a 4K plate and are using a proxy scale of 1:2, your plate is still 2K even in the proxy mode. Setting the down-rez factor to **1:2** in the Viewer scales the plate down further to 50% of the proxy resolution, that is to 1K. This gives you much faster (but less accurate) feedback.

**Pixel Aspect Ratio**

The pixel aspect ratio determines whether your images are displayed using square or rectangular pixels. By default, the Viewer uses the pixel aspect ratio defined in your project settings. To see the current setting, select **Edit > Project Settings** (or press **S**).
For example, a pixel aspect ratio of 2 accurately displays anamorphic footage the way it is projected, as shown in the image:

The Viewer uses the pixel aspect ratio defined for the script.

If you want to ignore the pixel aspect ratio, you can toggle it by pressing Ctrl/Cmd+Shift+P over the Viewer window.

Press Ctrl/Cmd+Shift+P over the Viewer window to ignore the pixel aspect ratio.

Full-frame processing

By default, when you display or play through a sequence in the Viewer, Nuke only calculates and caches the scanlines for the visible area. It's not caching the full frame, but a subset of the scanlines you're viewing. For example, if you have set the zoom level to be +4, Nuke is only caching 1/4 of the scanlines that make up the frame. In a lot of cases, this allows for faster playback and may be what you want.

However, if you play through a sequence and then pan around or change the zoom level, Nuke has to calculate the scanlines it didn't calculate before. This may not be what you want if you are performing tasks that require you to constantly pan around or zoom in and out of the plate (such as paint and roto). If this is the case, you can click on the **full frame processing** button in the controls at the top of the Viewer to force Nuke to process every scanline in the image. Compared to the default mode,
this may take slightly longer initially and requires more space in the Viewer cache, but once Nuke has cached the frames you require, you can pan around and change the zoom level without breaking the cache or affecting playback.

Region of Interest (ROI)

The ROI button lets you enable rendering only through a region of interest - a portion of the image you explicitly select. This is useful for quickly viewing render results in a process-heavy script.

To Define a Region of Interest:

1. Click on the ROI button in the Viewer controls. The ROI overlay appears.

2. Drag to resize and move the ROI overlay as necessary.

To Free-Draw a Region of Interest

1. Over the Viewer, press Alt+W once (do not hold the keys down). The ROI button turns red, but the ROI overlay does not appear. This allows you to freely draw your own ROI rather than adjust the default overlay.

2. Drag a marquee to draw the region of interest where you need it.
To Clear a Region of Interest:

1. After you've set a region of interest, you can clear it by pressing `Alt+W` over the Viewer. You can then drag a new marquee to define a new region of interest.

2. To turn off the feature and update the whole Viewer with the recent changes, click the ROI button again (or press `Shift+W`).
Viewer Overlays and Input Processes

Nuke's Viewer overlays and render modes help you to position elements correctly and compare shots in the Node Graph. You can also use Input Process and Viewer Process to modify the image from the viewed node before it is displayed on your monitor.

Guides and Masks

The Viewer guides and masks assist with placing effects and text within the current format. For example, text placed with the Title Safe guide is visible to the audience. Use the Guideline and Mask dropdown menus to select the required overlays.

Note: Guides and masks are not applied at render, they are simply Viewer overlays.

Guidelines are intended to highlight areas of the current format that won't appear in the final render. By default, no guidelines are selected. You can select any of the following guideline options:

- title safe - any text intended for the audience should reside within this zone.
• **action safe** - any visual elements intended for the audience should reside within this zone.

• **format center** - overlays a crosshair in the center of the format area.
• **format** - any formatting changes must be applied to the area outlined in red.

**Tip:** The above guideline options also exist in the Viewer Settings. Press S on the Viewer to display its settings and adjust the **safe zone** and **format center** controls.

Masks can be used to simulate a particular format, for example, **4:3** or **16:9**. You can also choose the mask overlay type.

• **none** - no masking is applied to the Viewer. This is the default state.
• **lines** - any mask applied is highlighted using a pair of lines in the Viewer.

• **half** - any mask applied is highlighted using semi-transparent shading.

• **full** - any mask applied is highlighted using black shading.

• **blanking ratio** - select the masking ratio applied to the Viewer, for example, 4:3 or 16:9.

**Tip:** The above mask options also exist in the Viewer Settings. Press S on the Viewer to display its settings and adjust the **mask region outside ratio** and **mask mode** controls.
Adding Custom Guides and Masks

You can add custom guides and masks to the Viewer dropdowns by creating a file called `custom_guides.py` and placing it in your `.nuke` folder.

Tip: For information on locating your `.nuke` folder by platform, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.

The guides and masks that ship with the application are kept in the Nuke bundle, here: `<installation_directory>/pythonextensions/site-packages/custom_guides.py`

Copy the `custom_guides.py` file to your `.nuke` folder and then add the guides and masks you require to the existing Python code inside the `.py` to append them to the Viewer dropdowns. For example:

```python
guides.Guide("myGuide", 0.75, 1, 0.8, 0.3, guides.kGuideMasked)
```

OR

```python
guides.MaskGuide("5:3",5.0/3.0)
```

Using the Viewer Composite Display Modes

The wipe control provides an option for displaying a split-screen of two images, which can help you compare before and after versions for color correction, filtering, and other image manipulation. This control also includes display compositing options to overlay different images.

To Display a Comparison Wipe:

1. Select a node in your script and press 1 to display its output in the Viewer.
2. Select the node you want to compare and press 2.
   
   The 2 keystroke connects the image to the Viewer (assigning the next available connection, number 2).
3. From the A and B dropdown menus on top of the Viewer, select the images you want to compare.
   
   The menus display a list of nodes most recently connected to the Viewer.
4. From the Viewer composite dropdown menu in the middle, select `wipe`. 
The two images are displayed split-screen in the Viewer. You can view their details in the A and B information bars at the bottom of the Viewer.

5. Drag the handles of the crosshair to adjust the wipe:
   • Drag the crosshair center to change its position.
   • Drag the long handle (on the right) to rotate the wipe.
   • Drag the “arc” handle to cross-dissolve the second image.

6. When finished with the split-screen, select none (-) from the Viewer composite dropdown menu.

Tip: If you press **Shift** while selecting a channel, your selection only affects the currently active input of the Viewer node. This way, you can display different channels from the
Viewer’s different inputs. For example, when keying it can be useful to view the RBG channels from one input and the alpha channel from another, and toggle between the two.

The display composite options - **over**, **under**, and **minus** - can also be selected to overlay two images. When the two images are 2D, this allows you to create a quick comp.

When one image is 2D and the other is a 3D node, you can use **under** to line up the wireframe preview with the 2D reference, and see how the 3D matches prior to a full render.

One example of this is when you want to preview a wireframe 3D scene with a background plate that you are trying to match, as shown below. For more information, see the 3D Compositing chapter.

Special thanks to [belowtheradar TV](https://www.belowtheradar.tv) for use of the above footage.

**Input Process and Viewer Process Controls**

**Input Process** and **Viewer Process** operations can be used to modify the image from the viewed node before it is displayed on your monitor. Both only affect the Viewer in which they are activated and do not affect your rendered output. Input Process is a legacy system which uses a node instantiated in the Node Graph to process the image. This is handy for script-specific, temporary, or experimental use, but can be error prone due to the node accidentally being deleted or changed and is limited to a single node. The Viewer Process system allows a gizmo (or compiled node) to be registered from the Python programming language at start-up. The registered item appears in a
dropdown menu in the Viewer and the node is instantiated internally within the Viewer when the item is selected so there is no danger of accidental deletion or modification. This also enables multiple Viewer Processes to be registered at different points of start-up (as Nuke works through the NUKE_PATH menu.py files).

The Viewer settings contain an option for the Input Process to be applied before or after the Viewer Process, so the two may be used in conjunction, for instance, with the Input Process applying a projection mask after the Viewer Process applies a film look profile. While you could combine the two into a single Viewer Process node, it can be advantageous to keep operations separated. Having both the Viewer Process and Input Process available provides a great deal of flexibility.

You can create an Input Process by creating a node in the Node Graph and naming it as an Input Process using Nuke’s Edit menu. Once an Input Process has been named, the IP button appears in the Viewer controls. When the IP button is activated, any image you view is passed through the Input Process.

Unlike Input Processes, Viewer Processes are registered using Python. They can be session independent and always appear in the Viewer’s Viewer Process dropdown menu. There are two predefined Viewer Processes, sRGB and rec709, but you can also build and add your own. When a Viewer Process is selected from the Viewer Process dropdown menu, any image you view is passed through that Viewer Process.

Whenever possible, the Input Process and Viewer Process are executed on the GPU. 1D LUT and 3D LUT (Vectorfield) have GPU implementations, so the built-in Viewer Processes run on the GPU (unless gl buffer depth has been set to byte in the Viewer settings, in which case all processing is done on the CPU). To get the GPU’d versions of the nodes for use in a custom Viewer Process gizmo, press x over the Node Graph, enter ViewerGain, ViewerGamma, or ViewerClipTest in the command entry window, and press Return.

The following table lists the differences between an Input Process and a Viewer Process.

<table>
<thead>
<tr>
<th>Input Process</th>
<th>Viewer Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set by selecting the node in the Node Graph and choosing <strong>Edit &gt; Node &gt; Use as Input Process.</strong></td>
<td>Registered using Python.</td>
</tr>
<tr>
<td>Activated using the IP button in the Viewer controls.</td>
<td>Activated using the Viewer Process dropdown menu in the Viewer controls.</td>
</tr>
<tr>
<td>Requires that the node exists in the Node Graph. Can quickly and easily be modified by artists. Can also be accidentally deleted, disabling the effect.</td>
<td>Is defined in a text file called menu.py that is run at start-up. Accessible for artists, but not likely to be accidentally modified or deleted.</td>
</tr>
<tr>
<td><strong>Input Process</strong></td>
<td><strong>Viewer Process</strong></td>
</tr>
<tr>
<td>-------------------</td>
<td>-------------------</td>
</tr>
<tr>
<td>Script dependent. Unless your Input Process node is saved in the template.nk file that is loaded at start-up, the Input Process is lost when you restart Nuke.</td>
<td>Session independent. The Viewer Processes registered in menu.py are always available in each new session of Nuke.</td>
</tr>
<tr>
<td>There can only be one Input Process at a time. Setting a new Input Process overrides any previously used Input Process.</td>
<td>There can be an unlimited number of Viewer Processes available in the Viewer Process dropdown menu. For example, it is possible to register Viewer Processes in any menu.py file at start-up, so Viewer Processes can be added at any directory in your NUKE_PATH.</td>
</tr>
<tr>
<td>Useful for temporary or non-critical viewing options that you want in the current shot for convenience, or for testing Viewer Processes before registering them. Can also be used for other things, such as field charts or masks that may be switched on or off and changed around in the shot.</td>
<td>Useful for viewing options that you often need or that should not be modified by artists on a shot-by-shot basis.</td>
</tr>
</tbody>
</table>

**Note:** Note that Input Processes and Viewer Processes are part of a built-in, fixed pipeline of nodes that are applied to images before they are displayed in the Viewer. This pipeline is either:

- gain > Input Process > Viewer Process > gamma > dither > channels > cliptest (if input process order has been set to before viewer process in the Viewer settings)
- OR
- gain > Viewer Process > Input Process > gamma > dither > channels > cliptest (if input process order has been set to after viewer process in the Viewer settings).

However, depending on what the Input Process and Viewer Process are doing, the order in the built-in pipeline may not be the correct order. Therefore, if your Input Process or Viewer Process have controls that also exist for the Viewer, such as float controls named gain, gamma, or cliptest, then the Viewer drives them from the corresponding Viewer controls and does not do that image processing itself. This allows you to implement these controls in your Input Process or Viewer Process node/gizmo using whatever nodes and order you want. If your Input Process and Viewer Process do not have these controls, then the Viewer applies the effects in its normal way according to the built-in pipeline.

In the built-in pipeline, dither is applied to diffuse round-off errors in conversion of floating
point data to the actual display bit depth. Although the clip test is drawn at the end, it is computed on the image as input to the Viewer.

**Note:** By default, the predefined Viewer Processes, sRGB and rec709, affect all channels. However, if you want them to only affect the red, green, and blue channels, you can activate **apply LUT to color channels only** in the individual Viewer Settings or on the **Viewers** tab of the Preferences.

### Input Process Controls

To activate or deactivate the effect of an Input Process, click the IP button in the Viewer controls. Note that the IP button only appears if the **input process** field in the Viewer settings is not empty. The button is also only enabled when a node in the Node Graph is set as an Input Process.

To open the Viewer settings, press S on the Viewer, or select **Viewer Settings** from the Viewer’s right-click menu. By default, **input process** is set to **VIEWER_INPUT**. If a node called VIEWER_INPUT exists in the Node Graph, it is automatically used as the input process for the Viewer. This ensures backwards compatibility with pre-5.2 scripts.

However, the Input Process node does not have to be named VIEWER_INPUT. You can use any node as an Input Process. Do the following:

1. Select the node in the Node Graph and choose **Edit > Node > Use as Input Process**.
   Alternatively, you can press S on the Viewer to open the Viewer settings and enter the name of the node in the **input process** field.

2. In the Viewer settings, you can also define whether the Input Process is applied before or after the Viewer Process currently in use. To do so, set **input process order** either to **before viewer process** or **after viewer process**.

The Input Process node should not be connected to other nodes in the Node Graph. If you attempt to connect it, an error is displayed in the Viewer. If you delete the Input Process node from the Node Graph, the effect of the Input Process is disabled.

### Viewer Process Controls

To activate a Viewer Process, select it from the Viewer Process dropdown menu in the top right corner of the Viewer. Any images you now view using this Viewer are passed through the selected
Viewer Process.

Nuke includes the following predefined Viewer Process gizmos: sRGB, rec709, and rec1886. By default, sRGB is used because it is a best-guess default for a typical computer monitor.

In addition to using the predefined Viewer Processes, you can also add your own by registering a node or gizmo as a Viewer Process. You can register as many Viewer Processes with custom Viewer LUTs as you like. For more information on creating and registering custom Viewer Processes, see Creating Custom Viewer Processes.

All available Viewer Processes (both predefined and custom ones) appear in the Viewer Process dropdown menu in the Viewer controls. To disable the use of a Viewer Process, select None from the Viewer Process dropdown menu.

To open the properties panel of the currently active Viewer Process, select show panel from the Viewer Process dropdown menu.

Note that if the control you want to adjust has the same name as any of the Viewer controls (for example, gain or gamma), you should adjust the control on the Viewer. This drives the control in the following:

• the Input Process’ controls if an Input Process is in use
• in the Viewer Process’ controls if no Input Process is in use.

Tip: If you want to render out a file with the Viewer Process effect baked in, you can select Edit > Node > Copy Viewer Process to Node Graph to create an instance of the Viewer Process node in the Node Graph.
Using the File Browser

Whenever you load or save files in Nuke, you’ll see a browser similar to the one shown in the image below. The directory navigation buttons let you create or access the directory from which you wish to read or write data.

The navigation controls let you move through the directory structure, bookmark favorite directories, and create new directory folders.

Windows only: You can show/hide the drives that Windows auto creates by right-clicking the required drive, selecting Show Defaults, and checking or unchecking the drive.
To Use the Navigation Controls

• Click the **Create New Directory** button to create a new directory at your current position in the file hierarchy.

• Click **Up one directory** to go up one directory closer to the root.

• Click **Previous directory** to go back one directory.

• Click **Next directory** to go forward one directory.

• Click **Home** to access the directory defined as your local working directory.

• Click **Root** to ascend to the very top of your local drive or server's file hierarchy.

• Click **Nuke** to access the directory you (or your system administrator) defined as your network working directory.

• Click the + button to add a directory bookmark.

• Click the edit button to edit the name or path name to a bookmark.

• Click the - button to remove a directory bookmark.

Path Name Field

The path name field displays the current directory path, lets you navigate to a new path, and also enter a file name for scripts and rendered images.

To Use the Path Name Field:

1. To navigate to a directory, type the path name in the field.
2. To enter a script name, browse to a directory path and enter the file name after the displayed path.

To limit the file list to specific file types, use the **filter** dropdown menu and **Sequences** checkbox.

To Use the Filter Dropdown Menu and Sequences Checkbox

• Select *.nk to display only Nuke script files.
• Select * to display all files (except hidden files), regardless of whether they're associated with Nuke.
• Select .* to display all files, including hidden files.
• Select */ to display directory names, but not their contents.
• Check Sequences to display image sequences as single titles, as in fgelement.####.cin 1-50 rather than fgelement.0001.cin, fgelement.0002.cin, fgelement. 0003.cin, and so on.

Note: File sequences with no file extension (for example, fgelement.0001, fgelement.0002, fgelement.0003, and so on) are not displayed as single titles the first time you view the directory in the File Browser. However, they are displayed as single titles once you have navigated to another directory and back again.

Note: By default, Nuke may not be able to display custom file extensions (for example, .cext) as single titles. To fix this, you can register your custom file extension as a sequence type using Python:
1. Create a file called init.py in your plug-in path directory if one doesn't already exist. For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.
2. Open the init.py file in a text editor and add an entry in the following format (replacing cext with your custom file extension):
nuke.addSequenceFileExtension("cext")

• You can also split incomplete sequences into separate Read nodes using the split seq checkbox.

To Preview Files in the File Browser

1. Click the black arrow in the top right corner of the file browser.

The file browser expands to include a small viewer.
2. Select the file you want to preview. Nuke displays the file in the file browser.
To Select Multiple Files with the File Browser

1. Browse to the folder where the files are located.
2. Ctrl+click on all the files you want to open to select them (Mac users Cmd+click).
3. You can open files from multiple directories by clicking Next and browsing to the next file location.
   As you browse around, files that you have previously selected appear highlighted in the browser.
4. Click Open.
   Nuke opens all the files you selected.

To Open Incomplete Sequences in Separate Read Nodes

1. Locate an incomplete sequence.
2. Check split seq and select all the resultant files.
3. Click Open to open the sequence in separate Read nodes.
Undoing and Redoing

Nuke generally gives you an undo history that extends back to the first action of the application’s current session.

- To undo an action in the workspace, select **Edit > Undo** (or press **Ctrl/Cmd+Z**). Repeat as necessary.
- To redo an action in the workspace, select **Edit > Redo** (or press **Ctrl/Cmd+Y**). Repeat as necessary.
- To undo a change in a properties panel, click the **Undo** arrow button in the properties panel.
- To redo a change in a properties panel, click the **Redo** arrow button in the properties panel.
- To undo all changes made after the properties panel was opened, click the **Revert** button.

OR

Right-click on the properties panel and select **Revert knobs**.
- To set all controls back to their default values, right-click on the properties panel and select **Set knobs to default**.
Progress Bars

Nuke displays a progress bar for each active task it performs. By default, progress bars appear in a pop-up dialog, but you can also display them in the Progress panel. To do so, click on a content menu button and select **Windows > Progress**. This opens a Progress panel. The next time you get a progress bar, it appears in the panel. If you delete the panel, progress bars appear in a pop-up dialog again.

![Progress Bar Screenshot](image)

If you want to have the Progress panel appear in the same position in the future, you can save it as part of a workspace. For more information, see [Customizing Workspaces](#).
Handling Errors

Sometimes things may not go as you planned and you might face an error message in Nuke. When this happens, an error alert displays in the Viewer and on the node that has a problem in the Node Graph.

You can choose to view the error message itself in the Error Console tab next to the Properties pane. If you can't see the Error Console, click the content menu button and select Windows > Error Console to display it. If you have an error in a Read or a Write node, or you are missing a plug-in, the error message also displays in a pop-up window.

If you see an error alert on your node or in the Viewer, you can click the Error Console tab to open it and view the error message.

In the Error Console error list, you can double-click on a message and, if possible, Nuke takes you to the control panel of the node that's in error. This isn't always possible because of the nature of the error. You can also click the clear output button on the Error Console to clear all error messages on the tab.

**Note:** If you have a node in your Node Graph that is producing an error, but it's not connected to the node tree, Nuke won't show a pop-up error message for the node and you can still view the resulting image in the Viewer, if you have one connected. This enables working without having to stop to close error messages in the event that you have erroring nodes in the script that aren't connected to your node tree.
Using Nuke Studio's Timeline Environment

This chapter is designed to help you learn how to use Nuke Studio's Timeline environment, including ingesting media and using the Timeline Viewer.

Achieving Real-time Playback

The following is a list of recommended hardware configurations for Windows and Linux that Foundry have certified for real-time playback in the Nuke Studio and Hiero timeline Viewer (see the note below). Nuke's compositing Viewer is not designed for real-time playback. Please note that playback may also work on other machine configurations, but those listed below have been tested.

*Note:* For real-time playback, clips and sequences should be 16-bit RGBA DPX files, localized to fast storage. See *Localizing Media* for more information.

In the case of complex or multi-format timelines, files must be cached to the fast storage using the Timeline Disk Caching. See *Caching Frames in the Disk Cache* for more information.

### Windows and Linux Certified Hardware

Nuke Studio is currently tested on the HP Z workstation series. The following are the tested components for an HP Z840.

<table>
<thead>
<tr>
<th>Component</th>
<th>Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Processor</td>
<td>2 x Intel Xeon E5-2687Wv3 CPU</td>
</tr>
<tr>
<td>RAM</td>
<td>128 GB DDR4-2133 (8x16 GB) 2 CPU Registered RAM</td>
</tr>
<tr>
<td>Internal system drives</td>
<td>1 x HP Z Turbo Drive G2 256 GB PCIe</td>
</tr>
<tr>
<td>GPU</td>
<td>NVIDIA Quadro M6000 12 GB</td>
</tr>
<tr>
<td>Video monitor output</td>
<td>AJA Kona 4K or BMD DeckLink 4K Extreme 12G</td>
</tr>
</tbody>
</table>

Real-time playback is currently tested with the following internal storage accessories:
Drives | 8 x Samsung SM863 960 GB
---|---
Dock | 2 x HP B8K60AA - 4 Bay SSD carrier
Raid card | LSI Megaraid ROC 9270-8i
Raid cable | 2 x Avago 05-26113-00 0.6 Meter internal cable SFF8643 To X4 SATA HDD (MINI SAS HD)
Raid type | RAID 5, Always write-through, with a 256k Stripe size

The table below shows guidelines for the data rates required for real-time playback of single track, 16-bit RGBA DPX sequences, with no soft effects in Nuke Studio's timeline Viewer. The Data rate is the required disk speed needed to achieve real-time streaming playback, for a given format, at a given playback rate.

**Note**: The sample rates are taken from the AJA System Test application, with the Settings > Disk Test set to File per frame, and should be used as guidelines only.

The AJA System Test application is available for download for Mac and Windows from [https://www.aja.com/products/aja-system-test](https://www.aja.com/products/aja-system-test)

<table>
<thead>
<tr>
<th>Frames per second (fps)</th>
<th>Format</th>
<th>Data rate MB/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>25 fps</td>
<td>4K Film (4096x3112)</td>
<td>2431.25</td>
</tr>
<tr>
<td></td>
<td>4K UHD (3840x2160)</td>
<td>1582.03</td>
</tr>
<tr>
<td></td>
<td>2K Film (2048x1556)</td>
<td>607.81</td>
</tr>
<tr>
<td></td>
<td>HD 1080p (1920x1080)</td>
<td>395.51</td>
</tr>
<tr>
<td>30 fps</td>
<td>4K Film (4096x3112)</td>
<td>2917.50</td>
</tr>
<tr>
<td></td>
<td>4K UHD (3840x2160)</td>
<td>1898.44</td>
</tr>
<tr>
<td></td>
<td>2K Film (2048x1556)</td>
<td>729.38</td>
</tr>
<tr>
<td></td>
<td>HD 1080p (1920x1080)</td>
<td>474.61</td>
</tr>
<tr>
<td>Frames per second (fps)</td>
<td>Format</td>
<td>Data rate MB/s</td>
</tr>
<tr>
<td>------------------------</td>
<td>-------------------------</td>
<td>----------------</td>
</tr>
<tr>
<td>60 fps</td>
<td>4K Film (4096x3112)</td>
<td>5835.00</td>
</tr>
<tr>
<td></td>
<td>4K UHD (3840x2160)</td>
<td>3796.88</td>
</tr>
<tr>
<td></td>
<td>2K Film (2048x1556)</td>
<td>1458.75</td>
</tr>
<tr>
<td></td>
<td>HD 1080p (1920x1080)</td>
<td>949.22</td>
</tr>
</tbody>
</table>
About Clips and Shots

The interface sorts your clips into three broad categories: Audio and Video, Audio Only, and Video Only. Clips are displayed differently depending on their content, location, and in the case of the Viewer, the current mode.

**Note:** The timeline Viewer currently treats all alpha channels as premultiplied, which can result in the Viewer background being "added" to the image. If you're working with un-premultiplied images, set the Viewer background to **Black**.

**Note:** There is currently no QuickTime audio support on Linux. Support for audio is scheduled for a later release.

Source Clips

Source clips are representations of files on disk. Shots on the timeline refer to source clips, so making changes to a clip in a bin affects all shots that refer to that source clip.

**Note:** The colored bars under the thumbnail represent the layers available in the clip, in this case **color**. Other layers include **alpha**, **depth**, and **motion**, similar to Nuke.
Shots

Shots are representations of source clips, they are not saved on disk. Making changes to a shot only affects that instance. See Managing Timelines for more information.

A Source Clip Opened in the Timeline View

Opening a source clip in the timeline view allows you to control its output and add soft effects to the source clip. Clips opened as timelines always contain all available frames, but you can adjust their output using the original range controls in the clip's properties or in and out points. See Using In and Out Markers and Soft Effects for more information.
Clip and Shot Properties

The Properties contain standard controls similar to Nuke's Read node Properties and format-specific controls, depending on the selected source clip or shot. You can also control localization for individual clips and shots using the localization policy dropdown.

See Localizing Media for more information on localization.

If the clip Properties are not displayed in your workspace, double-click a source clip or shot or navigate to Window > Properties to open them in a floating window.

**Note:** The Properties panel allows you to override the Preferences and Project Settings on a per file basis. See Appendix A: Preferences and Timeline Environment Project Settings for more information.

As an example, .mov files allow you to control the decoder and ycbcr Matrix for clips, as well as the standard controls available for all clips. R3D and ARRIRAW media use their own software development kits (SDKs) to control the extensive settings usually seen on RED and ARRIRAW Cameras.

**Note:** The RED Decode Resolution and ARRIRAW Resolution and Proxy dropdowns control the maximum allowed Viewer resolution, overriding the Viewer Image Quality setting.
In addition, source clip properties are accessible through the same Python API as Nuke, improving scripting capabilities and integration into existing pipelines.

You can also add OCIO context Viewer processes to the Viewer color transform dropdown in the same way as regular Nuke Read and OCIODisplay nodes. See Changing the Viewer Color Space for more information.

Setting Source Clip Ranges

The original range controls in the source clip/shot Properties panel allows you to set the output range for source clips when you add them to a sequence as shots. The entire frame range of the clip is read on import, but altering the original range changes the output of the shot on the timeline.

**Note:** You can reset source clip to its original frame range at any time by clicking rescan. Rescanning also appends new frames as they become available.

Changing the original range affects source clips and shots differently. Opening a source clip in the Viewer, or in the right-click Timeline View, and adjusting the original range limits or extends the clip to that range of frames.

Extending the range past the original range adds handles, shown in red.
Adjusting the **original range** changes the output of any existing shots, but still allows you to trim and slip the shot range using the available handles. New shots pick up the new **original range** specified in the **Properties**.

**Using Relative Paths to Media**

Nuke Studio's timeline Read nodes support relative paths in the **file** control, allowing you to import media relative to a specific project directory. The **Project Settings** control this behavior for the current project.
Tip: If you want to set the project path for all new projects, navigate to the Preferences under Project Defaults > General and set the project directory there.

To set a Read file path to a relative location:
1. Open the Project Settings by clicking Project > Edit Settings.
2. Enter the required file path in the Project Directory field.

Tip: Click Hrox Directory to automatically enter an expression that evaluates to the .hrox location.

3. Double-click the required source clip or shot to display its Properties.
4. In the file field, adjust the path to the relative location of the .hrox project file. For example, if the project directory and absolute path to the source clip are:

   C:/projects/timeline_read/scripts/
   C:/projects/timeline_read/footage/renders/shot110/A009C005_101029_L1O3.%04d.dpx

Then the relative path to the clip is:

   ../../../footage/renders/shot110/A009C005_101029_L1O3.%04d.dpx
Ingesting Media

Adding media is as simple as drag-and-drop from a file browser or selecting File > Import File(s) or Import Folder(s). The application imports your media into the bin view providing you with a thumbnail of all of your clips and preserving the original folder and file hierarchy.

The media is soft imported, creating symbolic links to locations on disk. See Using the Copy Exporter for information on how to quickly consolidate your media and projects, or Localizing Media to help stabilize playback.

**Note:** Projects containing large amounts of movie files (for example .r3d and .mov) may exceed the number of available file handles per process, causing problems opening new files or projects and exporting.

You can increase the default limit of 1024 by entering the following command from the terminal, then running the application from the same session:

```
ulimit -Sn 2048
```

Clips with no inherent frame rate information are assigned a frame rate at ingest as specified in the Preferences.

1. Open the Preferences dialog by pressing Shift+S.
2. Select Behaviors > Timecode from the sub-menu on the left.

3. Use the RED file timecode dropdown to determine R3D clip behavior:
   - **Default from File** - use the default set by the R3D file in question.
   - **Absolute Timecode** - force the use of the Absolute Timecode as specified in the clip metadata.
4. Use the **other media timecode** dropdown to determine clip behavior for all other clips:
   - **File Header** - the file metadata header is used to derive the timecode, if it exists. This option defaults to **Frame Number** if the header is missing.
   - **Frame Number** - ignores the metadata header, even when present, and derives the timecode from the frames in the clip.
5. Set the **max valid timebase** allowed from the image header, above which the value is clamped. Image files are often created with application specific timebase values in the header description. This can lead to reading in spuriously high frame rates, and the clamp aims to prevent this from happening.
   If your clips do have extremely high frame rates, increase this value as necessary to avoid clamping.
6. Enable or disable **EDL style spreadsheet timecodes**:
   - When disabled, the **srcOut** and **dstOut** values use the film convention, representing the **last** frame of the cut.
   - When enabled, the **srcOut** and **dstOut** values use the video convention, representing the frame directly **after** the cut.
7. Click **OK** to save your settings.
Using Drag-and-Drop

Locate your media in a file browser and drag the frame range, clip, folder, or folders into the Project tab.

Ingest behavior depends on the target:

• Dragging a folder into the Project tab automatically ingests all the contents of the folder, including other folders and their contents.

• Dragging a movie file, such as a .mov or .r3d, automatically ingests the entire clip.

• Dragging a single file or file range, that is part of an image sequence, is controlled by the Preferences > Behaviors > File Handling > Scan for file sequence range checkbox:
  • Enabled - the default setting, dragging a single file or file range, that is part of an image sequence, creates a clip in the bin view containing all available frames.
    For example, dragging frames 1-5 and 11-20 ingests the entire frame range.
  • Disabled - only the dragged frame or range is imported into the bin.
    For example, dragging frames 1-5 and 11-20 ingests two distinct clips, one containing 5 frames and one containing 10 frames.
Using the File Browser

If you prefer to work with menus, you can also import clips using the file browser. You can import individual clip files, ranges, or entire folders, depending on the amount of media you intend to use.

Whenever you load or save files, a browser similar to the one shown below is displayed. The directory navigation buttons let you create or access the directory from which you wish to read or write data.

The navigation controls let you move through the directory structure, bookmark favorite directories, and create new directory folders.

**Note:** If you import folders, use the Import Options dialog to filter your ingest using inclusion and exclusion parameters, separated by spaces. The dialog’s Include patterns field defaults to {supportedfiles}, which resolves to a list of all known supported file extensions. To add your own custom extensions to this, you can use {supportedfiles} *.ext (replacing .ext with your custom file extension).

**Windows only:** You can show/hide the drives that Windows auto creates by right-clicking the required drive, selecting ShowDefaults, and checking or unchecking the drive.
To Use the Navigation Controls

- Click the **Create New Directory** button to create a new directory at your current position in the file hierarchy.
- Click **Up one directory** to go up one directory closer to the root.
- Click **Previous directory** to go back one directory.
- Click **Next directory** to go forward one directory.
- Click the + button to add a directory bookmark.
- Click the edit button to edit the name or path name to a bookmark.
- Click the - button to remove a directory bookmark.

Path Name Field

The path name field displays the current directory path, lets you navigate to a new path, and also enter a file name for scripts and rendered images.

- To navigate to a directory, type the path name in the field.
- To enter a script name, browse to a directory path and enter the file name after the displayed path.
- To limit the file list to specific file types, use the filter dropdown menu and **Sequences** checkbox.
To Use the Filter Dropdown Menu and Sequences Checkbox

- Select *.nk to display only Nuke script files.
- Select * to display all files (except hidden files), regardless of what they’re associated with.
- Select .* to display all files, including hidden files.
- Select */ to display directory names, but not their contents.
- Check sequences to display image sequences as single titles, as in fgelement.###.cin 1-50 rather than fgelement.0001.cin, fgelement.0002.cin, fgelement.0003.cin, and so on.

**Note:** File sequences with no file extension (for example, fgelement.0001, fgelement.0002, fgelement.0003, and so on) are not displayed as single titles the first time you view the directory in the File Browser. However, they are displayed as single titles once you have navigated to another directory and back again.

**Note:** By default, the application may not be able to display custom file extensions (for example, .cext) as single titles. To fix this, you can register your custom file extension as a sequence type using Python:

1. Create a file called init.py in your plug-in path directory if one doesn’t already exist. For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.
2. Open the init.py file in a text editor and add an entry in the following format (replacing cext with your custom file extension):

   ```python
   nuke.addSequenceFileExtension("cext")
   ```

- You can also split incomplete sequences into separate Read nodes using the split seq checkbox.

To Preview Files in the File Browser

1. Click the black arrow in the top-right corner of the file browser.
The file browser expands to include a small viewer.

2. Select the file you want to preview in the file browser to view it.
Using the File Browser | To Select Multiple Files with the File Browser

To Select Multiple Files with the File Browser

1. Browse to the folder where the files are located.
2. **Ctrl**+click on all the files you want to open to select them (Mac users **Cmd**+click).
3. You can open files from multiple directories by clicking **Next** and browsing to the next file location.
4. Click **Open**.
All the selected files open.
Sorting and Searching Media

Nuke Studio's Project panel includes several sorting and searching methods so you can organize, manage, and navigate through media more easily. You can sort items alphabetically, by type, or manually and select the appearance of media within bins. The search functionality allows you to enter strings and apply searches on all or partial matches. You can control the poster frame displayed by items in the project, selecting either an absolute or relative frame.

You can also color-code items in the Project panel and timeline to quickly sort and locate media. See Color-coding Source Clips and Shots for more information.

Sorting the Project Panel

The Project panel can be sorted in the left-hand directory side and right-hand file side independently, though files have more sorting and display options.

In the left-hand directory pane, items are sorted alphabetically in ascending order by default. To sort project items manually:
1. Click and hold, or right-click, the sorting dropdown and select Manual.
2. Drag-and-drop items into your preferred order.
The right-hand bin pane is also sorted alphabetically in ascending order by default, and there are more options available in the sorting dropdown. Click and hold the dropdown to display the various options, including **Duration**, **Resolution**, and **Type**.

You can also change the layout and size of items in the right-hand bin pane. Choose a layout from thumbnails, list, and details and use the slider to determine the size of the items. The slider is disabled in details mode.
Searching for Media

Nuke Studio’s search functionality allows you to enter strings and apply searches on all or partial matches with the option to include metadata searches. Nuke Studio searches for items that match any of the input string and displays only those items by default.

**Tip:** Nuke Studio also examines file metadata for the search string by default, but you can disable this behavior by disabling **Include Metadata** in the search dropdown.

- **Match All Criteria** - media that matches all the entries in the search string are matched. For example, entering A01 DPX only matches media containing both A01 and DPX.
• **Match Any Criteria** - media that matches any of the entries in the search string are matched. Using the same example, A01 DPX matches any media containing A01 or DPX.

![Tip:](image) This functionality extends to the media spreadsheet if you prefer to search for items there. See About the Media Spreadsheet for more information.

Filtering and flagging produce the same search results, but they are presented differently. Filtering only displays files that match some or all of the search string, whereas flagging displays all content and flags files that don’t match some or all of the search string.

You can also filter and flag media by tag or metadata. See Filtering and Flagging Media Using Tags and Filtering and Flagging Media Using Metadata for more information.

### Setting Poster Frames

You can assign poster frames to image items in the Project panel so that they represent the frames you intend to use if your sequences. This is particularly useful if your source clips contain handles of blank frames at the beginning of the footage.

To set a poster frame for source clips:

1. Select the clip or clips in the Project panel.
2. Right-click the selection, and select **Set Poster Frame**.
3. Select a preset frame or click **Custom** to select an absolute or relative frame as the poster frame for the source clip:

- **Absolute** - the poster frame number is derived from the file(s) on disk. For example, a `.dpk` sequence **myClip.###.dpk 1001-1500** can have an absolute poster frame between **1001** and **1500**.

- **Relative** - the poster frame number is derived from the number of frames in the clip starting at **0**. For example, a `.dpk` sequence **myClip.###.dpk 1001-1500** can have a relative poster frame between **0** and **499**.

**Note:** If you select more than one source clip, you can't set an **Absolute** frame as the poster frame.
Color-coding Source Clips and Shots

In large projects, the Project bin and timeline can quickly become busy and difficult to manage. Adding colors to items or file types can help you find what you're looking for in bins and sequences. You can quickly enable and disable colors in the Project panel and timeline in the Preferences under Panels > Project Items. See the Appendix A: Preferences page under Panels for more information.

You can set general defaults for items such as bins, sequences, and source clips as well as for specific file types such as .exr or .mov files. You can also set custom colors for individual selections or groups using the right-click menu or color picker button at the top left of the Project panel.

Setting Default Colors

Project items have a default color assigned to them by type, such as orange for sequences and gray for source clips. You can change these assignments using the Preferences > Panels > Project Items dialog.

Note: You can disable the color scheme in the Project panel, timeline, and spreadsheet at any time using the display in controls in the Preferences.
To change a default color, click a button in the **Project Items** preferences and use the color wheel to change the color.

You can also set the label color for the **Project** panel and timeline or change the color used to indicate the state of a shot on the timeline, such as **offline**.

**Setting Colors by Selection**

You can keep track of particular clips and shots by manually assigning them a color of their own. This custom color overrides any color you set by type. You can set colors on items individually or by selecting multiple instances in the **Project** panel or timeline.
1. In the **Project** panel or timeline, select the items you want to color.

   **Tip:** The selection tools can help you make multiple selections quickly in the timeline. See [Using the Selection Tools](#) for more information.

2. Right-click the selection and choose **Color > Color Picker** or click a color under **Recent**.

   ![Color Picker](image)

   **Tip:** You can also color selections by clicking the button and picking the color.

3. Click **Clear Color** to remove any custom colors applied to the selection. The selection reverts to the default item color set in the **Preferences**.

### Setting Colors by File Type

Another way to organize your project is to assign colors to items by file extension. This custom color overrides any color you set by item type.

1. Open the **Preferences** by pressing **Shift+S**.
2. Navigate to the **Panels > Project Items** sub-menu.
3. Under file types, click the + button.
   A new entry is added to the file types table.
4. Click the **extension** dropdown and select the required file type.
5. Double-click the **color** swatch and select the new color using the color wheel.

6. Click **OK** to apply your changes.
Reconnecting and Refreshing Clips

During the post process, media inevitably changes location or form. You can reload or replace your media using the reconnect, refresh, and rescan functions.

Though all three options deal with reloading source clips, each has a particular use dependent on context:

- **Reconnect Media** - allows you to redirect the filepath when the source file location changes.
- **Refresh Clips (F8)** - allows you to reload the clip when the source file location has not changed, such as when work has been done on the clip offline. Selecting refresh only refreshes the clip’s current frame range.
- **Rescan Clip Range (Alt+F5)** - similar to Refresh Clips, above, but rescan also checks for additional frames that may have been added to the source file and adds them to the source clip’s frame range.
Localizing Media

Nuke Studio has the facility to cache source clips locally, either individually or by setting an automatically localized folder (NUKE_TEMP_DIR/localize, by default), to help guarantee playback stability. Local caching is controlled initially in the Preferences dialog, then on a clip-by-clip basis.

Setting Localization Preferences

The Preferences control how Nuke Studio deals with new source clips as they are ingested, but does not affect existing source clips in the project. If you plan to localize source clips in your project, it's a good idea to set the Preferences before ingesting your source clips.

1. Press Shift+S to open the Preferences dialog and navigate to Performance > Localization.

2. Set the required localization mode:
   - **on** - checks for updates to source clips for all localization policies, and localizes those files set to On or From auto-localize path automatically. **On demand** source clips must always be localized manually.
   - **manual** - checks for updates to source clips and prompts you to update them manually. Only those with the policy set to **off** are not updated.
3. Set the default localization policy for new source clips using the dropdown:

**Note:** The localization policy for existing source clips in the project must be set individually. See Managing Localization for more information.

- **on** - always localize source clips with this policy.
- **from auto-localize path** - localize these source clips automatically if they reside in the auto-localize from directory.
- **on demand** - only localize these source clips when you manually update them. See Updating On Demand Clips for more information.
- **off** - never localize these source clips.

4. Enter a file path for auto-localize from, if required.

Any files that reside in this directory are automatically cached when ingested into Nuke Studio, providing that the Localization Policy is set to From auto-localize path.

**Note:** Localization in both the timeline and Node Graph is paused during compositing Viewer playback so that performance is not affected.

5. Enter a file path for localize to. Leaving this field as the default creates a sub-directory in the Temp Directory as the local cache.

**Note:** On Windows, files saved to the localize to directory replace \ (double back slashes) and : (colon drive signifiers) with underscores so that the file path works as expected between operating systems. For example:

- `\windowspath\to\my\network\file.dpx` is saved as `_windowspath\to\my\network\file.dpx`
- `t:\my\network\path\file.dpx` is saved as `t\_my\network\path\file.dpx`

6. Enter a value for limit to (GB) to control how much disk space is available in the cache directory.
7. You can specify the time interval (in minutes) before localized files are checked for updates using the *check for updated files every ## mins* control.

The default setting checks for new versions of files every 30 minutes, but you can set this control to any value. Any files that have changed since the last update check are flagged red in the **Project** bin.

**Tip:** You can change the default localization indicators using the **Appearance** controls.

If *read source files when localized files are out of date* is enabled, source clips referencing cached files that have changed since they were localized revert to reading the source files. Source clips that are reading from the source files are marked with a striped red bar on the clip.
Enabling **hide out of date progress bar** hides the localization state of out of date files so that the source clip appears the same as regular clips.

**Note:** The out of date localized files are not discarded, disabling **read source files when localized files are out of date** picks up the out of date files instead of reading the source files.

### Managing Localization

As well as the overall **Preferences** for when new source clips should be localized, you can set localization on a file-by-file basis. The **localization policy** for any existing source clips in your project must be managed individually.

**Tip:** If you find that localization is slow to copy files, you can increase the number of threads that Nuke uses to process jobs. Set the NUKE_LOCALIZATION_NUMWATCHERS environment variable to the number of threads you want to use. See **Environment Variables** for more information.

To set the localization policy for source clips:

1. Select the clip(s) in the bin view.
2. Right-click and select **Localization Policy** to display the available options:
   - **On** - the files are localized, regardless of location, as long as the **limit to (GB)** limit is not breached.
• **From auto-localize path** - the files are localized if they reside in the **auto-localize from** directory, as long as the **limit to (GB)** limit is not breached.

• **On Demand** - the files are only localized when you update them manually. See **Updating On Demand Clips** for more information.

• **Off** - the files are never localized, regardless of location.

As clips are localized, an amber progress bar displays in the thumbnail. Fully cached clips are marked with a green bar at the top of the thumbnail and out-of-date clips are marked in red.

**Note:** Container formats, such as `.mov` and `.r3d`, do not display progress bars during localization. The source clip only shows the green indicator once the entire file is localized successfully.

![Image of Nuke Studio workspace showing localized clips]

**Note:** Nuke Studio also features a playback cache and timeline disk cache, allowing frames to be cached in RAM or disk. See **Caching Frames in the Playback Cache** and **Caching Frames in the Disk Cache** for more information.

3. If you need to pause localization temporarily, navigate to **Cache > Localization** and select **Pause**.
4. If you find that your cache is filling up regularly, you can:
   • Increase the amount of available space for localization by raising the limit to (GB) preference,
   • Navigate to Cache > Localization > Clear Unused Local Files (see Clearing Localized Files), or
   • Manually clear files from the cache directory in NUKE_TEMP_DIR/localize, by default.

5. You can force Nuke Studio to check for updated source files, rather than waiting for the check for updated files every interval to expire, by selecting Force Update for All files, just Selected source clips, or On Demand only.

   ![Screenshot of Nuke Studio interface]

   **Note:** Each file has its own update time, which is reset whenever the source files are checked.

   The following table is a quick reference guide to when and how source clips are localized, where:
   • **green** - clips are localized automatically.
   • **amber** - clips are localized when updated manually.
   • **red** - clips are not localized.

<table>
<thead>
<tr>
<th>System Preference</th>
<th>Source Clip Preference</th>
</tr>
</thead>
<tbody>
<tr>
<td>on</td>
<td>auto-path</td>
</tr>
<tr>
<td>on</td>
<td>Green</td>
</tr>
</tbody>
</table>
### Updating On Demand Clips

Source clips with their localization policy set to on demand are polled to check for updates at the interval set in the Preferences. If the file has changed, the source clip is marked with a red bar to show it is out of date.

To update a single on demand source clip:

1. Double-click the source clip in the Project bin to display its Properties.
2. Click Update.

![Source Clip Properties](image)

**Note:** If a Read node's localization policy is set to on demand and it hasn't been localized previously, clicking the Update button localizes the file.

The local copy of the source clip is updated from the remote source clip.

To update all on demand source clips:

1. Navigate to Localization > Force Update.
2. Click On demand only.
The local copy of all on demand source clips is updated from the remote source clip.

Clearing Localized Files

Localizing a large amount of files can fill up the localization cache quite quickly if you leave the limit to (GB) preference at the default 10 GB. When the cache runs out of space, a Failed to Localize File dialog displays and localization pauses.

You can delete localized files by clicking Delete Unused Local Files (or by navigating to Cache > Localization > Clear Unused Local Files). Nuke Studio displays a dialog containing all the files that are marked for delete.

Tip: You can also open the Preferences from this dialog to adjust the localization behavior, such as increasing the limit to (GB) preference.
Click **Continue** to delete the localized files or **Cancel** to keep the cached files.
Using the Timeline Viewer

Nuke Studio’s Timeline environment supports two distinct Viewer types: clip and sequence. This chapter describes the difference between the two and how to use them.

Clip Viewers, sometimes referred to as source Viewers, are marked with the ![icon] and deal exclusively with source clips. You can set In and Out points and apply tags to the Viewer, but the source clips are unaffected.

Sequence Viewers, also known as record Viewers, are marked with the ![icon] and deal with sequences and shots on the timeline. You can set In and Out points and apply tags here too, but you can also edit the shots on the timeline by trimming, retiming, and so on. See Timeline Editing Tools for more information.

The Editing workspace combines both clip and sequence Viewers by default, enabling you to add source clips to the timeline using insert and overwrite functions. See Insert, Overwrite, and 3-Point Editing for more information.

To view your media in a Viewer, simply drag-and-drop a clip or sequence from the Project tab on to a Viewer input, or double-click the item to send it to the appropriate Viewer.

Note: The Viewer currently treats all alpha channels as premultiplied, which can result in the Viewer background being “added” to the image. If you’re working with un-premultiplied images, set the Viewer background to Black.
See Appendix A: Preferences for more information.

Deleting Media

To remove media from the bin view, select the clip(s) or bin and press Backspace or Delete.

If any of the media is in use in a sequence, the following warning displays:

Click Yes to delete the media from the bin view, but bear in mind that all instances of the deleted media are removed from your current sequences.
Timeline Playback Tools

There are many useful tools at the top of the Viewer, some of which allow you to select channels, adjust gain and gamma, and zoom and scale down the image in the Viewer.

For more information about the tools above the Viewer, see Timeline Viewer Tools.

The tools at the bottom of the Viewer allow you to adjust the playback settings, including setting the frame range, selecting the playback mode, and locking the Viewer playback range.

Drag the orange marker along the timeline to quickly cue to a specific frame or timecode. The number of the current frame or timecode appears below the center of the timeline. You can also cue to a frame or timecode by typing its number directly into this field.

Tip: The current frame and in and out point fields accept simple mathematical functions, such as +/-20 to jump forward or backward 20 frames or +/-00002000 to jump forward or backward 20 seconds.

By default, Nuke Studio automatically adjusts the timeline of every Viewer window to show the frame range defined in your Project Settings. If no frame range is defined, the frame range of the first image you read in is used as the global frame range.
Viewer timeline controls also have a frame range source dropdown menu that you can use to define where the timeline gets its frame range from. You can set this menu to **Global**, **Input**, or **Custom**. **Global** is the default setting described above.

The **playback rate** field (frames-per-second) initially displays the project's playback speed. Nuke Studio attempts to maintain this speed throughout playback, although this adjusts depending on the resolution of the imagery and your hardware configuration.

**Note:** The asterisk (*) denotes the **Sequence** playback speed selected using the **Frame Rate** dropdown or, for new projects, the **Project Settings > Sequence > Frame Rate** dropdown.

## In and Out Points

In and Out markers enable you to alter the duration of a clip to just the portions of the source that you require.

When a clip containing In and Out points is added to a timeline, you can slip the clip around the markers to adjust the clip's output. See **Timeline Editing Tools** for more information.

You can also use In and Out points to export certain portions of a clip or sequence. See **Transcoding** for more information.

To set In and Out markers:

1. Right-click on the required clip or sequence and select **Open In > Timeline View**.
   - Clips opened in a timeline have a purple background in the timeline.
2. Move the playhead to the location of the In point and press I on your keyboard.
   - The In point is marked by the In tab and the time is recorded in the playback controls.
3. Move the playhead to the location of the Out point and press O on your keyboard.
   - The Out point is marked by the Out tab and the time is recorded in the playback controls.

**Note:** You can also set markers by navigating to **View > Mark In** or **Mark Out**, by using the **Timeline** menu to **Mark Selection** or **Mark Clip** dependent on clip selections on the timeline, or by right-clicking a shots and selecting **Open In > Viewer**.
Click and drag the markers to adjust their position, or hold Ctrl/Cmd to move both markers at once, retaining their relative positions. A Viewer preview shows the current frame for the selected marker(s) and a timecode/frame popup helps to set the new position.

Clear the markers from your clip by navigating to **Viewer > Clear In Point** (Alt+I) and **Clear Out Point** (Alt+O). The markers are removed completely, but you can reapply them by repositioning the playhead and pressing I or O.

**Tip:** You can also press Alt+U to remove both markers at once.

When the playhead is positioned near In and Out markers, the top half of the timecode scale controls the playhead and bottom half controls the markers.
Playback Controls

The **playback rate** field (frames-per-second) initially displays the project’s playback speed. The Viewer attempts to maintain this speed throughout playback, although this adjusts depending on the resolution of the imagery and your hardware configuration.

The following table lists the functions of the playback buttons:

<table>
<thead>
<tr>
<th>Buttons</th>
<th>Functions</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>The <strong>Play backward</strong> and <strong>Play forward</strong> buttons play the sequence backward or forward at the script’s frame rate. When you press a play buttons, it toggles to a stop a button.</td>
</tr>
<tr>
<td></td>
<td>The <strong>Back 1 Frame</strong> and <strong>Forward 1 Frame</strong> buttons cue the sequence to the previous or next frame.</td>
</tr>
<tr>
<td></td>
<td>The <strong>Previous keyframe</strong> and <strong>Next keyframe</strong> buttons cue the sequence to the script’s previous or next keyframe.</td>
</tr>
<tr>
<td></td>
<td>The <strong>First frame</strong> and <strong>Last frame</strong> buttons cue the sequence to the first and last frame.</td>
</tr>
<tr>
<td></td>
<td>The <strong>Frame Increment</strong> field allow you to specify the number of frames by which the Previous increment/Next increment buttons cue the sequence. This is set to 10 frames by default.</td>
</tr>
</tbody>
</table>

The J, K, and L keyboard shortcuts also control playback. The K keyboard shortcut is mapped to Pause/Play. J and L are mapped to backward and forward. Combinations are also supported:

- K+J - frame backward.
- K+L - frame forward.
- K+drag in the top third of the Viewer - standard jog controls. Dragging the cursor left and right moves the playhead backward and forward, frame-by-frame.
- The jog controls also detect rotary motion to jog through frames. Clockwise motion in the top third of the Viewer, while holding K, advances the playhead and anti-clockwise reverses the playhead.
- K+drag in the middle third of the Viewer - standard shuttle controls. Dragging the cursor left and right plays backward and forward, with increased frame rate toward the edges of the Viewer.
- K+drag in the bottom third of the Viewer - skips the playhead to absolute timeline position.
The **Playback Mode** button lets you control how many times and in what direction the Viewer plays back the sequence. Click the button to toggle between the following modes:

<table>
<thead>
<tr>
<th>Button</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>🔄 Repeat</td>
<td>Repeatedly plays the sequence in a loop.</td>
</tr>
<tr>
<td>🔄 Bounce</td>
<td>Repeatedly plays the image back and forth from head to tail.</td>
</tr>
<tr>
<td>⬅️ Stop</td>
<td>Plays once through the section between in and out point and stops at the out point. If these are not marked, then it plays from the beginning to the end of the sequence.</td>
</tr>
<tr>
<td>⬅️ Continue</td>
<td>Plays once from the beginning to the end of the sequence, ignoring any in and out points.</td>
</tr>
</tbody>
</table>
# Timeline Viewer Tools

The Viewer has two sets of tools for manipulating your media: the Viewer tools and the playback tools. The Viewer tools, located at the top of the Timeline Viewer, are used to affect the mouse pointer as you move over the Viewer, and to select Viewer preferences:

<table>
<thead>
<tr>
<th>Icon</th>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt=" RGBA Icon " /></td>
<td>Layers</td>
<td>Select the layer to output to the Viewer, for example <em>forward</em> motion vectors or <em>disparity</em>. Only layers available in the clip are displayed - check the clip's thumbnail to see at a glance which layers are present:</td>
</tr>
<tr>
<td><img src="image" alt=" Channels Icon " /></td>
<td>Channels</td>
<td>Select the channel(s) to output to the Viewer, for example <em>RGB</em>, single channel, <em>Alpha</em>, or <em>Luma</em>.</td>
</tr>
</tbody>
</table>

- red color layer.
- green color layer.
- blue color layer.
- alpha layer.
- depth layer.
- forward motion vector layers.
- backward motion vector layers.
- all other custom layers, such as disparity.

**Note:** You can scroll through available layers using *PgUp* or *PgDn*. 
<table>
<thead>
<tr>
<th>Icon</th>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
</table>
| ![sRGB](image.png) | Viewer color transform | Set the colorspace used to display images in the Viewer, for example sRGB and rec709.  
**Note:** If you have specified an OpenColorIO configuration file in the Preferences, you may have more colorspace choices available. |
| ![A/B](image.png) | Viewer Output | Click the A or B dropdown and select what you want to view. This can be selected tracks or tracks with selected tags.  
When both Viewer buffers contain an image, enable wipe to compare the two images. You can also use the center drop down to set the blend mode between images in the Viewer, for example Onion Skin or Difference, and the A/B buffer configuration. |
| ![Guides](image.png) | Guides | Enable or disable Viewer overlays:  
• title safe - any text intended for the audience should reside within this zone.  
• action safe - any visual elements intended for the audience should reside within this zone.  
• format center - adds a crosshair in the center of the format currently in the Viewer.  
• Format - adds a red, format-dependent box for the clip or sequence in the Viewer. Sequences support multi-format clips, see Viewing Multi-Format Timelines for more information. |
| ![Mask](image.png) | Mask | Enable or disable a range of Viewer masks, for example 16:9 or 1.85:1. |
| ![Clipping](image.png) | Clipping | Enable or disable Viewer warnings:  
• No Warnings - all clipping warnings are disabled.  
• Exposure - alerts you when the image is under (blue) or over (red) exposed. |
<table>
<thead>
<tr>
<th>Icon</th>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Annotations</td>
<td>Click to enable the Annotations tool bar. Annotations allow you draw and add text to clips in the Viewer. See Annotations for more information.</td>
</tr>
<tr>
<td>Note: The Annotations button also controls existing annotation visibility.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>ROI</td>
<td>Click and drag to define a Region of Interest (ROI) in the Viewer. The scopes only display information within the ROI, when active.</td>
</tr>
<tr>
<td></td>
<td>Pause playback caching</td>
<td>Pause or release Viewer playback caching, indicated by the green bar under the Viewer.</td>
</tr>
<tr>
<td>Scale</td>
<td></td>
<td>Set the scale applied to the clip in the Viewer, for example 25%, 75%, or Fit.</td>
</tr>
<tr>
<td>Image Quality</td>
<td></td>
<td>Set the Viewer image quality, for example 1:1, 1:4, or 1:16. The default setting, Auto, resizes the image dependent on the Viewer zoom level, which may re-cache the image at a higher resolution.</td>
</tr>
<tr>
<td>Note: Image quality, or proxy, for RED clips is dependent on the clip's Decode Resolution in the Media panel. For example, if you're viewing a 4K file and the Decode Resolution is set to Half Premium, a 1:1 proxy value is equal to 2K, 1:2 is equal to 1K, and so on.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Non RT Playback</td>
<td>Sets the Viewer playback mode:</td>
</tr>
<tr>
<td></td>
<td>Play All Frames - the default setting, plays all frames in real-time (dependent on hardware).</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Skip Frames - plays frames in real-time skipping where necessary to maintain the frame rate.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Play All Frames, Buffering - plays all frames by buffering</td>
<td></td>
</tr>
<tr>
<td>Icon</td>
<td>Function</td>
<td>Description</td>
</tr>
<tr>
<td>------</td>
<td>----------</td>
<td>-------------</td>
</tr>
<tr>
<td></td>
<td>See through missing media</td>
<td>and playing frames back as they become available. When disabled, any offline media on a timeline is treated as a blank clip so the Viewer cannot display the track underneath. This setting also applies to missing frames within a clip.</td>
</tr>
<tr>
<td>![v]</td>
<td>View</td>
<td>Select the Viewer display mode, for example <strong>Audio and Video</strong> or <strong>Video Only</strong>.</td>
</tr>
</tbody>
</table>
| ![f] | Obey Alpha | Allows you to control the alpha channel independent of the Viewer Blend Mode.  
  - **Enabled** - any alpha channel present in the image is treated as premultiplied transparency.  
  - **Disabled** - the alpha channel is ignored. |
| ![f/8] | Audio latency | Sets the audio latency, in milliseconds, for the current Viewer only. Audio latency allows you to correct audio and video synchronization by changing the point at which audio playback starts. Positive values cause the audio track to start earlier in relation to the video track, and vice versa. |
| ![y] | Gain | Adjusts the gain applied to the linear input image before viewing, but doesn't affect your exported image. |
| ![y] | Gamma | Adjusts the gamma applied to the image after the viewing transform, but doesn't affect your exported image. |
| ![sil] | Mute / Audio | Click to mute audio output for the current Viewer or use the slider to control the audio output level.  
  - **Tip:** You can also control volume on a per track and per shot basis. See **Audio and the Timeline** for more information. |
<table>
<thead>
<tr>
<th>Icon</th>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Color Sample</td>
<td>Enable or disable the RGBA color information bar in the Viewer.</td>
</tr>
</tbody>
</table>

**Note:** The **Color Sample** tool displays color information from the source file, not the colorspace selected in the Viewer color transform dropdown.

See [Working with Colors](#) for more information.

### Using In and Out Markers

In and Out markers enable you to alter the duration of a clip to just the portions of the source or sequence that you require.

**Tip:** You can use the source clip/shot properties **original range** controls in similar way to In and Out points. See [Setting Source Clip Ranges](#) for more information.

When a clip containing In and Out points is added to a timeline, you can slip the clip around the markers to adjust the clip’s output. See [Timeline Editing Tools](#) for more information. You can also use In and Out points to export certain portions of a clip or sequence. See [Transcoding](#) for more information.

To set In and Out markers:

1. Right-click on the required clip or sequence and select **Open In > Timeline View**.

**Tip:** Source clips opened in the timeline view have a purple background in the timeline.

2. Move the playhead to the location of the In point and press **I** on your keyboard.
   
   The In point is marked by the In tab and the time is recorded in the playback controls.

3. Move the playhead to the location of the Out point and press **O** on your keyboard.

   The Out point is marked by the Out tab and the time is recorded in the playback controls.
**Note:** You can also set markers by navigating to View > Mark In or Mark Out, by using the Timeline menu to Mark Selection or Mark Clip dependent on clip selections on the timeline, or by right-clicking a shots and selecting Open In > Viewer.

Click and drag the markers to adjust their position, or hold Ctrl/Cmd to move both markers at once, retaining their relative positions. A Viewer preview shows the current frame for the selected marker(s) and a timecode/frame popup helps to set the new position.

Clear the markers from your clip by navigating to View > Clear In Point (Alt+I) and Clear Out Point (Alt+O). The markers are removed completely, but you can reapply them by repositioning the playhead and pressing I or O.

**Tip:** You can also press Alt+U to remove both markers at once.

When the playhead is positioned near In and Out markers, the top half of the timecode scale controls the playhead and bottom half controls the markers.
Using In and Out Markers

In and Out markers enable you to alter the duration of a clip to just the portions of the source or sequence that you require.

**Tip:** You can use the source clip/shot properties **original range** controls in similar way to In and Out points. See [Setting Source Clip Ranges](#) for more information.

When a clip containing In and Out points is added to a timeline, you can slip the clip around the markers to adjust the clip’s output. See [Timeline Editing Tools](#) for more information. You can also use In and Out points to export certain portions of a clip or sequence. See [Transcoding](#) for more information.

To set In and Out markers:
1. Right-click on the required clip or sequence and select **Open In > Timeline View**.

   **Tip:** Source clips opened in the timeline view have a purple background in the timeline.

2. Move the playhead to the location of the In point and press I on your keyboard.
   The In point is marked by the In tab and the time is recorded in the playback controls.
3. Move the playhead to the location of the Out point and press O on your keyboard.
The Out point is marked by the Out tab and the time is recorded in the playback controls.

**Note:** You can also set markers by navigating to **View > Mark In** or **Mark Out**, by using the **Timeline** menu to **Mark Selection** or **Mark Clip** dependent on clip selections on the timeline, or by right-clicking a shots and selecting **Open In > Viewer**.

Click and drag the markers to adjust their position, or hold **Ctrl/Cmd** to move both markers at once, retaining their relative positions. A Viewer preview shows the current frame for the selected marker(s) and a timecode/frame popup helps to set the new position.

Clear the markers from your clip by navigating to **View > Clear In Point** (Alt+I) and **Clear Out Point** (Alt+O). The markers are removed completely, but you can reapply them by repositioning the playhead and pressing **I** or **O**.

**Tip:** You can also press **Alt+U** to remove both markers at once.

When the playhead is positioned near In and Out markers, the top half of the timecode scale controls the playhead and bottom half controls the markers.
Working with Colorspace changes are applicable to clips in bins and shots, as well as in the Viewer using the Media tab.

To apply colorspace changes to clips in bins:
1. Select the clip or clips in the bin view.
2. Right-click a selected clip and navigate to Set Media Color Transform.
3. Select the colorspace to apply to the clip selection.

Note: Only colorspaces applicable to the selection are displayed. For example, REDspace and LogC - CameraNative are only available for R3D and ARRIRAW clips, respectively.

4. Selecting multiple formats supporting different colorspaces, for example R3Ds and ARRIRAW, breaks the available LUTs into sub-menus:
To apply colorspace changes to shots:

1. Select the item(s) on the timeline.
2. Right-click a selected item and navigate to Set Media Color Transform.
3. Select the colorspace to apply to the selection.
Previewing on a Broadcast Monitor

The Monitor Out feature allows you to preview Viewer images and audio on an external broadcast video monitor to check the final result, including the correct colorspace and aspect ratio. This option requires additional hardware, such as a monitor output card or a FireWire port.

**Note:** Audio scrubbing is not currently available through monitor output cards. Audio scrubbing is only supported through internal audio output devices.

Our monitor out architecture interfaces directly with the AJA and BlackMagic device drivers, which are unified across their respective hardware lines, meaning all current supported cards for the versions detailed in Third-Party Libraries and Fonts should work. We’ve tested the following AJA and Blackmagic hardware:

<table>
<thead>
<tr>
<th>AJA Card:</th>
<th>KONA LHi</th>
<th>KONA 3G</th>
<th>KONA 4</th>
<th>KONA iOXT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Formats</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SD</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>HD</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>2K</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>UHD</td>
<td>●</td>
<td></td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>4K</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>BNC</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>HDMI</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
</tbody>
</table>

**Stereoscopic Support**

<table>
<thead>
<tr>
<th></th>
<th>No</th>
<th>Yes</th>
<th>Yes</th>
<th>No</th>
</tr>
</thead>
</table>

**Platforms**
AJA Card: | KONA LHi | KONA 3G | KONA 4 | KONA iOXT  
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Win, Mac, Linux</td>
<td>Win, Mac, Linux</td>
<td>Win, Linux</td>
<td>Mac 10.9 and 10.10</td>
</tr>
</tbody>
</table>

Drivers

- Windows - 12.5.1
- Mac - 12.5.1
- Linux - 12.5.2.7

- Windows - 12.5.1
- Mac - 12.5.1
- Linux - 12.5.2.7

- Windows - 12.5.1
- Mac - 12.5.1

Note: The following should be taken into account when using the Monitor Out functionality with AJA cards:
If you’re running AJA cards on Linux, you can contact www.aja.com/support to obtain the correct drivers.
12-bit monitor output is only supported with dual connection cards, that is cards with two physical connections, not dual links combining two separate streams of data.
Hiero is unable to send out the right eye separately using the 2nd output cable of KONA 3G cards. Instead, both views are sent through the 1st output and can be viewed using the side-by-side, anaglyph, and interlacing options.

Blackmagic Card: | DeckLink SDI | DeckLink HD Extreme 2 | DeckLink Extreme 3D+ | Intensity Pro 4K  
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Formats</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SD</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HD</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>UHD</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BNC</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HDMI</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Blackmagic Card:</td>
<td>DeckLink SDI</td>
<td>DeckLink HD Extreme 2</td>
<td>DeckLink Extreme 3D+</td>
</tr>
<tr>
<td>-----------------</td>
<td>--------------</td>
<td>-----------------------</td>
<td>----------------------</td>
</tr>
<tr>
<td>Stereoscopic Support</td>
<td>No</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>Platforms</td>
<td>Win, Mac, Linux</td>
<td>Win, Mac, Linux</td>
<td>Win, Mac, Linux</td>
</tr>
<tr>
<td>Drivers</td>
<td>Desktop Video 10.7 to 10.8.4</td>
<td>Desktop Video 10.7 to 10.8.4</td>
<td>Desktop Video 10.7 to 10.8.4</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Blackmagic Card:</th>
<th>DeckLink Studio 4K</th>
<th>DeckLink 4K Extreme</th>
<th>DeckLink 4K Extreme 12G</th>
</tr>
</thead>
<tbody>
<tr>
<td>Formats</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SD</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HD</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>UHD</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BNC</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HDMI</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stereoscopic Support</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>(Both views through one output, so the <strong>Full Resolution</strong> option is not available.)</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Blackmagic Card: | DeckLink Studio 4K | DeckLink 4K Extreme | DeckLink 4K Extreme 12G  
----------------|-----------------|-----------------|-----------------  
Platforms        | Win, Mac, Linux | Win, Mac, Linux | Win, Mac, Linux  
Drivers          | Desktop Video 10.7 to 10.8.4 | Desktop Video 10.7 to 10.8.4 | Desktop Video 10.7 to 10.8.4  

Some monitor out cards allow you to extend or mirror your desktop so that Hiero's user interface is visible on the monitor. Please refer to your card's documentation for more information.

To preview output on an external broadcast video monitor:
1. Navigate to **Window > Monitor Output**.
   The **Monitor Output** toolbar displays in a floating pane.

![Monitor Output toolbar](image)

**Note:** If you're working with multi-view footage, additional controls display to determine the stereo mode and view to output.

See [Stereoscopic and Multi-View Projects](#) for more information.

2. Select the external device you want to use from the output device dropdown. All available devices are automatically detected and listed in this menu, along with the following default options:
   - **None** - disables the monitor out feed.
   - **Floating Window** - opens a pseudo output monitor window, without the need for a monitor device and card. This is intended for full-screen use without displaying any of the interface.
3. Select the Viewer to feed to the output monitor using the source viewer dropdown. Selecting **Active Viewer** always displays the current Viewer.
4. Select the view mode using the A/B selection dropdown.
Previewsing on a Broadcast Monitor | To preview output on an external broadcast video monitor:

**Note:** For multi-view/stereo footage, selecting A/B mode in this dropdown forces the monitor to output the timeline Viewer settings and the view to output controls are disabled.

5. Click ![ ] to apply the active Viewer’s filtering, gamma, and gain to the monitor output.
6. Click ![ ] to flip the output vertically.
7. Click ![ ] to switch between full-range 0-255 (default) and 16-236 (ITU-R BT.610-4). This button can correct the image output for certain monitor out cards.
8. Select the colorspace to apply to the image. If you’ve specified an OCIO configuration file in the preferences, these custom LUTs are also applicable.

**Note:** If you plan to use the OCIO config file specified during exports, ensure that the Preferences > Project Defaults > Color Management > Export > use OCIO nodes when exporting to a Comp checkbox is enabled.
Using Scopes

Nuke provides scopes to help you evaluate your media. There are a number of global controls (Preferences > Panels > Scopes) that affect how the Scopes display information:

- **black point** - sets the black out of range warning level.
- **white point** - sets the white out of range warning level.
- **luma/chroma encoding** - sets the video standard to use when converting RGB to luma or chroma values in the scope displays, either REC601 or REC709.
- **Include viewer color transforms** - when enabled, scope data includes the applied Viewer color transforms (gain, gamma, and LUT). When disabled, scope data does not include the applied Viewer color transforms. This may slow down rendering, as it may require image calculation.
- **Force full frame** - When enabled, scopes display data for the full frame, regardless of what portion of that frame is displayed in the Viewer. When disabled, scopes only display data for the current area requested by the Viewer rather than the full frame.

To open a scope, navigate to **Window > New Scope** and select the required scope from the list.

**Histogram**

The **Histogram** provides three color channel and luma channel information that describes the distribution of red, green, blue, and luma pixels throughout the current frame.

The Histogram graphs the number of pixels at each brightness level, and from left to right, the areas of the Histogram represent shadow, mid tones, and highlights.
Tip: You can pan the view area by holding Alt, or the middle mouse button, and dragging in the panel.

There are also Viewer and Channel selection controls on the Histogram tab:

- **Viewer selection** - if you have multiple Viewers open, use the dropdown menu to associate Histogram output to the required clip.
  
The default value, Active Viewer, automatically displays details on the last Viewer you selected.

- **Channel selection** - select the channels to output. The default setting displays RGB, but you can also view channels separately.

- **Mode selection** - select the mode to output. The default setting displays ganged RGB, but you can also view the channels separately.

- **Current View** - describes the view currently displayed in the scope, whether it's the A or B buffer and the view. The view defaults to main, unless main has been replaced in multi-view scripts or projects.

Depending on which Viewer tools and views you have active, you can have up to four scopes displayed at once.

For example, with two stereo Read nodes, one in each input buffer, and wipe and Side by Side active, the scopes display something like this:

The scopes feature global customizable guides to help you view your clips. Navigate to Preferences > Panels > Scopes and enter values between 0 and 1 for the Black and White points. Note that this also sets the values for the Waveform display.

The guides at the edges of the Histogram turn red to warn you when the distribution is out of range:
Waveform

The **Waveform** scope provides information on clip luminance, or brightness, which you can use to decide whether the clip is over or under exposed. The white traces represent luminance values from 0 - 100% (black through the spectrum to white). The higher the waveform, the brighter the image in the Viewer.

**Tip:** You can pan the view area by holding Alt, or the middle mouse button, and dragging in the panel.

The upper white marker is used to measure when over exposure could be a problem. If your waveform has a lot of traces over the white marker, you should consider reducing the brightness of the clip. The opposite is true of the lower black marker.

There are also **Viewer** and **Mode** selection controls on the **Waveform** tab:

- **Viewer selection** - if you have multiple Viewers open, use the dropdown menu to associate Waveform output to the required clip.
The default value, **Active Viewer**, automatically displays details on the last Viewer you selected.

- **Channel selection** - select the channels to output. The default setting displays RGB, but you can also view channels separately.
- **Mode selection** - select the mode to output. The default setting displays ganged RGB, but you can also view the channels separately.
- **Current View** - describes the view currently displayed in the scope, whether it’s the A or B buffer and the view. The view defaults to **main**, unless **main** has been replaced in multi-view scripts or projects.

Depending on which Viewer tools and views you have active, you can have up to four scopes displayed at once.

For example, with two stereo Read nodes, one in each input buffer, and **wipe** and **Side by Side** active, the scopes display something like this:

![Waveform](image)

The scopes feature global customizable guides to help you view your clips. Navigate to **Preferences > Panels > Scopes** and enter values between 0 and 1 for the **Black** and **White** points. Note that this also sets the values for the Histogram display.

The guides at the top and bottom of the Waveform turn red to warn you when the distribution is out of range:

![Waveform](image)
Vector

The Vector scope displays color, saturation, and hue information for the current frame. Similar to color wheels, Vector scopes display information radially, from the center outward. The farther from the center the data spans, the more saturation is represented.

In the image on the left, you can see that the frame represented contains mostly yellows and reds, but the values are not oversaturated. The image on the right represents a badly saturated frame. Notice the spill of red traces distributed toward the edge of the scope pass the target (the highlighted square).

Tip: You can pan the view area by holding Alt, or the middle mouse button, and dragging in the panel.

There is also a Viewer selection control and Current View label on the Vectorscope tab:

- **Viewer selection** - if you have multiple Viewers open, use the dropdown menu to associate Vector scope output to the required clip.

  The default value, **Active Viewer**, automatically displays details on the last Viewer you selected.

- **Current View** - describes the view currently displayed in the scope, whether it's the A or B buffer and the view. The view defaults to **main**, unless **main** has been replaced in multi-view scripts or projects.

  Depending on which Viewer tools and views you have active, you can have up to four scopes displayed at once.
For example, with two stereo Read nodes, one in each input buffer, and **wipe** and **Side by Side** active, the scopes display something like this:

![Vectorscope](image-url)
About Anamorphic Media

The Viewer automatically recognizes anamorphic clips and displays them with the correct aspect ratio.

If for any reason you want to display an anamorphic clip with a 1:1 aspect ratio, right-click in the Viewer displaying the clip and enable the **Ignore Pixel Aspect** checkbox, or use the **Ctrl/Cmd+Shift+P** keyboard shortcut.
About QuickTime Media

Working with .mov files can be unpredictable when compared to other formats, so Nuke gives you a few QuickTime options when reading and writing .mov files.

Nuke attempts to select the 'best fit' combination by reading an extended list of metadata key/value pairs from the QuickTime header, including nclc atom, gama atom, and ProRes codec headers.

If you place a clip in the Viewer, or open a shot in the Viewer, and open the Media tab, you'll see that Nuke has a number of media-specific controls that you can manually override if the 'best fit' is not what you're looking for:

- **YCbCr Matrix** - sets the way Y'CbCr is converted to RGB. You can choose to use the new Rec 601 and Rec 709 or the Legacy encoding methods, which are the methods used previously in Nuke.
- **Codec** - sets the codec used to read (write already had a similar control) the QuickTime file.

The codec dropdown defaults to a codec appropriate for the QuickTime in question, where available, and only lists those that declare themselves able to read the file.

- **Pixel Format** - sets the read and write pixel format, which includes bit depth, colorspace, pixel packing, and ranges.

This setting defaults to the best format accepted by the codec, allowing Nuke to perform the conversion to RGB without the use of an unknown QuickTime transform, where possible. RGB pixel types rely on QuickTime to do the conversion from YCbCr when dealing with a non-RGB codec.

In addition to the nclc, gama, and ProRes data Nuke, and by extension Nuke, also write additional metadata into the file headers during export, retaining your QuickTime preferences. This combined metadata represents myriad potential QuickTimes preferences, so Nuke reads the available metadata in the following order, reverting down each level as the level above is unavailable or set to a reserved or unknown value:
• Foundry-specific metadata
• ProRes header data
• nclc atom data
• gama atom data
• The defaults associated with the chosen codec

In this way, the worst case scenario is that you end up with the chosen codec class' default values.
About RED Media

When working with RED clips, using a RED Rocket card can increase the rendering speed significantly, especially at higher resolutions.

Note: The RED Rocket icon is only visible if you have a RED Rocket installed.

The RED Rocket icon has three states:

- **Inactive** - the RED Rocket card is inactive.
- **Firmware error** - there is a problem with the card firmware. Hover the mouse over the icon for more information.
- **Active** - the RED Rocket card is present and active.

To modify the RED Rocket options:

1. Click the 🚀 icon in the **Viewer**.

Note: You must have use RED Rocket enabled in the **Preferences > Performance > Hardware** dialog to access these options. See **Appendix A: Preferences** for more information.

The **RED Rocket Settings** dialog displays.

2. Temporarily disable the RED Rocket card by deseleting **Use RED Rocket card**. Unlike the option in the **Preferences** dialog, changing this setting does not affect the application at startup.
3. Click **OK** to save your settings.
Note: Projects containing large amounts of movie files (for example .r3d and .mov) may exceed the number of available file handles per process, causing problems opening new files or projects and exporting. You can increase the default limit of 1024 by entering the following command from the terminal, then running the application from the same session:

```
ulimit -Sn 2048
```
Compositing with Nuke

Each chapter in this section explains in detail a key feature of Nuke. You can use the section to familiarize yourself with the features you are particularly interested in, or to get answers to specific problems that arise during compositing. For information on the features in NukeX and Nuke Studio, see Advanced Compositing with NukeX and Nuke Studio or Timeline Editing in Nuke Studio.

Organization of the Section

These are the topics covered by this section:

- **Managing Scripts** describes the basics for creating, saving, and loading scripts or comps.
- **Reformatting Elements** describes how you can reformat images through scaling, cropping, and pixel aspect adjustments. This chapter also covers working with bounding boxes.
- **Channels** shows you how to manage image data using Nuke's 1023-channel workflow.
- **Merging Images** teaches you how to layer background and foreground elements together, create contact sheets, and copy rectangles from one image to another.
- **Removing Noise with Denoise** teaches you to use Denoise node to remove noise from your footage. It uses spatial filtering to remove noise without losing image quality.
- **Keying with ChromaKeyer** teaches you to use the blue/greenscreen keyer ChromaKeyer in Nuke.
- **Keying with Primatte** teaches you to use the blue/greenscreen keyer Primatte in Nuke.
- **Keying with Keylight** teaches you to use the keyer tool Keylight in Nuke.
- **Keying with Ultimatte** shows you how to use the Ultimatte keyer in Nuke.
- **Using RotoPaint** shows how to use Nuke's RotoPaint node.
- **Tracking and Stabilizing** shows how to generate and edit 2D tracking data for purposes of removing unwanted motion or applying it to other elements.
- **Transforming Elements** covers the tools for changing the size, location, and orientation of an image, including how to translate, scale, rotate, and skew elements in 2D and 3D space. This chapter also describes adding motion blur.
- **Warping Images** teaches you to use the GridWarp and SplineWarp nodes to warp and morph images.
- **Temporal Operations** explains how to apply time-based effects like clip retiming and motion blur. This chapter also explains how to perform editorial tasks, such as trimming and slipping.
- **Working with Color** explains a broad sampling of Nuke's many color correction tools.
- **Filtering and Spatial Effects** deals with applying filters, such as convolves and blurs.
- **Creating Effects** describes how you can create effects, such as star filter effects, on your images.
• **Analyzing and Matching Clips** explains how to use the CurveTool node to analyze and match image sequences.

• **3D Compositing** teaches you how to create and manipulate 3D scenes composed of objects, materials, lights, and cameras.

• **Stereoscopic Scripts** describes how to composite stereoscopic material in Nuke.

• **Deep Compositing** goes through using the deep compositing node set in Nuke.

• **Working with File Metadata** describes how to use Nuke's MetaData nodes to work with information embedded in images.

• **Audio in Nuke** covers using audio clips in Nuke.

• **Previews and Rendering** teaches you how to write out image sequences from scripts in order to preview results or create final elements.

• **Organizing Scripts** is designed to help you organize your Nuke scripts in a clear and meaningful way.

• **Configuring Nuke** explains how to set up Nuke for multiple artists working on the same project.

• **Expressions** explains how to apply expressions or scripting commands to Nuke parameters.

• **The Script Editor and Python** takes you through using Nuke's Script Editor for executing Python commands.
Managing Scripts

In this chapter, you learn about Nuke's project files called **scripts** or **comps**. The topics covered include setting up, saving, and loading scripts. You'll also learn about managing your node tree in the Node Graph, using Precomp nodes, and working with file metadata.

Setting Up Your Script

When you start working on a script, you should first define the settings for it. This involves assigning the script a name, frame range, frame rate, and default full and proxy resolution format.

**Name, Time Span, and Frame Rate**

To set the script name, frame range, and frame rate:

1. Select **Edit > Project Settings**, or simply press **S** over a blank portion of the workspace.

   The **Project Settings** panel appears.
2. On the **Root** tab, type a name for the script (say, `firstcomp.nk`) in the **name** field. Nuke’s scripts always have the extension `.nk`.

3. Type the numbers of the first and last frames in the **frame range** fields to define length of time for your “shot”.

4. In the **fps** field, enter the rate in frames per second (fps) at which you want your script’s Viewers to play back footage. For film-based elements, 24 fps is appropriate.

---

**Setting the Default Project Directory**

In the Project Settings (**Edit > Project Settings**) you can define your default project directory. You can then refer to it with `./` in your file paths, for example `./test.jpg` for an image called `test` in your
default project directory.

You can also click the Script Directory button to create an expression that sets your project directory to be the directory where your current script is saved.

Full-Size Formats

When you start a new script in Nuke, you need to set up a full-size format. The full-size format determines the size of the image that you get from any disconnected node inputs. It also sets the default size of any script-generated elements, such as Constants and ColorBars.

**Note:** The full-size format does not affect the format of the elements you read into your script. Nuke is resolution-independent, which means it respects and keeps the resolution of your images. It won't automatically crop or pad elements to match the project settings. If you want elements you read in to conform to the project settings, you can do this manually using the Reformat node. For more information, see [Reformatting Image Sequences](#).

The full-size format is also used to calculate proxy scaling if a proxy format is used. For more information on the proxy mode and proxy formats, see Proxy Mode.

To Set Up a Full-Size Format

1. If it's not already open, select **Edit > ProjectSettings** (or press **S**) to display the Project Settings panel.
2. From the **full size format** dropdown menu, select the resolution for the final output of rendered images. If the format you want to use is not in the menu, select **new**. The New format dialog displays.
In the **name** field, enter a name for the new format.
In the **file size** fields, define the width and height of the format.
If you like, you can also define additional information, such as offsets and pixel aspect ratio.
Click **OK** to save the format. It now appears in the dropdown menu where you can select it.

## Proxy Mode

When compositing with Nuke, you can work in two different modes: the **full-size mode** or **proxy mode**. In the full-size mode, images are read in exactly as they are on the disk, and all positions are in actual pixels in these images. This is the mode you want to use for accurate feedback and when rendering the final output.

In proxy mode, instead, a proxy scale factor is used. All images and all x/y positions are scaled by this factor. This produces the same (or at least very similar) composite at a different scale. For example, you can use a fraction of the full output resolution to speed up rendering and display calculation.

In addition to the above, a separate proxy file can also be read in place of a full-size image, provided you have specified one in the relevant Read node. This can further speed up the preview, by using a smaller image that reads faster and also saves time by not needing to be scaled. For more information, see Read Nodes and Proxy Files.

The proxy settings you define in the project settings affect both the proxies Nuke generates using the proxy scale factor and proxies read from files. Below, we discuss setting a proxy format and/or a proxy scale and defining how Read nodes use proxy files.

**Note:** Note that proxy versions of images are only used if you have activated the proxy mode. When the proxy mode is off, Nuke always uses the full-res files.

## Proxy Format and Proxy Scale

In the **Project Settings** panel, you have the option of defining a **proxy format** and/or a **proxy scale** that you use in the proxy mode.

For the proxy **format**, you can define the image resolution as well as additional information about offsets and pixel aspect ratio. When using the proxy format, the scaling is proportionate to the full-size/proxy format relationship (not scaled to the proxy format).
For the proxy **scale**, you only define a simple scale factor by which your images are scaled down whenever the proxy mode is activated. For example, you can use the scale factor of 0.5 to scale your images down to half the size.

If you like, you can define both a proxy format and a proxy scale, and then choose which one to use in proxy mode. A proxy scale is easier to set up, but a proxy format gives you more control over the low-res versions of your images. Below, we first describe how to set up proxy formats and then how to define a proxy scale.

### To Set Up Proxy Formats

1. If it's not already open, select **Edit > Project Settings** (or press **S**) to display the **Project Settings** panel.
2. If you want to use a proxy format (rather than a proxy scale) whenever the proxy mode is activated, select **proxy mode > format** from the dropdown.
3. From the **proxy format** dropdown menu, select the resolution to use while working to speed things up. Notice that your images are not scaled to this resolution, but the scaling is proportionate to the full-size.proxy format relationship. Nuke divides the proxy format width by the full-size width and uses the result as the scale factor.

If the proxy format you want to use is not in the dropdown menu, select **new**. The **New format** dialog displays.

![New format dialog](image)

- In the **name** field, enter a name for the new format.
- In the **file size** fields, define the width and height of the format.

**Tip:** You can type formulas in numeric fields to do quick calculations. For example, if your full-size format width is 4096 and you want your proxy format width to be 1/2 of that, you
can enter 4096/2 in the **New format** dialog’s **file size w** field and press **Enter**. Nuke then calculates the new width for you.

Click **OK** to save the format. It now appears in the dropdown menu where you can select it.

4. To activate the proxy mode and use the low-res format for calculations and display, check **proxy mode**.

Alternatively, you can use the proxy toggle in the Viewer controls, or press **Ctrl+P** (**Cmd+P** on a Mac). For more information, see Using the Viewer Controls.

### To Set Up a Proxy Scale

1. If it’s not already open, select **Edit > Project Settings** (or press **S**) to display the **Project Settings** panel.
2. Select **scale** from the dropdown in the **Project Settings**.
3. Using the **proxy scale** input field or slider, specify the factor by which you want to scale the width and height of your images. For example, if you want to scale them down by 50%, use the value of **0.5**.
4. To activate the proxy mode and use the low-res format for calculations and display, check **proxy mode**.

Alternatively, you can use the proxy toggle in the Viewer controls, or press **Ctrl+P** (**Cmd+P** on a Mac).

### Read Nodes and Proxy Files

As an alternative to letting Nuke generate proxies on the fly, proxy files can be specified using a second file name in the Read nodes (for how to do this, see Loading Image Sequences). If you don’t
have a proxy file, you can create one by activating the proxy mode and rendering your full-size images using a Write node (see Rendering Output in the Previews and Rendering chapter).

The proxy file does not need to match the proxy resolution in use. Depending on your project settings, either the full-res or proxy file is scaled to the required proxy size (that is, the size calculated by taking the full-size format and scaling it by the current proxy settings). However, if your proxy images match your typical proxy settings, you can save this time.

To Define Which File (Full-Res or Proxy) is Used in the Proxy Mode:
1. If it's not already open, select Edit > Project Settings (or press S) to display the Project Settings panel.
2. From read proxy files, select when to use the proxy file (rather than the full-res file) in a Read node:
   • never - Never use the proxy file in the proxy mode. Instead, scale the full-size file as necessary.
   • if larger - Use the smaller of the two images if it is larger or equal to the desired size, scaling down as needed. Otherwise, use the larger one, scaling down or up as needed. This is the default option.
   • if nearest - Use the image that is closest to the desired size, scaling up or down as needed.
   • always - Always use the proxy image in the proxy mode, scaling it up or down as necessary.

The option you choose affects all Read nodes in your script, provided that a proxy file is named and the proxy mode is on.

Write Nodes and Proxy Files

It is worth mentioning here that when a script is rendered in proxy mode, processing is done at the proxy scale and image output goes to the file name in the Write node's proxy field. If you do not specify a proxy file name, the render fails with an error. It never resizes the proxy image, and it does not write the proxy image over the full-size one.

For more information, see Rendering Output in the Previews and Rendering chapter.

Using Proxy Mode to Enlarge a Script

If you find it necessary to render a comp at higher than the original resolution it was intended for, you can also use the proxy resolution to scale up from the root full size format. For example, you could use a 2K composite to produce 4K or higher images in the proxy mode.

The reason you'd probably want to do this is because the comp you did needs to be re-run at higher resolution for a different output target and possibly with some new higher resolution elements. For
example, you may need to re-render with new CG elements at print resolution rather than for film out, but you don't want to go through your script and modify everything that has an x/y position.

When scaling up the output of a script in the proxy mode, image generator nodes render at the larger size, larger images can be specified in the proxy field, and as with scaling down, all x/y positions are scaled to the proxy output resolution.

Using Small Proxy Files

If you have previously set up your script to use small proxy files, you do not have to remove these. Make sure read proxy files in Project Settings is set to anything other than always, and Nuke reads the larger original files and scales them up.

Using Large Proxy Files

If you actually have larger proxy files, you should enter them into the Read nodes’ proxy field, and set read proxy files to anything other than never. Nuke then uses these larger files in the proxy mode. For maximum quality, these should be at exactly the desired proxy size so that no scaling is done to them. For example, if the required proxy size (defined by the project settings) is 4K, then the proxy image should be exactly 4K. Otherwise, Nuke scales it to match the project settings, which reduces the quality.

Rendering the Scaled-up Output

To render the larger output of a scaled-up script, you need to activate the proxy mode and enter a file name in the proxy field of the Write nodes. The larger images are then written to these files.

To Toggle Between Full Resolution and Proxy Mode

It's usually smart to work in proxy mode because most operations work quickly and more efficiently under the low-res display. You can switch between low- and high-resolution when you need greater precision (for example, when pulling a key or tracking), or when you're ready for final rendering.

1. Click on an empty area of the Nuke window.
2. Press Ctrl+P to toggle the display mode (Cmd+P on a Mac).

Nuke automatically scales script elements - Bezier shapes, B-spline shapes, paint curves, garbage masks, tracking curves, and so on - to keep the original placement on the image.
Loading Image Sequences

When you are ready to start compositing, you may want to begin by importing a background or foreground image sequence. Typically, you would read in both full- and proxy-resolution versions of the sequence. You can read in several image sequences in one go.

Importing Image Sequences

To import an image sequence using Nuke's file browser:

1. Select Image > Read (or press R over the Nuke Node Graph).

   Tip: Pressing R with an existing Read node selected, opens the file browser at the location specified by that node.

2. Browse to the image sequence you want to import. For instructions on using the file browser, see Using the File Browser. Select the file you want to open. If you want to open several files at the same time, Ctrl/Cmd+click on the files, and then click Open.

   A Read node is inserted in the Node Graph.

   ![Read node with file browser interface]

   Nuke imports the image sequence and displays it as a thumbnail on the Read node. Generally, the Read node does not reformat or resize the sequence in any way, and the node's properties panel is updated to display the native resolution and the frame range for the sequence. Note that the format and proxy format fields in the controls indicate the format of the images, they do not cause the images read from files to be resized to this format.

3. You can cycle through the available versions of a file using the Alt+Up/Down arrow keys. If you want to skip ahead to the highest version available, press Alt+Shift+Up arrow key.
Versions must written in the following format in order for Nuke recognize them:

../myFiles/grades/A003_C007_071105_v01_001.r3d
../myFiles/grades/A003_C007_071105_v02_001.r3d
../myFiles/grades/A003_C007_071105_v03_001.r3d

4. If your sequence has a red, green and blue channel but no alpha channel, check the auto alpha box in the Read node control panel to set the alpha channel to 1. This prevents possible problems from occurring if Nuke tries to read the alpha channel and one doesn't exist. The auto alpha box is unchecked by default.

**Note:** Nuke reads images from their native format, but the Read node outputs the result using a linear colorspace. If necessary, you can change the Colorspace option in the Read node's properties panel, or insert a Color > Colorspace node to select the color scheme you want to output or calculate.

**Note:** The maximum image size the Nuke Viewer can display is $2^{28} = 268,435,456$ pixels. This is the same as 16k x 16k, 32k x 8k, 64k x 4k, or 128k x 2k. If your image is larger than this, it is resized and you get the following warning:

“Viewer image is clipped to <size> x <size>!”

For example, if your image resolution is 60,000 x 4473, Nuke is able to display the image because the number of pixels is less than $2^{28}$. However, if the resolution is 60,000 x 4474 (more than $2^{28}$ pixels), the image is resized to 59998 x 4474. In addition to the Viewer, this limit is also applied to the bounding box of the images being passed between each node.

5. If you have a proxy version of the image sequence, click the proxy field's folder icon and navigate to the proxy version. Select Open. If you don't have a proxy version, don't worry: Nuke creates one on the fly according to the proxy scale or proxy format settings you specified in the project settings.

The proxy file does not need to match the proxy resolution in use. Depending on your settings, either the full-res or proxy file is scaled to the required proxy size. For more information, see Proxy Mode.

6. You can access the metadata contained within the read file by clicking the Metadata tab. Once you know which keys exist in the file, you can reference them in expressions. See Expressions for more information.

The metadata displayed depends on the file type. For example, a .jpg might only contain input/keys, whereas QuickTimes contain input/ and quicktime/ keys. See Working with File Metadata for more information.
To Import an Image Sequence from an External File Browser

To load an image, you can also drag and drop the image into the Node Graph from an external file browser (such as Windows Explorer or Mac Finder). To load an entire image sequence this way, drag and drop the directory that contains the images into the Node Graph.

Notes on Importing QuickTime Files

When reading QuickTime .mov files, Nuke attempts to select the "best fit" combination by reading an extended list of metadata key/value pairs from the QuickTime header, including nclc atom, gama atom, and ProRes codec headers. You can manually override the following mov Options in the Read node properties panel:

• **decoder** - sets the decode library used to read the file:
  • **mov32** - uses the full range of QuickTime codecs, but can be slow to process due to extra complexity during decode.
  • **mov64** - uses its own packing and unpacking and streams decode/encode for extra processing speed, but only supports a sub-set of QuickTime codecs.

**Note:** Nuke defaults to the fastest decoder for the codec used in the file - if you're reading in a type supported by the mov64 sub-set, Nuke defaults to that reader. Otherwise, the fallback mov32 reader is used.

• **ycbcr matrix** - This is only enabled when working with a YCbCr-based pixel type. Rec 601 and Rec 709 follow the ITU.BC specifications, whilst Nuke Legacy, Nuke Legacy Mpeg, and Nuke Legacy YUVS are retained for backwards compatibility.

**Note:** The default option, Format-based, selects Rec 601 or Rec 709 automatically based on the format size.

When you set the decoder to mov32, two additional controls are displayed:

• **codec** - sets the codec used to read the QuickTime file. The codec dropdown defaults to a codec appropriate for the QuickTime in question, where available, and only lists those that declare themselves able to read the file.
Note: If you're using the Avid DNxHD codec, Avid AVDn, avoid setting the pixel format control to r408 as there is a known issue within the codec causing frames to darken with each frame progression in the sequence.

• **Pixel format** - sets the read pixel format, which includes colorspace, bit depth, layout, and range. This setting defaults to the best format accepted by the codec, allowing Nuke to perform the conversion to RGB without the use of an unknown QuickTime transform where possible. RGB pixel types rely on QuickTime to do the conversion from Y'CbCr when dealing with a non-RGB codec.

Note: When reading QuickTime files, Nuke looks for metadata in the following order and uses it to govern the settings on the Read node, falling down to each level when the level above is unavailable or set to a reserved or unknown value:

1. Foundry-specific user data
2. prores header
3. nclc atom
4. gama atom
5. defaults based on codec type

QuickTime .mov files may appear different in Nuke relative to Apple's Final Cut Pro, because Final Cut Pro introduces a gamma compensation based on assumptions about the content of the files and the viewing environment.

To limit the number of background processes that Nuke can run when reading QuickTime files, go to Preferences and set the number for QuickTime decoders to use in the Performance > Threads/Processes tab.

Notes on Importing AVI Files

.avi files can be supported by default or only via Nuke's reader that is based on the FFmpeg open source library. If you get an error when using .avi files in Read nodes, you may need to use the prefix mov64: before the file path and file name, for example, mov64:\z:\job\FILM\IMG\final_comp_v01.####.avi.

Notes on Importing OpenEXR Files

Nuke supports multi-part OpenEXR 2.2 images, which allow you to store your channels, layers, and views in separate parts of the file in order to speed up processing. You can load multi-part OpenEXR files in exactly the same way as single-part OpenEXR files.
The OpenEXR file format allows the display window to have the lower left corner in any position. Unfortunately, Nuke needs all formats to have a lower left corner coordinate of 0,0. In the Read node control panel, under exrOptions, you can check the offset negative display window box to tell Nuke to offset the image so that the display window left side x coordinate is 0. If you uncheck the box, Nuke shrinks the format from both sides the amount that it’s negative in the x coordinate, in other words treat the area as overscan.

By default, the exr prefix is attached to metadata keys to make them distinct from other metadata in the tree. If you’d rather read metadata in “as is” without attaching a prefix, enable do not attach prefix.

When reading in .exr files, you can determine how pixels at the edge of the data window, or bounding box in Nuke terms, are treated using the edge pixels dropdown:

• plate detect - if the bounding box and format match exactly, then repeat all edges. Otherwise, add black at the edges.
• edge detect - for each matching edge, repeat the edge pixels. Add black at mismatched edges.
• repeat - always repeat the edge pixels outside the bounding box.
• black - always add black pixels outside the bounding box.

On Linux, the Nuke OpenEXR reader uses a memory-mapping function to improve performance reading PIZ-compressed .exr files. However, some customers have experienced hanging when reading large (frame size and number of channels) PIZ-compressed .exr files across an NFS network. If you experience this problem, you can tell Nuke not to use the mmap function by enabling this option. You can set it on a case-by-case basis or use a knobDefault in your init.py to always have it disabled. For more information on knobDefault, see the Nuke Python documentation (Help > Documentation).

Notes on Importing PSD Files

When loading a layered .psd file, a button called Breakout Layers appears under psd Options in the Read node control panel. Clicking this “breaks out” the .psd file into separate layers using some Shuffle nodes, and recombines these layers with a number of PSDMerge nodes. Each layer is grouped and labeled using a Backdrop node for clarity. In addition, a Crop node is inserted between the Shuffle and PSDMerge nodes, allowing you to adjust the bounding box for each layer individually.

The PSDMerge node is a type of merge node exclusive to this feature. In the control panel, there is an operation dropdown menu, a mask field (with an invert checkbox) and a mix slider. If the blend mode for a layer was set in Photoshop®, Nuke automatically sets this in the operation dropdown.
Note: The blend modes in PSDmerge are approximated and do not match Photoshop® exactly.

Naming Conventions

The file names of image sequences generally end in a number before the extension, for example image0001.rgb, image0002.rgb, image0003.rgb, and so on. When browsing for files like this, you may notice that the sequence appears as image####.rgb. Here, #### is Nuke’s way of indicating that the number is in a 4-digit incremental format. For a 3-digit format, such as image001.rgb, the frame number variable would be ###.

Nuke’s file browser also understands unpadded file names, such as image1.rgb, image2.rgb, image3.rgb, and so on. They appear as image#.rgb.

Changing the Relation Between the Current Frame and the Frame Read In

By default, Nuke assumes an exact relation between the current frame processed, and the frame read in. For example, at frame 15, Nuke reads in image.0015.rgb. However, you can change this behavior using the frame parameter on the Read node. For instance, if you have a sequence that runs from image.0500.rgb to image.1000.rgb, you may want to read in image.0500.rgb at frame 1. Nuke lets you do this using expressions, specified start frames, and constant offsets. Each method is described below.

Using Expressions

1. Select Image > Read to import an image sequence.
2. In the Read node controls, set the frame dropdown menu to expression. Enter an expression in the field on the right. The expression changes the relation between the current frame and the frame read in.
For example, if your clip begins from image.0500.rgb and you want to place this first frame at frame 1 rather than frame 500, you can use the expression \textbf{frame+499}. This way, 499 frames are added to the current frame to get the number of the frame that's read in. At frame 1, image.0500.rgb is read in; at frame 2, image.0501.rgb is read in; and so on.

Another example of an expression is \textbf{frame*2}. This expression multiplies the current frame by two to get the number of the frame that's read in. This way, only every other frame in the clip is used. At frame 1, image.0002.rgb is read in; at frame 2, image.0004.rgb is read in; at frame 3, image.0006.rgb is read in; and so on.

**Specifying a Start Frame for a Clip**

1. Select \textbf{Image} > \textbf{Read} to import an image sequence.
2. In the Read node controls, set the \textbf{frame} dropdown menu to \textbf{start at}. Enter a start frame number in the field on the right. This specifies the frame where the first frame in the sequence is read in. In other words, all frames are offset so that the clip starts at the specified frame.

   For example, if your sequence begins from image.0500.rgb and you enter \textbf{1} in the field, image0500.rgb is read in at frame 1. Similarly, if you enter \textbf{100} in the field, image0500.rgb is read in at frame 100.

**Offsetting All Frames by a Constant Value**

1. Select \textbf{Image} > \textbf{Read} to import an image sequence.
2. In the Read node controls, set the \textbf{frame} dropdown menu to \textbf{offset}. Enter a constant offset in the field on the right. This constant value is added to the current frame to get the number of the frame that's read in.

   For example, if your clip begins from image.0500.rgb and you want to place this first frame at frame 1 rather than frame 500, you can use \textbf{499} as the constant offset. This way, 499 is added to
the current frame to get the frame that’s read in. At frame 1, image.0500.rgb is read in; at frame 2, image.0501 is read in, and so on.

You can also use negative values as the constant offset. For example, if you use the value -10, Nuke subtracts ten from the current frame to get the frame that’s read in. At frame 20, image.0010.rgb is read in; at frame 21, image.0011.rgb is read in; and so on.
Reformatting Image Sequences

When you import image sequences, Nuke stores their format settings and makes them available to the Reformat node. You can then use the Reformat node to resize and reposition your image sequences to a different format. Reformat nodes also allow you to use plates of varying image resolution on a single script without running into issues when combining them.

To Insert a Reformat Node:

1. Make sure the Read node you added is currently selected.
2. Select Transform > Reformat.
   The Reformat node is inserted in the script, and its properties panel opens.

3. From the output format dropdown menu, select the format to which you want to output the sequence. If the format does not yet exist, you can select new to create a new format from scratch. The default setting, [root.format], resizes the image to the format indicated on the Project Settings dialog box.
4. You can now use the same Reformat node for any other Read nodes in the script. Simply select the Reformat node and Edit > Copy. Select another Read node in the script and Edit > Paste.
Image Caching

To ensure fast playback, Nuke uses several ways of caching data. Some of these include the following:

• The Viewer cache (also referred to as disk cache, which shares the DiskCache node's location) saves the scanlines of the displayed image to the disk cache directory. This location can be set in the Preferences panel (see The Cache Directory and Defining the Settings for Caching). When the Viewer displays an image, it reads from this cache of pre-rendered scanlines. This way, Nuke can quickly display image data when returning to a previously cached set of scanlines. You can specify the total amount of space for disk cache in Preferences > Performance > Caching > comp disk cache size. If the disk cache needs more space, the cache that was least recently used is disposed of to free more space. You can clear all of the disk cache by selecting Cache > Clear Disk Cache from the top menu bar.

Note: Clearing the Viewer cache (disk cache) also clears the DiskCache node data.

• The Buffer cache stores scanline data as calculated at selected intermediate stages in the Node Graph. This speeds up the end result of the Node Graph by caching parts of the Node Graph that haven't changed. Nuke automatically determines these intermediate cache points based on several things, including the Node Graph structure, which properties panels are open, and so on.

Tip: You can also force this behavior manually by selecting the cached checkbox in the node properties panel.

The Buffer cache uses RAM and you can specify the total amount of space in Preferences > Performance > Caching > comp cache size (%). When the buffer cache needs more space, it disposes of the first cached data that was stored (first in, first out). To clear all of the buffer cache you can either select Cache > Clear Buffers from the top menu bar, or press F12 on the keyboard.

• The Playback cache also helps improve smooth playback. Playback cache uses RAM to temporarily store the frames in the Viewer. When you first play back a clip in the Viewer, an orange bar appears automatically displaying the progress of the playback cache. After the playback cache is complete (and the orange line is complete) the frames play back smoothly. You can temporarily disable playback caching by either selecting the pause button above the Viewer or by pressing P. To clear playback cache select Cache > Clear Playback Cache.

• You can use the DiskCache node to cache parts of the Node Graph. This caches scanlines to disk from its input as they are requested by its output. DiskCache data is stored in the same place as the Viewer cache data and can share the data provided the Viewer is operating in full float (32-bit). See Using the DiskCache Node for more information.
Note: You can clear all caches by selecting **Cache > Clear All** from the menu bar.

- If you find that loading files from a remote server slows down your processing, you can localize the files that you're working with for speedier processing. See [Localizing Files for Better Performance](#).
The Cache Directory

Both the automatic caching and the DiskCache node use the same cache directory defined using the Preferences dialog. In the Preferences, you can also set the maximum size you allow the disk cache to consume. For more information on the caching preferences, see Defining the Settings for Caching below.

Note that the cached images have unique names reflecting their point of output location in the script. This means that you can cache images from multiple nodes in the script without overwriting previously cached images.

Defining the Settings for Caching

You can define the settings for caching in the Preferences > Performance > Caching.
1. Select Edit > Preferences (or press Shift+S).
   The Preferences dialog displays.
2. Under Caching > Disk Caching, specify where you want Nuke to cache out data to disk. Pick a local disk (for example, c:/temp), preferably with the fastest access time available.
   The environment variable NUKE_DISK_CACHE can be used to override this setting. For more information, see Environment Variables.
3. Using the comp disk cache size control, pick the maximum size the disk cache can reach. Ensure there is enough space on the disk for this to be reached. The default value is 10 GB, but a larger value, such as 50 GB, can often be used.
   The environment variable NUKE_DISK_CACHE_GB can be used to override this setting. For more information, see Environment Variables.
4. Click OK to update preferences and then restart Nuke.

Once these settings are defined, the automatic caching of images is enabled. The Viewer caches each frame it displays in the directory specified. If you add a DiskCache node into your script, it also uses the same directory.

The disk cache is preserved when you quit Nuke so it can be used again later. However, when the cache becomes full, old items are automatically deleted to make room for newer items.

Clearing the Disk Cache

When the disk cache becomes full, old items are automatically deleted. If necessary, you can also empty the disk cache manually. You may want to do this if, for some reason, wrong images are
displayed in the Viewer.

To Empty the Disk Cache

From the menu bar, select **Cache > Clear Disk Cache**.
Using the DiskCache Node

The DiskCache node caches to disk scanlines from its input as they are requested by its output. This can be useful, for example, if:

- you are working on a large, complex node tree. Using the DiskCache node, you can break the node tree into smaller sections and cache any branches that you are no longer working on.
- you are reading in images from a network. If you insert a DiskCache node after a Read node, the image is cached locally and displayed faster.
- you are painting or rotoscoping. If you insert a DiskCache node before a RotoPaint node, flipping frames becomes faster.

The cached images are saved in the same directory as the images the Nuke Viewer caches automatically. You can set the location and size of this directory in the Preferences. For more information, see Defining the Settings for Caching.

**Note:** Even though the DiskCache node and the automatic Viewer caching use the same cache directory, they do not share the same cache files. Therefore, using a DiskCache node does not create cache files for the Viewer and does not necessarily speed up playback. Instead, if placed at strategic, expensive parts of a node tree, it can speed up calculations, as Nuke can reference the cached data rather than recalculate it. Unlike the images in the Viewer cache, the images created by the DiskCache node affect your rendered output and are always saved as full floating point images. If you make a change in the nodes upstream, the affected cached images are discarded and automatically recalculated.

**Note:** When executing a script on the command line, the DiskCache nodes are NOT automatically executed.

To Cache Images Upstream

1. Set the zoom level in the Viewer. By default, only the lines displayed in the Viewer are cached.
2. Select **Other > DiskCache** to insert a DiskCache node after the last node in the section of the node tree that you want to cache.
3. From the **channels** dropdown menu, select which channels to cache.
Nuke caches the selected channels of the current frame at the current zoom level. From this point on, Nuke references the cached data instead of constantly recalculating the output of the preceding nodes.

As you pan and zoom around, new parts of the image are cached.

4. If you want to cache more than the current frame and zoom level, click the **Precache** button in the DiskCache properties and enter a frame range in the dialog that opens. This forces Nuke to cache all frames specified. All lines are cached regardless of what is shown in the Viewer. Where the required images are partly cached already, Nuke only calculates what is missing.
Localizing Files for Better Performance

You may find that sometimes loading files from a remote server slows down processing. Nuke has the facility to cache files locally, either individually or by setting an automatically localized folder (NUKE_TEMP_DIR/localize, by default), to help guarantee playback stability. Local caching is controlled initially in the Preferences dialog, then on a file-by-file basis.

Setting Localization Preferences

1. Press Shift+S to open the Preferences dialog and navigate to Performance > Localization.

2. Set the required localization mode:
   - **on** - checks for updates to source clips for all localization policies, and localizes those files set to **On** or **From auto-localize path** automatically. **On demand** source clips must always be localized manually.
   - **manual** - checks for updates to localized source clips and prompts you to update them manually. Only those with the policy set to **off** are not updated.
351

3. Set the default localization policy for new Read nodes using the dropdown:

- **off** - no source clips are localized, regardless of the their localization policy.

- **on** - always localize Read nodes with this policy.
- **from auto-localize path** - localize these Read nodes automatically if they reside in the auto-localize from directory.
- **on demand** - only localize these Read nodes when you manually update them. See Updating On Demand Clips for more information.
- **off** - never localize these Read nodes.

4. Enter a file path for **auto-localize from**, if required.
   Any files that reside in this directory are automatically cached when brought into Nuke, providing that the Read node’s localization policy is set to **from auto-localize path**.

5. Enter a file path for **localize to**. Leaving this field as the default creates a sub-directory in the Temp Directory as the local cache.

6. Enter a value for **limit to (GB)** to control how much disk space is available in the cache directory.

   - **Note:** The current localization mode is displayed in the status bar at the bottom-right of the interface.

   ![Status Bar](image)

   - **Note:** The current localization mode is displayed in the status bar at the bottom-right of the interface.

   - **Note:** The localization policy for existing Read nodes in the script must be set individually. See Managing Localization for more information.

   - **Note:** Localization in both the timeline and Node Graph is paused during compositing Viewer playback so that performance is not affected.

   - **Note:** On Windows, files saved to the localize to directory replace \ (double back slashes) and : (colon drive signifiers) with underscores so that the file path works as expected between operating systems. For example:

     - ```\windowspath\to\my\network\file.dpx``` is saved as ```_windowspath\to\my\network\file.dpx```  
     - ```windowspath\to\my\network\file.dpx```  
     - ```t:\my\network\path\file.dpx``` is saved as ```t_my\network\path\file.dpx```
Note: Negative values in this field reserve the specified amount of space at all times. For example, -2 stops 2 GB of memory being used for caching.

7. You can specify the time interval (in minutes) before localized files are checked for updates using the check for updated files every ## mins control. The default setting checks for new versions of files every 30 minutes, but you can set this control to any value. Any Read nodes that have changed since the last update check are flagged red in the Node Graph.

Tip: You can change the default localization indicators using the Appearance controls.

If read source files when localized files are out of date is enabled, Read nodes referencing cached files that have changed since they were localized revert to reading the source files. Read nodes that are reading from the source files are marked with a striped red bar on the Read node.
Enabling hide out of date progress bar hides the localization state of out of date files so that the Read node appears the same as regular Read nodes.

**Note:** The out of date localized files are not discarded, disabling read source files when localized files are out of date picks up the out of date files instead of reading the source files.

Managing Localization

As well as the overall Preferences for when new Read nodes should be localized, you can set localization on a file-by-file basis. The localization policy for any existing Read nodes in your project must be managed individually.

**Tip:** If you find that localization is slow to copy files, you can increase the number of threads that Nuke uses to process jobs. Set the NUKE_LOCALIZATION_NUMWATCHERS environment variable to the number of threads you want to use. See Environment Variables for more information.

1. Open the Read or ReadGeo node's Properties panel.
2. Set the localization policy dropdown to one of the following options:
• **On** - the files are localized, regardless of location, as long as the limit to (GB) limit is not breached.

• **From auto-localize path** - the files are localized if they reside in the auto-localize from directory, as long as the limit to (GB) limit is not breached.

• **On Demand** - the files are only localized when you update them manually. See <XREF> for more information.

• **Off** - the files are never localized, regardless of location.

As Read nodes are localized, an amber progress bar displays in the thumbnail. Fully cached clips are marked with a green bar at the top of the thumbnail and out-of-date clips are marked in red.

**Note:** Container formats, such as `.mov` and `.r3d`, do not display progress bars during localization. The Read node only shows the green indicator once the entire file is localized successfully.

3. If you need to pause localization temporarily, navigate to **Cache > Localization** and select **Pause**.
4. If you find that your cache is filling up regularly, you can:
   • increase the amount of available space for localization by raising the limit to (GB) preference,
   • navigate to Cache > Localization > Delete Unused Local Files (see Clearing Localized Files), or
   • manually clear files from the cache directory in NUKE_TEMP_DIR/localize, by default.
5. You can force Nuke to check for updated source files, rather than waiting for the check for updated files every interval to expire, by selecting Force Update for All files, just Selected Read nodes, or On Demand only.

   **Note:** Each file has its own update time, which is reset whenever the source files are checked.

The following table is a quick reference guide to when and how Read nodes are localized, where:
   • **green** - Read nodes are localized automatically.
   • **amber** - Read nodes are localized when updated manually.
   • **red** - Read nodes are not localized.

<table>
<thead>
<tr>
<th>System Preference</th>
<th>Read Node Preference</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>on</td>
</tr>
<tr>
<td>on</td>
<td>green</td>
</tr>
<tr>
<td>manual</td>
<td>amber</td>
</tr>
<tr>
<td>off</td>
<td>red</td>
</tr>
</tbody>
</table>

### Updating On Demand Clips

Read nodes with their localization policy set to on demand are polled to check for updates at the interval set in the Preferences. If the file has changed, the Read node is marked with a red bar to show it is out of date.

To update a single on demand Read node:
1. Double-click the Read node in the Node Graph to display its Properties.
2. Click Update.
Note: If a Read node's localization policy is set to on demand and it hasn't been localized previously, clicking the Update button localizes the file.

The local copy of the files is updated from the remote source files.

To update all on demand Read nodes:
1. Navigate to Localization > Force Update.
2. Click On demand only.

The local copy of all on demand Read nodes is updated from the remote source files.

Clearing Localized Files

Localizing a large amount of files can fill up the localization cache quite quickly if you leave the limit to (GB) preference at the default 10 GB. When the cache runs out of space, a Failed to Localize File
dialog displays and localization pauses.

You can delete localized files by clicking **Delete Unused Local Files** (or by navigating to **Cache > Localization > Clear Unused Local Files**). Nuke displays a dialog containing all the files that are marked for delete.

**Tip:** You can also open the **Preferences** from this dialog to adjust the localization behavior, such as increasing the **limit to (GB)** preference.

Click **Continue** to delete the localized files or **Cancel** to keep the cached files.
Saving Scripts and Recovering Back-Ups

You know the mantra: save and save often.

Nuke provides three ways to save your scripts, or comps, making it easy to version them. There's also an automatic timed backup, which you can turn off if you're feeling brave - but we sure don't recommend it.

Saving Scripts

There are three ways of saving scripts:
• To save a new script, select File > Save Comp as (or press Shift+Ctrl/Cmd+S).
• To update changes to a script already saved, File > Save Comp (or press Ctrl/Cmd+S).
• To save and upgrade to the next version, File > Save New Comp Version (or press Alt+Shift+S).

To Save a Script
1. Select File > Save Comp as. The Save script as dialog opens.
2. Browse to the directory where you want to store the script. For instructions on using the file browser, see Using the File Browser.
3. In the field in the bottom of the dialog, enter a name for the script after the folder path, for example firstscript_v01.nk.
4. Click Save.

Tip: The _v01 string in the end of a script name allows you to use the Save New Comp Version feature. Selecting File > Save New Comp Version saves the current version of your script and increments its name (that is, saves the different versions under different names using _v01, _v02, _v03, and so on, in the end of file names). This only works when the file name includes a number that can be incremented.
Automatic Back-Up of Scripts

You can define where and how often Nuke makes automatic back-ups your files, or turn off the autosave function.

To Define Autosave Options for a Script

1. Select Edit > Preferences.
   The Preferences > General panel opens.
2. Edit the following settings:
   - idle comp autosave after - to define how long (in seconds) Nuke waits before performing an automatic back-up after you have left the system idle.
   - force comp autosave after - to define how long (in seconds) Nuke waits before performing an automatic back-up regardless of whether the system is idle.
   - autosave comp filename - to define where and under what name Nuke saves your automatic back-up files. By default, the files are saved in the same folder as your project files with the extension .autosave. To change this, enter a full directory path name in the autosave filename field.
3. Click Save.

   Note: For the automatic back-up to work, you must save your script first so that the autosave can reference the file. We’d hate for you to lose your work, so please do this early on in the process!

To Turn off Automatic Back-Up

1. Select Edit > Preferences.
   The Preferences > General panel opens.
2. Set the idle comp autosave after and force comp autosave after fields to 0.
   From now on, Nuke does not perform any automatic back-ups, and you are more likely to lose your work in the case of a system or power failure.

Recovering Back-Ups

After experiencing a system or power failure, you are likely to want to recover the back-up files created by Nuke’s autosave function.
   A dialog opens that asks you if you want to recover the autosave file.
2. Click **OK**.
   Nuke opens the back-up file for your use.

There may be times when you don’t want to load the autosave file and rather need to load the last saved version. For example, consider a situation where you modified a script, but decided not to commit the changes and so exited Nuke without saving. In all likelihood Nuke autosaved some or all of your changes, in which case if you open the autosaved file you are not working on the original script, as intended. If you accidentally open an autosaved script, then simply close it and reload the last saved version.

---

**Note:** Breakpad crash reporting allows you to submit crash dumps to Foundry in the unlikely event of a crash.

By default, crash reporting is enabled in GUI mode and disabled in terminal mode. You can toggle reporting on and off using the `--crashhandling 1` or `0` option on the command line or by setting the environment variable `NUKE_CRASH_HANDLING` to 1 or 0.

When crash handling is enabled in GUI mode, you can control whether reports are automatically submitted or not using the `--nocrashprompt` command line option or by setting the environment variable `NO_CRASH_PROMPT` to 0.

Crashes in terminal mode are automatically submitted when crash handling is enabled.
Closing Scripts

To close a script:

1. Select **File > Close Comp** (or press **Ctrl/Cmd+W**).
2. If you have made any unsaved changes to the script, Nuke prompts you to select whether to save them. Click **Yes** to save your changes or **No** to ignore them.

Nuke quits and relaunches, as though you ran it again. It does everything it does at start-up, apart from displaying the splash screen. Therefore, you can use **Ctrl/Cmd+W** as a quick way to clear memory, reread plug-in paths, and reload the **init.py** and **menu.py** files. (The **init.py** and **menu.py** files are files that Nuke runs at start-up and can be used for configuring Nuke. You may want to reload these files if you have made any changes to them.)
Loading Scripts

When you have built a script, or comp, and saved it and want to come back to it later, you need to load in an entire script file. You recognize Nuke's script files from the extension .nk (for example firstscript.nk).

1. Select File > Open Comp (or press Ctrl/Cmd+O).
   The Script to open dialog appears.
2. Browse to the script you want to open. For instructions on using the file browser, see Using the File Browser.
3. Click Open.

Note: Some NukeX plug-ins are not supported by Nuke, and likewise, some Nuke nodes are not supported in Nuke Assist. Unsupported plug-ins are displayed in the Node Graph with errors, and unsupported nodes are outlined in red.

The Viewer renders the output of the node tree whether the components are supported or not, but you cannot modify the output of unsupported plug-ins and nodes.
Defining Frame Ranges

Several dialogs in Nuke, such as the **Frames to render** and **Frames to flipbook** dialogs, prompt you for a frame range. To define one, you need to enter a starting frame and an ending frame, separated by a dash. For example, to restrict an action to frames 1, 2, 3, 4, and 5, you would use **1-5** as the frame range.

The following table gives you more examples of frame ranges you can define.

<table>
<thead>
<tr>
<th>Frame Range</th>
<th>Expands To</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>-3</td>
<td>-3</td>
</tr>
<tr>
<td>1 3 4 8</td>
<td>1, 3, 4, 8</td>
</tr>
<tr>
<td>1-10</td>
<td>1, 2, 3, 4, 5, 6, 7, 8, 9, 10</td>
</tr>
<tr>
<td>-3-4</td>
<td>-3, -2, -1, 0, 1, 2, 3, 4</td>
</tr>
<tr>
<td>-8--5</td>
<td>-8, -7, -6, -5</td>
</tr>
<tr>
<td>1-10×2</td>
<td>1, 3, 5, 7, 9</td>
</tr>
<tr>
<td>(frame range from 1 to 10 in steps of 2)</td>
<td></td>
</tr>
<tr>
<td>1-10×3</td>
<td>1, 4, 7, 10</td>
</tr>
<tr>
<td>(frame range from 1 to 10 in steps of 3)</td>
<td></td>
</tr>
<tr>
<td>1-4×1 8-10×1 12-14×1 (multiple ranges separated by spaces)</td>
<td>1, 2, 3, 4, 8, 9, 10, 12, 13, 14</td>
</tr>
</tbody>
</table>

You can use the above ways of defining a frame range everywhere in Nuke. In addition to dialogs, they can be used on the command line (where any frame ranges should be preceded by the -F switch) and in Python statements. For more information, see Command Line Operations and the Nuke Python documentation (*Help > Documentation*).
Reformatting Elements

These pages teach you how to reformat images through scaling, cropping, and pixel aspect adjustments. You will also learn to adjust bounding boxes to minimize processing and rendering times.

Using the Reformat Node

You can use the Reformat node for three different purposes:

1. To generate image sequences that match a desired image format in terms of both resolution and pixel aspect ratio (the width to height ratio of the format's individual pixels).
2. To create thumbnails (low resolution frames which you might post to the web in order to storyboard a sequence). The node scales the frame until it fits inside a rectangle whose dimensions you specify. It also sets pixel aspect ratio to one (square).
3. To scale images. The scale factor is rounded slightly so that the output image has an integer number of pixels in the direction you select in the Reformat node's controls.

Converting Images to a Desired Image Format

When you read in elements, Nuke stores their format settings and makes them available to the Reformat node. You can then apply one of the existing formats to your images, or create, edit, and delete formats yourself.

When creating a new format from scratch, you define the overall resolution, the cropped resolution (optional) and the pixel aspect ratio. As you define these parameters, the Reformat operator graphically displays them for you in the manner shown below.

![Diagram showing full dimensions, pixel dimensions, and crop dimensions]
To Create a New Output Format:

1. Click **Transform > Reformat** to insert a Reformat node at an appropriate place in your script (generally before a Write node).
2. Connect a Viewer to the output of the Reformat node so you can see the effect of your changes.
3. Select **new** from the **output format** dropdown menu. The **New format** dialog appears.
4. Type a name for the new format in the **name** field.
5. In the **file size** fields, type the full output resolution (in pixels).
6. If you want to crop the full output resolution (for example, to create a letter box):
   - Check **image area**.
   - Increment the **x** field to define the left boundary of the crop. (The display updates to show you the left boundary of the crop relative to the full-size input.)
   - Increment the **y** field to define the bottom boundary of the crop.
   - Increment the **r** field to define the right boundary of the crop.
   - Increment the **t** field to define the top boundary of the crop.
7. If the destination display device for the image sequence uses non-square pixels, type the appropriate pixel aspect ratio in the **pixel aspect** field (for example, if your destination is a digital video display, type 1.1).

**Note:** You can also add formats to Nuke via entries to the **menu.py** file:
1. Open the **menu.py** file (located in same directory as your Nuke executable).
2. Add an entry similar to the following example:
   ```python
   nuke.addFormat ("720 486 0 0 720 486 0.9 NTSC_video")
   ```
   where the numbers specify, respectively, the format's full horizontal resolution, full vertical resolution, left crop position, bottom crop position, right crop position, top crop position, and pixel aspect ratio; and where the final text string designates the format's name.
3. Save and close the **menu.py** file. The next time you launch Nuke the format is available for selection from the **Project Settings** dialog, Reformat node properties panel, and elsewhere.

To Edit a Format:

1. Select the format you wish to edit from the **output format** dropdown menu.
2. From the same dropdown menu, select **edit**. The **Edit format** dialog appears.
3. Edit the **name, file size, image area**, and **pixel aspect** fields as necessary.
4. Click **OK** to save the changes to the format.
To Delete a Format:
1. Select the format you wish to delete from the output format dropdown menu.
2. From the same dropdown menu, select delete. The format is removed from the menu.

To Apply a Format:
1. If necessary, click Transform > Reformat to insert a Reformat node at an appropriate place in your script (generally before a Write node).
2. Connect a Viewer to the output of the Reformat node so you can see the effect of your changes.
3. From the type dropdown menu, select to format.
4. Select the format you wish to apply from the output format dropdown menu.
5. From the resize type field, select the method by which you want to preserve or override the original aspect ratio. Select:
   - width to scale the original until its width matches the format's width. Height is then scaled in such a manner as to preserve the original aspect ratio.
   - height to scale the original until its height matches the format's height. Width is then scaled in such a manner as to preserve the original aspect ratio.
   - fit to scale the original until its smallest side matches the format's smallest side. The original's longer side is then scaled in such a manner as to preserve original aspect ratio.
   - fill to scale the original until its longest side matches the format's longest side. The input's shorter side is then scaled in such a manner as to preserve original aspect ratio.
   - distort to scale the original until all its sides match the lengths specified by the format. This option does not preserve the original aspect ratio, so distortions may occur.
6. When cropping the output, check center to position the crop area at the center of the frame.
7. Select the appropriate filtering algorithm from the filter dropdown menu (see Choosing a Filtering Algorithm).
8. When scaling an image with Key, Simon, and Rifmen filters, you may see a haloing effect which is caused by pixel sharpening these filters employ. If necessary, check clamp to correct this problem.

Creating Thumbnails
1. Click Transform > Reformat to insert a Reformat node at an appropriate place in your script (generally before a Write node).
2. Connect a Viewer to the output of the Reformat node so you can see the effect of your changes.
3. From the type dropdown menu, select to box.
4. In the width and height fields, type the output dimensions. The units are pixels.
5. Use the **resize type** dropdown menu to select the method by which you preserve or override the original pixel aspect ratio. Select:
   - **width** to scale the original until its width matches the value in the **width** field. Height is then scaled in such a manner as to preserve the original aspect ratio (this means that the output you specified in **height** may not match the result).
   - **height** to scale the original until its height matches the value in the **height** field. Width is then scaled in such a manner as to preserve the original aspect ratio (this means that the output you specified in **width** may not match the result).
   - **fit** to scale the original until its smallest side matches the corresponding value in **width/height**. The longer side is then scaled in such a manner as to preserve the original aspect ratio.
   - **fill** to scale the original until its longest side matches the corresponding value in **width/height**. The smallest side is then scaled in such a manner as to preserve the original aspect ratio.
   - **distort** to scale the original until its sides match the values in the **width/height** fields. This option does not preserve the original aspect ratio, so distortions may occur.

6. Select the appropriate filtering algorithm from the **filter** dropdown menu (see **Choosing a Filtering Algorithm**).

7. When scaling an image with Key, Simon, and Rifmen filters, you may see a haloing effect which is caused by pixel sharpening these filters employ. If necessary, check **clamp** to correct this problem.

**Scaling Image Sequences**

1. Click **Transform > Reformat** to insert a Reformat node at an appropriate place in your script (generally before a Write node).

2. Connect a Viewer to the output of the Reformat node so you can see the effect of your changes.

3. From the type dropdown menu, select **scale**.

4. In the **scale** fields, enter scale factors for the width and the height. To scale each direction separately using different scale factors, click the **2** button.

5. Use the **resize type** dropdown menu to select the method by which you preserve or override the original pixel aspect ratio. Select:
   - **width** to scale the original so that it fills the output width. Height is then scaled in such a manner as to preserve the original aspect ratio.
   - **height** to scale the original so that it fills the output height. Width is then scaled in such a manner as to preserve the original aspect ratio.
   - **fit** to scale the original so that its smallest side fills the output width or height. The longest side is then scaled in such a manner as to preserve the original aspect ratio.
   - **fill** to scale the original so that its longest side fills the output width or height. The smallest side is then scaled in such a manner as to preserve the original aspect ratio.
• **distort** to scale the original so that both sides fill the output dimensions. This option does not preserve the original aspect ratio, so distortions may occur.

6. Select the appropriate filtering algorithm from the **filter** dropdown menu (see Choosing a Filtering Algorithm).

7. When scaling an image with Key, Simon, and Rifmen filters, you may see a haloing effect which is caused by pixel sharpening these filters employ. If necessary, check **clamp** to correct this problem.

### Cropping Elements

To **crop** a frame is to cut out the unwanted portions of the image area.

The original image.  
Cropping the image.

1. Click **Transform > Crop** to insert a Crop node at an appropriate place in your script.
2. Connect a Viewer to the output of the Crop node so you can see the effect of your changes.
3. Define the crop boundaries:
   • In the Viewer, drag on any side of the frame to reposition it.

• Or, in the Crop properties panel, increment or decrement the **box** field (x stands for left side, y for bottom side, r for right side, and t for top side).

4. To fill the cropped portion with black, check **black outside**. To fill the cropped portion by expanding the edges of the image, uncheck **black outside**. To adjust the image output format to match the cropped image, check **reformat**.

5. If you wish to vignette the edges of the cropped portion, increment the **softness** field.
Reformatting Elements | Cropping Elements
Adjusting the Bounding Box

The bounding box defines the area of the frame that Nuke sees as having valid image data. The larger the bounding box is, the longer it takes Nuke to process and render the image. To minimize processing and rendering times, you can crop the bounding box. Occasionally, the bounding box may also be too small, in which case you need to expand it.

**Note:** Other Nuke functions, such as Transforms and Merges, can also affect the size of the bounding box, see Transforming in 2D and Merging Images for more information.

Resizing the Bounding Box

To adjust the bounding box, you can use the AdjBBox and CopyBBox nodes. The AdjBBox node crops and expands the bounding box edges, whereas with the CopyBBox node you can copy a bounding box from one input to another. If needed, you can also add a black outside edge to the bounding box using the BlackOutside node.

The AdjBBox node expands or crops the edges of the bounding box by a specified number of pixels.

![An expanded bounding box. A cropped bounding box.](image)

**Tip:** You can enable a warning to indicate when the bounding box is greater that the format in Nuke's Preferences. See Bounding Box Warnings for more information.

For example, if you have an image with lots of black (0,0,0,0), you can adjust the bounding box to contain just the useful area so that Nuke won't waste time computing results where there is no change.
1. Select **Transform > AdjustBBox** to insert an AdjBBox node after the image whose bounding box you want to resize.

2. Connect a Viewer to the AdjBBox node, so you can see the effect of your changes.

3. In the AdjBBox controls, adjust the **Add Pixels** slider to increase or decrease the size of the bounding box. By default, 25 pixels are added to the edges of the bounding box.

   Nuke expands or crops the edges of the bounding box. If the bounding box is cropped, whatever is outside the bounding box area is replicated towards the edges of the image.

![Adjusted Bounding Box Example](image)

**Copying a Bounding Box from One Input to Another**

Some Nuke operations, such as a merge, can cause an expansion of the bounding box area because Nuke does not know that the extra area is going to be black or another constant color. Often, you can fix this by copying the bounding box from one of the inputs to the resulting image, thus cutting off this extra area. For this, you can use the CopyBBox node.

1. Select **Merge > CopyBBox** to insert a CopyBBox node after the node whose bounding box you want to use.

2. Connect the image whose bounding box you want to copy to the CopyBBox node's input A, and the image onto which you want to copy the bounding box to input B.

   Nuke copies the bounding box from input A to input B. Whatever is outside the copied bounding box area in image B gets replicated towards the edges of the image.

![Copied Bounding Box Example](image)
Adding a Black Outside Edge to the Bounding Box

If you adjust a bounding box with the AdjBBox or CopyBBox node, you may notice that whatever is outside the bounding box area gets replicated towards the edges of the image. If necessary, you can remove these replicated edge pixels and fill everything outside the bounding box area with black. To do this, use the BlackOutside node.

A cropped bbox with replicated edges. The effect of the BlackOutside node.

1. Select the image whose edges outside the bounding box you want to fill with black.
2. Select Transform > BlackOutside to add a BlackOutside node in an appropriate place in your script.
   Nuke fills everything outside the bounding box area with black.

Bounding Box Warnings

Zooming into the Viewer to work on a shot means that you can't always see the extent of the bounding box in relation to the format, which can result in unnecessary processing.

To make it easier to see the state of your bounding box, Nuke can display visual warnings on the nodes that affect the bounding box.

⚠️ Warning: If you enable the Bounding Box Warning, Nuke performs extra processing steps to identify problematic nodes, which may result in performance degradation.

To enable the warnings, in Nuke's Preferences under Panels > Node Graph, enable Bounding Box Warning:
- red rectangle with dotted stroke - the indicated node creates a bounding box greater than the format.
• **dotted stroke without the red rectangle** - the bounding box size is greater than the format at the indicated node, but the bounding box size has been set by an upstream node.

The bbox warning **threshold** controls how far past the edge of the format the bounding box can expand before the warning is displayed in the Node Graph. For example, if you’re working with UHD_4K footage and the default 10% threshold, you can expand the bounding box horizontally by 384 pixels before the warning is displayed.
Tip: You can set the color of the warning rectangle in the Preferences under Panels > Node Graph > Bounding Box Warning.
Channels

Digital images generally consist of the four standard channels: red, green, blue, and alpha. Nuke allows you to create or import additional channels as masks, lighting passes, and other types of image data.

Introduction

A Nuke script can include up to 1023 uniquely named channels per compositing script. For example, you can combine multiple render passes from a 3D scene - an image from the red, green, and blue channels, a depth mask (z-depth channel), a shadow pass, a specular pass, lighting passes, and multiple mattes all stored within one image sequence in your composite.

Note: When a script is saved, any channels that are not referenced in the script are discarded automatically.

When creating channels and layers, bear in mind these good practice guidelines:

• Ensure that all layers use the same channel names in the same order. This avoids complications with multilayer .exr files imported into Nuke.
• Always use proper names for channels, never just a single letter.
• Always create a custom layer for custom channels, don't add to the existing default layers.
• Never use more than four channels per layer. Nuke only has a four channel interface.

A current Channel Count is displayed in the bottom-right of the interface, which changes color as the number of channels increases. The default threshold is 1023, but you can set the limit in the Preferences under Project Defaults > Channel Management > Channel Warning Threshold.

The Channel Count turns yellow if you exceed the Channel Warning Threshold and red if the Channel Count is equal to or greater than the maximum channel value 1023.

Channel Count: 1023, Localization Mode: On, Memory: 0.5 GB (1.7%)
Quick Start

Here's a quick overview of the workflow:

1. Channels in Nuke are always a part of a layer. You can create new channels and layers using the **new** option in the channel selection dropdown menus (such as **output** and **mask**) in a node's properties panel. For more information, see Object Material Properties.

2. Using the channel selection controls you can select which channels the node is processing and outputting, or using as a mask when color correcting for instance. For more information, see Calling Channels and.

3. The channels can also be linked to other channel controls through the **Link menu**. For more information, see Linking Channels Using the Link Menu.

4. Using the Shuffle and ShuffleCopy nodes, you can rearrange your input channels and apply the result in the output. For more information, see Swapping Channels.
Understanding Channels and Layers

At a very basic level, **channels** in Nuke carry image data and **layers** are containers for these channels, up to the maximum of 1023 channels per script. When elements are rendered out in the OpenEXR format, for example, you can operate on multiple channels from a single image.

![Diagram of channels and layers]

The layer name, in this case rgba, depth, or masks, followed by the channel name.

Channels

Think of a channel as a container that contains image data. Once created or read into your composite, the image data stored in a channel is available downstream in the network until the value is replaced with something else or the channel is removed. The channel may even be “empty” - depending on where you reference it in the compositing network.

Layers

All channels in a script must exist as part of a layer (also called a channel set). You’re probably familiar with the default layer - rgba - which includes the channels with pixel values of red, green, and blue, and also the alpha channel for transparency.

All channels in a composite must belong to at least one layer. Some channels, like alpha, may be available in other layers, too. Channel names always include the layer name as a prefix, like this:
layer_name.channel_name.

By default, every script has a layer called `rgba`. When you first import an image element, Nuke automatically assigns its channels to the rgba layer - that is, the image channels are named `rgba.red`, `rgba.blue`, `rgba.green`, and `rgba.alpha`.

The `rgba` layer allows for the standard four-channel workflow of most node-based compositing systems. However, you're not limited to these four channels. You can create new channels and assign them to new layers up to the limit of 1023 channels per script.
Creating Channels and Layers

It's important to understand that many types of nodes allow you to direct their output to a specific channel and parent layer. You have the option of processing these channels in each subsequent node, or leaving them unchanged.

Many nodes feature an output or channels setting, which lets you direct the output of the current node to a specific layer and channel. You can also use the output or channels dropdown menu to create new layers and channels.

Some nodes do not include an output or channels setting in their parameters. For these, you can connect other nodes, such as Channel Copy or Shuffle, to create and manage channel output in the node tree.

To Create a New Layer and/or Channel

1. Open the properties panel for the node whose output creates the new channel.
2. From the output or channels dropdown menu, select new.
3. Under Name, enter the name of the layer, and under Channels the new channel name.

   ![Layer and Channel Creation](image)

   **Note:** You can either use a new layer name to create a new layer, or enter a layer you've created previously. You can't create new channels into layers that are built into Nuke (such as mask).

4. Click OK.
Note: You can also create new channels with the Shuffle and ShuffleCopy nodes. These are explained later, under Swapping Channels.
Calling Channels

By default, most nodes in Nuke attempt to process the current channels in the rgba set and place the output in those same channels. However, many nodes also contain an input dropdown menu which lets you select the channels you want to process, and an output dropdown menu to select the channel(s) where the results should be stored.

Some nodes also contain mask controls and a mask input connector, which let you select a channel for use as a matte to limit operations such as color corrections. Using these mechanisms, you can point the output of almost any node in the script to any available channel.

The script below attempts to clarify these concepts. Note the script generates six channels (though it could just as well generate 1023). The steps below describe how each channel was created.
A six-channel script.

1. The script reads in the foreground, creating three channels (red, green, and blue), which are by default assigned to the rgba set. Channel count: 3
2. A low contrast key (soft) is pulled and assigned to a new layer called mattes. Channel count: 4
3. A high contrast key (hard) is pulled and also assigned to the mattes layer. Channel count: 5
4. The mattes.hard and mattes.soft channels are mixed to form the final matte (alpha), which is assigned to the rgba layer. Channel count: 6

Suppose now that you wanted to perform a color correction using the output of the Soft Matte node as a mask for the correction. There's no need to pipe the output from that Soft Matte node - it already exists in the data stream along with the other five channels that were created.

You simply attach a color correction node, such as HueCorrect, then select the appropriate channel from the mask controls, for example mattes.soft.

**Note:** The mattes portion of the name indicates the parent layer.

### Viewing Channels in the Viewer

You can view the individual red, green, blue, and alpha channels in the Viewer using the R,G,B, and A keys on the keyboard. For more information, see Compositing Viewers and Using the Timeline Viewer.
Selecting Input Channels

A node's **channels** field lets you select one or several channels for processing.

**To Select a Single Input Channel**

1. Open the properties panel of the node into which you wish to feed a channel.
2. From the **channels** field, select **none**.
3. From the right most channel field - the one which typically calls the alpha channel - select the single channel you wish to process.

![single channel](image)

**To Select Multiple Input Channels**

1. Open the properties panel of the node into which you wish to feed channels.
2. From the **channels** field, select the layer containing the channels you wish to process.
   - The layer's channels appear with check boxes.

![selected channel set available channels](image)

3. Deselect those channels which you don't wish to process. The node processes all those you leave selected.

Selecting Masks

The **mask** controls in a node's properties panel let you select a single channel for use as a matte in a given process (typically, a color correction). The given process thereafter is limited to the non-black areas of the selected channel.

You can use one of the script's existing channels as the matte, or attach a mask to the node with a mask input connector.

You can find mask input connectors on color correction and filter nodes, such as HueCorrect and Blur. At first, they appear as triangles on the right side of the nodes, but when you drag them, they turn into arrows labeled **mask**. You connect them the same way as any other connectors. If you
cannot see a mask input connector, open the node's properties panel and make sure \textbf{mask} is set to \textbf{none}.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{mask_input_connector.png}
\caption{Before dragging the mask input connector.}
\end{figure}

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{mask_input_connector.png}
\caption{When dragging the mask input connector.}
\end{figure}

\textbf{To Select a Channel for Use as a Matte from the Mask Input}

1. Connect a mask to the node with its mask input connector.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{mask_input_connector.png}
\caption{Selecting a Channel for Use as a Matte from the Mask Input}
\end{figure}

If you cannot see the mask input connector, open the node's controls and make sure \textbf{mask} is set to \textbf{none}.

By default, when a mask input is connected, the node uses the alpha channel from it as a matte.

2. If you don't want to use the alpha channel as the matte, select the channel you want to use from the \textbf{mask} dropdown menu.

3. If you want the mask from the \textbf{mask} input copied into the predefined \textbf{mask.a} channel, check \textbf{inject}. This way, you can use the last mask input again downstream. You can also set a stream of nodes to use \textbf{mask.a} as the mask, and then change the masking of all of them by simply connecting a new mask into the mask input connector of the first node.

4. If necessary, check the \textbf{invert} box to reverse the mask.

\textbf{Tip:} The \textbf{invert} control also affects any injected masks from upstream nodes.
5. If the overall effect of the node is too harsh, you can blend back in some of the input image by adjusting the **mix** slider.

**To Select a Channel for Use as a Matte from the Main Input**

1. Make sure nothing is connected to the node’s mask input connector. If you disconnect a mask input, the mask input connector disappears, as it is no longer being used.
2. Select the channel you want to use from the **mask** dropdown menu.
3. If necessary, check the **invert** box to reverse the mask.
4. If the overall effect of the node is too harsh, you can blend back in some of the input image by adjusting the **mix** slider.
Linking Channels Using the Link Menu

You can create expression links to connect channel and layer controls with other controls in various nodes. Since these controls aren't meant to be animated, you can't use the full range of Nuke expressions, nor can you use Python or Tcl languages. You can link controls using the Link menu next to the control on the properties panel:

1. Click the Link menu and select Set link. An Expression dialog opens.
2. Enter your expression in the Expression field and click OK. For more information on expressions, see Expressions.
3. You can edit an existing link by clicking the Link menu and selecting Edit link.
4. You can also Ctrl/Cmd+drag the Link menu to another control to create a link between the two.
5. To remove a link, click the Link menu and select Remove link.
Tracing Channels

You may have noticed that nodes visually indicate the channels which they are processing (that is, treating in some way) and passing (that is, conveying without any treatment). This is done via a system of colored rectangles, which allows you to trace the flow of channels throughout a script.

Look closer at this example node. The wide rectangles indicate channels which Nuke processes (in this case, the red, green, blue, and alpha channels). The narrow rectangles indicate channels that Nuke passes onto the next node without processing (in this case, the `mattes.soft` and `mattes.hard`).

![Visual confirmation of extra channels.](image)
Renaming Channels

In the course of building your script, you may find it necessary to replace certain channels in a layer.

1. Open the properties panel for a node which has the channel selected on the **channels**, **input**, or **output** dropdown menu.
2. Click on the dropdown menu where the channel is displayed and select **rename**.
   The **Rename Channel** dialog appears.
3. Enter a new name for the channel and click **OK**.
Removing Channels and Layers

When you are done using a layer or a channel within a layer, you may wish, for the sake of clarity, to remove it so that it is no longer passed to downstream nodes. Note that leaving channels in the stream does not itself cause them to be computed, only channels required are computed.

Note: When a script is saved, any channels that are not referenced in the script are discarded automatically.

To Remove a Layer or a Channel Within a Layer

1. Click Channel > Remove to insert a Remove node at the appropriate point in your script.
2. In the Remove properties panel, select the layer you wish to remove from the channels fields.
3. If you don't wish to remove the entire layer, uncheck the boxes corresponding to the channels which you still wish to be able to call downstream.
4. Click OK to close the properties panel.
   The layer and/or the channels you removed are no longer displayed in node parameters downstream from the Remove node.

Note: Removing layers and or channels does not free up space for the creation of new channels and layers. Once you create a channel, it permanently consumes one of the script's 1023 available channel name slots. You are free, however, to rename channels and/or assign them new outputs.
Swapping Channels

Nuke features two main nodes for channel swapping: Shuffle and ShuffleCopy. Shuffle lets you rearrange the channels from a single image (1 input) and then output the result to the next node in your compositing tree. ShuffleCopy lets you rearrange channels from two images (2 inputs) and output the result. Let’s take a look at the ShuffleCopy node first.

The ShuffleCopy matrix.

• **Channels from Input 1** - the first group of channel boxes are the channels supplied by input 1 on the node. As shown above, the foreground element’s default rgba set is selected.

• **Channels from Input 2** - the second group of channel boxes are the channels supplied by input 2 on the node.

To swap channels, do the following:

1. Click **Channel > Shuffle** or **ShuffleCopy** to insert a Shuffle or ShuffleCopy node. Remember you use Shuffle when you only want to swap channels in a single upstream node, and ShuffleCopy when you want to swap channels in two separate nodes, like a foreground and background branch.

2. Select the incoming channels from the **In 1** and **In 2** (optional) dropdown menus. You can select up to eight channels in this manner.

3. Select the layers to which you wish to direct the incoming channels from the dropdown menus on the right. You can select up to eight channels in this manner.

4. If the outgoing layer to which you wish to direct channels does not yet exist, create it using the **new** option on the dropdown menus on the right.

5. Click as necessary on the resulting matrix to swap channels.
Tip: If you just need to copy a channel from one data stream into another, use Channel > Copy, instead of ShuffleCopy. Then, specify the channel to copy and the destination channel for output.

Channel Outputs

The combination of the boxes checked in the channel matrix create the list of channels that are output to the layer selected in the top dropdown menu on the right.

This four channel stream acts as the second set of outputs from the node. It allows you to output another four channels from the node, for a total of eight channels of output to match the possible eight channels of input.

In this case, this second set of outputs has not been utilized.

Tip: While not required, it's good practice to use the first set of outputs for swapping channels in the current data stream and the second set of outputs for creating new channels. This protects the default rgba set from unintentional overwriting, and makes it easier for other artists to understand the workings of your script.

The basic process then for swapping channels is to first select your incoming layers from the 1 in and 2 in (or, in the case of the Shuffle node, the in 1 and in 2) dropdown menus. Then, select your outgoing layers from the dropdown menus on the right. Then make the actual channel swaps by clicking on the resulting matrix.

For example, to take the simplest case, suppose, you wanted to copy the red channel of an rgba set into its alpha channel. You would click on the matrix to create the following configuration.

Configuring the channel matrix.
You can see that the matrix makes use of the r channel (standing for red) twice. It goes out once as the red channel, and another time as the alpha channel.

Assigning Constants

The shuffle nodes also include parameters that let you assign white (1) or black (0) constants to any incoming channel. So, for example, to reset the alpha channel to a full-frame image, you would configure the matrix as follows:

Assigning constants to channels.

Creating Swap Layers

Finally, note that if the layer to which you wish to output channels does not yet exist, you can create it using the new option on the dropdown menus on the right. Once you select the new option, you follow the same process for creating layers as is described under Creating Channels and Layers.
Merging Images

With Nuke, you can merge images in a wide variety of ways. In these pages, we teach you how to use the Merge, ContactSheet, and CopyRectangle nodes.

Layering Images Together with the Merge Node

The Merge node with its compositing algorithms allows you to control just how your images are combined.

**Note:** When using most of the available merge algorithms, Nuke expects premultiplied input images. However, with the **matte** operation you should use unpremultiplied images.

**To Layer Images with the Merge Node**

1. Select **Merge > Merge** (or press M on the Node Graph) to insert a Merge node after the images you want to layer together.
2. Connect your images to the Merge node's **A** and **B** inputs.
3. If necessary, you can connect multiple A images to the Merge node. Once you have got the A and B inputs connected as instructed in step 2, drag more connectors from the left side of the Merge node to the images you want to use as additional A inputs.
   
   Each input is merged in the order connected, for example **A3, A2, A1, B**.
4. Connect a Viewer to the output of the Merge node so you can see the effect of your merge operation.
5. In the Merge node's controls, select how you want to layer the images together from the **operation** dropdown menu. The default and the most common operation is **over**, which layers input A over input B according to the alpha of input A. For descriptions of all the available operations, see **Merge Operations**.
6. Set which input's bounding box you want to use for the Merge output:
   
   • **union** - resize the output bbox to fit both input bboxes completely.
   • **intersection** - use only those parts of the image where the input bboxes overlap.
   • **A** or **B** - use the selected input's bbox for the output.

   See **Adjusting the Bounding Box** for more information on bounding boxes.
7. Select which input's metadata and frame range is passed down the tree using the **metadata from** and **range from** dropdowns.

8. Using the **A channels** and **B channels** dropdown menus, select which channels to use from the A and B inputs and which channels to use as the A and B alpha. If you want to merge more channels than these and output them into the same channels, select them from the **also merge** dropdown menus and checkboxes.

9. From the **output** dropdown menu, select the channels you want to write the merge of the A and B channels to. Channels named in the **also merge** dropdown menu are written to the same output channels.

10. If necessary, you can also adjust the following controls:
   - To select which input's metadata to pass down the tree, use the **metadata from** dropdown menu.

   **Note:** When **metadata from** is set to **All** and there are keys with the same name in both inputs, keys in **B** override keys in **A**.

   For more information on file metadata, see **Working with File Metadata**.
   - To dissolve between the original input B image (at 0) and the full Merge effect (at 1), adjust the **mix** slider. A small light gray square appears on the node in the node graph to indicate that the full effect is not used.
   - If you want to mask the effect of the Merge operation, select the mask channel from the **mask** dropdown menus. To invert the mask, check **invert**. To blur the edges of the mask, check **fringe**.

   Note that you should not use the alpha of the inputs for the mask. It produces erroneous results (though the error is often hard to see); you can achieve better results by turning on alpha masking.
   - From the **Set BBox to** dropdown menu, select how you want to output the bounding box. The default is **union**, which combines the two bounding boxes. You can also select **intersection** to set the bounding box to the area where the two bounding boxes overlap, **A** to use the bounding box from input A, or **B** to use the bounding box from input B.
   - By default, Nuke assumes that images are in linear color space. However, if you want to convert colors to the default 8-bit color space defined in the **LUT** tab of your project settings (usually, sRGB), check **Video colorspace**. The conversion is done before the images are composited together, and the results are converted back to linear afterwards. Any other channels than the red, green, and blue are merged without conversion.
Checking this option can be useful if you want to duplicate the results you obtained from an application that uses the standard compositing math but applies it to non-linear images (for example, Adobe® Photoshop®). In this case, you typically also need to make sure premultiplied is not checked in your Read node controls.

- By default, the same math is applied to the alpha channel as the other channels. However, according to the PDF/SVG specification, many of the merge operations (for example, overlay and hard-light) should set the alpha to \((a+b - ab)\). This way, the input images remain unchanged in the areas where the other image has zero alpha. If you want to enable this, check alpha masking.

**Bounding Box Warnings**

Zooming into the Viewer to work on a shot means that you can't always see the extent of the bounding box in relation to the format, which can result in unnecessary processing.

To make it easier to see the state of your bounding box, Nuke can display visual warnings on the nodes that affect the bounding box. To enable the warnings, in Nuke's Preferences under Panels > Node Graph, enable Bounding Box Warning:

- **red rectangle with dotted stroke** - the indicated node creates a bounding box greater than the format.

- **dotted stroke without the red rectangle** - the bounding box size is greater than the format at the indicated node, but the bounding box size has been set by an upstream node.
The bbox warning **threshold** controls how far past the edge of the format the bounding box can expand before the warning is displayed in the Node Graph. For example, if you’re working with UHD_4K footage and the default 10% threshold, you can expand the bounding box horizontally by 384 pixels before the warning is displayed.

**Tip:** You can set the color of the warning rectangle in the **Preferences** under **Panels > Node Graph > Bounding Box Warning**.
Merge Operations

When layering images with the Merge node, you need to select a compositing algorithm that determines how the pixel values from one input are calculated with the pixel values from the other to create the new pixel values that are output as the merged image.

The operation dropdown menu in the Merge node’s properties panel houses a large number of different compositing algorithms, giving you great flexibility when building your composite. The available algorithms are listed in alphabetical order.

**Tip:** With many compositing algorithms available, it may sometimes be difficult to find what you’re looking for in the operation dropdown menu. Luckily, there’s a quick way of finding a particular operation. With the menu open, you can type a letter to jump to the first operator that starts with that letter. To move to the second operation that starts with the same letter, press the letter again. For example, to select the **soft-light** operation, open the menu and press **S** twice.

The following table describes each operation and its associated compositing algorithm. There are example images to illustrate the effects, one that combines the letters A and B to a merged image and another that has an image of fire merged with the familiar checkerboard. You may want to spend some time familiarizing yourself with each algorithm in order to be able to determine which operation to use in each situation.

<table>
<thead>
<tr>
<th>Operation</th>
<th>Algorithm</th>
<th>Description</th>
<th>Illustration</th>
<th>Example Uses</th>
</tr>
</thead>
<tbody>
<tr>
<td>atop</td>
<td>Ab+B(1-a)</td>
<td>Shows the shape of image B, with A covering B where the images overlap.</td>
<td><img src="image1" alt="atop Illustration" /></td>
<td></td>
</tr>
<tr>
<td>Operation</td>
<td>Algorithm</td>
<td>Description</td>
<td>Illustration</td>
<td>Example Uses</td>
</tr>
<tr>
<td>---------------</td>
<td>----------------</td>
<td>---------------------------------------------------------------</td>
<td>--------------</td>
<td>-----------------------------------</td>
</tr>
<tr>
<td>average</td>
<td>(A+B)/2</td>
<td>The average of the two images. The result is darker than the original images.</td>
<td><img src="image" alt="average" /></td>
<td></td>
</tr>
<tr>
<td>color-burn</td>
<td>darken B towards A</td>
<td>Image B gets darker based on the luminance of A.</td>
<td><img src="image" alt="color-burn" /></td>
<td></td>
</tr>
<tr>
<td>color-dodge</td>
<td>brighten B towards A</td>
<td>Image B gets brighter based on the luminance of A.</td>
<td><img src="image" alt="color-dodge" /></td>
<td></td>
</tr>
<tr>
<td>conjoint-over</td>
<td>A+B(1-a/b), A if a&gt;b</td>
<td>Similar to the over operation, except that if a pixel is partially covered by both A and B, conjoint-over assumes A completely</td>
<td><img src="image" alt="conjoint-over" /></td>
<td></td>
</tr>
<tr>
<td>Operation</td>
<td>Algorithm</td>
<td>Description</td>
<td>Illustration</td>
<td>Example Uses</td>
</tr>
<tr>
<td>------------</td>
<td>-----------</td>
<td>------------------------------------------------------------------------------</td>
<td>--------------</td>
<td>------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>copy</td>
<td>A</td>
<td>Only shows image A.</td>
<td><img src="image" alt="Copy Illustration" /></td>
<td>This is useful if you also set the <strong>mix</strong> or <strong>mask</strong> controls so that some of B can still be seen.</td>
</tr>
<tr>
<td>difference</td>
<td>abs(A-B)</td>
<td>How much the pixels differ. Also available from <strong>Merge &gt; Merges &gt; Absminus</strong>.</td>
<td><img src="image" alt="Difference Illustration" /></td>
<td>Useful for comparing two very similar images. This mode can also be used as a difference keyer.</td>
</tr>
<tr>
<td>disjoint-over</td>
<td>A+B(1-a)/b, A+B if a+b&lt;1</td>
<td>Similar to the over operation, except that if a pixel is partially covered by both a and b, disjoint-over assumes the two objects do not overlap. For instance, two</td>
<td><img src="image" alt="Disjoint-over Illustration" /></td>
<td>This can be useful if you want to merge element a over element b, and element a has element b already held out. For example, you may</td>
</tr>
<tr>
<td>Operation</td>
<td>Algorithm</td>
<td>Description</td>
<td>Illustration</td>
<td>Example Uses</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------</td>
<td>-------------</td>
<td>--------------</td>
<td>--------------</td>
</tr>
<tr>
<td></td>
<td></td>
<td>polygons that touch and share an edge. Normal over produces a slightly transparent seam here.</td>
<td><img src="image1.png" alt="Illustration" /></td>
<td>have a CG character whose hair, skin, and clothing are rendered separately so that each object has the other objects held out of the render. In this case, using the over operation would produce dark lines around the comped objects. This is because over does a hold-out of the background image, meaning the background is held out twice.</td>
</tr>
<tr>
<td>divide</td>
<td>( \frac{A}{B}, 0 \text{ if } A&lt;0 \text{ and } B&lt;0 )</td>
<td>Divides the values but stops two negative values from becoming a positive number.</td>
<td><img src="image2.png" alt="Illustration" /></td>
<td>This does not match any photographic operation, but can be used to undo a multiply.</td>
</tr>
<tr>
<td>exclusion</td>
<td>( A+B-2AB )</td>
<td>A more photographic form of difference.</td>
<td><img src="image3.png" alt="Illustration" /></td>
<td></td>
</tr>
<tr>
<td>Operation</td>
<td>Algorithm</td>
<td>Description</td>
<td>Illustration</td>
<td>Example Uses</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------</td>
<td>-------------</td>
<td>--------------</td>
<td>--------------</td>
</tr>
<tr>
<td>from</td>
<td>B-A</td>
<td>Image A is subtracted from B.</td>
<td><img src="image1.jpg" alt="Illustration" /></td>
<td><img src="image2.jpg" alt="Example Uses" /></td>
</tr>
<tr>
<td>geometric</td>
<td>2AB/(A+B)</td>
<td>Another way of averaging two images.</td>
<td><img src="image3.jpg" alt="Illustration" /></td>
<td><img src="image4.jpg" alt="Example Uses" /></td>
</tr>
<tr>
<td>hard-light</td>
<td>multiply if A&lt;0.5, screen if A&gt;0.5</td>
<td>Image B is lit up by a very bright and sharp light in the shape of image A.</td>
<td><img src="image5.jpg" alt="Illustration" /></td>
<td><img src="image6.jpg" alt="Example Uses" /></td>
</tr>
<tr>
<td>Operation</td>
<td>Algorithm</td>
<td>Description</td>
<td>Illustration</td>
<td>Example Uses</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------</td>
<td>-------------</td>
<td>--------------</td>
<td>--------------</td>
</tr>
<tr>
<td>hypot</td>
<td>sqrt ((A<em>A+B</em>B))</td>
<td>Resembles the plus and screen operations. The result is not as bright as plus, but brighter than screen. Hypot works with values above 1.</td>
<td><img src="image1.png" alt="Illustration" /></td>
<td>This is useful for adding reflections, as an alternative to screen.</td>
</tr>
<tr>
<td>in</td>
<td>Ab</td>
<td>Only shows the areas of image A that overlap with the alpha of B. Also available from <strong>Merge &gt; Merges &gt; In</strong>.</td>
<td><img src="image2.png" alt="Illustration" /></td>
<td>Useful for combining mattes.</td>
</tr>
<tr>
<td>mask</td>
<td>Ba</td>
<td>This is the reverse of the in operation. Only shows the areas of image B that overlap with the alpha of A.</td>
<td><img src="image3.png" alt="Illustration" /></td>
<td></td>
</tr>
<tr>
<td>matte</td>
<td>Aa+B(1-a)</td>
<td>Premultiplied over. Use unpremultiplied images with this operation. Also available from <strong>Merge &gt; Merges &gt; Matte</strong>.</td>
<td><img src="image4.png" alt="Illustration" /></td>
<td></td>
</tr>
<tr>
<td>Operation</td>
<td>Algorithm</td>
<td>Description</td>
<td>Illustration</td>
<td>Example Uses</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------</td>
<td>-------------</td>
<td>--------------</td>
<td>--------------</td>
</tr>
<tr>
<td>max</td>
<td>max (A,B)</td>
<td>Takes the maximum values of both images. Also available from <strong>Merge &gt; Merges &gt; Max</strong>.</td>
<td><img src="max.png" alt="Illustration" /></td>
<td>This is a good way to combine mattes and useful for bringing aspects like bright hair detail through.</td>
</tr>
<tr>
<td>min</td>
<td>min (A,B)</td>
<td>Takes the minimum values of both images. Also available from <strong>Merge &gt; Merges &gt; Min</strong>.</td>
<td><img src="min.png" alt="Illustration" /></td>
<td></td>
</tr>
<tr>
<td>minus</td>
<td>A-B</td>
<td>Image B is subtracted from A.</td>
<td><img src="minus.png" alt="Illustration" /></td>
<td></td>
</tr>
<tr>
<td>Operation</td>
<td>Algorithm</td>
<td>Description</td>
<td>Illustration</td>
<td>Example Uses</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------</td>
<td>-------------</td>
<td>--------------</td>
<td>--------------</td>
</tr>
<tr>
<td>multiply</td>
<td>AB, A if A&lt;0 and B&lt;0</td>
<td>Multiplies the values but stops two negative values from becoming a positive number. Also available from Merge &gt; Merges &gt; Multiply.</td>
<td>Used to composite darker values from A with the image of B - dark gray smoke shot against a white background, for example. This is also useful for adding a grain plate to an image regrained with F_Regrain.</td>
<td></td>
</tr>
<tr>
<td>out</td>
<td>A(1-b)</td>
<td>Only shows the areas of image A that do not overlap with the alpha of B. Also available from Merge &gt; Merges &gt; Out.</td>
<td>Useful for combining mattes.</td>
<td></td>
</tr>
<tr>
<td>over</td>
<td>A+B(1-a)</td>
<td>This is the default operation. Layers image A over B according to the alpha of image A.</td>
<td>This is the most commonly used operation. Used when layering a foreground element over a background plate.</td>
<td></td>
</tr>
<tr>
<td>Operation</td>
<td>Algorithm</td>
<td>Description</td>
<td>Illustration</td>
<td>Example Uses</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------</td>
<td>-------------</td>
<td>--------------</td>
<td>--------------</td>
</tr>
<tr>
<td>overlay</td>
<td>multiply if (B&lt;0.5), screen if (B&gt;0.5)</td>
<td>Image A brightens image B.</td>
<td><img src="overlay.png" alt="overlay" /></td>
<td></td>
</tr>
<tr>
<td>plus</td>
<td>(A+B)</td>
<td>The sum of image A and B. Also available from Merge &gt; Merges &gt; Plus. Note that the plus algorithm may result in pixel values higher than 1.0.</td>
<td><img src="plus.png" alt="plus" /></td>
<td>Useful for compositing laser beams, but you're better off not using this one for combining mattes.</td>
</tr>
<tr>
<td>screen</td>
<td>(A \text{ or } B \leq 1)? (A+B-AB: \text{max } (A,B))</td>
<td>If (A \text{ or } B) is less than or equal to 1 the screen else use the maximum example, resembles Plus. Also available from Merge &gt; Merges &gt; Screen.</td>
<td><img src="screen.png" alt="screen" /></td>
<td>This is useful for combining mattes and also for adding laser beams.</td>
</tr>
<tr>
<td>soft-light</td>
<td>(B(2A+(B(1-AB))) \text{ if } AB&lt;1, 2AB \text{ otherwise})</td>
<td>Image B gets lit up. Not as extreme as the hard-light operation.</td>
<td><img src="soft-light.png" alt="soft-light" /></td>
<td></td>
</tr>
<tr>
<td>Operation</td>
<td>Algorithm</td>
<td>Description</td>
<td>Illustration</td>
<td>Example Uses</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------</td>
<td>-------------</td>
<td>--------------</td>
<td>--------------</td>
</tr>
<tr>
<td>stencil</td>
<td>B(1-a)</td>
<td>This is the reverse of the out operation. Only shows the areas of image B that do not overlap with the alpha of A.</td>
<td><img src="image1.png" alt="Illustration" /></td>
<td><img src="image2.png" alt="Example Uses" /></td>
</tr>
<tr>
<td>under</td>
<td>A(1-b)+B</td>
<td>This is the reverse of the over operation. Layers image B over A according to the matte of image B.</td>
<td><img src="image3.png" alt="Illustration" /></td>
<td></td>
</tr>
<tr>
<td>xor</td>
<td>A(1-b)+B(1-a)</td>
<td>Shows both image A and B where the images do not overlap.</td>
<td><img src="image4.png" alt="Illustration" /></td>
<td></td>
</tr>
</tbody>
</table>
**Tip:** If you have used older versions of Nuke, you may have seen Merge operations called diagonal, nl_over, and nl_under. Diagonal has been renamed and is now called hypot. To get the results of nl_over and nl_under, you can check *Video colorspace* and use over and under.
Generating Contact Sheets

In order to demonstrate, document or manage what you are doing for a project, it can be useful to generate a contact sheet that shows your frame sequence(s) lined up next to each other in a matrix. For this, you can use the ContactSheet node. It generates a contact sheet from all its inputs or from the frames of one input.

A contact sheet generated from the frames of one image sequence.

To Generate a Contact Sheet

1. Select **Merge > ContactSheet** to insert a ContactSheet node in your script.
2. Connect the image(s) you want to include in your contact sheet to the numbered input(s) of the ContactSheet node. If you want to include several different image sequences in the contact sheet, use multiple inputs. If you want the contact sheet to include the frames of just one image sequence, use only one input.

3. Connect a Viewer to the ContactSheet node so you can see the effect of your changes.
4. In the ContactSheet properties, define the **Resolution** (width and height) of the entire contact sheet in pixels.

5. If you want to create a contact sheet from the frames of one input, check **Use frames instead of inputs**. In the **Frame Range** field, define the frame range you want to include in the contact sheet.

6. In the **rows/columns** field, specify into how many rows and columns you want to arrange the input images or frames.

7. To adjust the size of the gaps between the images in the contact sheet, increment or decrement the **gap** value.

![The gap value set to 0.](image)

![The gap value set to 50.](image)

8. From the **Row Order** and **Column Order** dropdown menus, select how you want to order the images or frames in the contact sheet:

- **Row order**: TopBottom
  - **Column order**: LeftRight

- **Row order**: BottomTop
  - **Column Order**: RightLeft

- **Row order**: TopBottom
  - **Column Order**: Snake

- **Row order**: BottomTop
  - **Column Order**: RightLeft
Tip: If you want to add any text, such as the frame number, on top of the images in the contact sheet, insert a Text node between the input image(s) and the ContactSheet node.
Copying a Rectangle from one Image to Another

With the CopyRectangle node, you can copy a rectangle from one input on top of another.

Input A.                         Input B.

The output of CopyRectangle.

The CopyRectangle node can also be used to limit effects, such as color corrections, to a small region of an image. To do so, you need to use the same image in both input A and B and only perform the color correction on one input.

A rectangle from input A color corrected and copied on top of input B.
To Copy a Rectangle from One Image to Another

1. Select **Merge > CopyRectangle** to insert a CopyRectangle node after the image that has a region you want to copy (input A) and the image you want to copy the region to (input B). Create the following setup:

2. In the CopyRectangle controls, use the **channels** dropdown menu to select the channels you want to copy from input A.
3. To define the rectangle you want to copy, resize and reposition the CopyRectangle overlay in the Viewer. Drag the center of the overlay to reposition, and the edges to resize. If you cannot see the overlay in the Viewer, open the CopyRectangle properties panel and double-click on the Viewer node in the Node Graph.

![Repositioning the rectangle.](image1)  ![Resizing the rectangle.](image2)

4. To control how soft the edges of the rectangle seem, adjust the **softness** slider. The higher the value, the softer the edges.

![A low softness value.](image3)  ![A high softness value.](image4)

5. To dissolve between the full CopyRectangle effect and input B, adjust the **mix** slider.
Removing Noise with Denoise

The Denoise node is an efficient tool for removing noise or grain from your footage. It uses spatial or temporal filtering to remove noise without losing image quality.

Quick Start

Here's a quick overview of the workflow:

1. Connect Denoise to the footage from which you want to remove noise.
   See Connecting Denoise.
2. Position the analysis box over a suitable analysis region. Denoise automatically analyzes the noise structure inside this region and removes noise from the footage.
   See Analyzing and Removing Noise.
3. Review the results.
   See Reviewing the Results.
4. If you’re not happy with the results, you can fine tune them by adjusting the noise profile, frequencies, and channels.
   See Fine Tuning.

Tip: You can check Use GPU if available to have the node run on the graphics processing unit (GPU) rather than the central processing unit (CPU).

For more information on the minimum requirements, please see Windows, Mac OS X and
macOS, or Linux or refer to the Nuke Release Notes available in Help > Release Notes.

You can select the GPU to use in the Preferences. Press Shift+S to open the Preferences dialog, make sure you're viewing the Preferences > Performance > Hardware tab, and set default blink device to the device you want to use. You must restart Nuke for the change to take effect.

If you are using a render license of Nuke, you need to add --gpu on the command line.
Connecting Denoise

1. Create a Denoise node by clicking Filter > Denoise.
2. Connect the Denoise node’s Source input to the image you want to denoise.
3. If you’re working with complex footage that doesn’t have a convenient area for analyzing noise (a flat area free from image detail, edges, and luminance variations), you can attach an optional noise clip to the Noise input. When a Noise clip is supplied, the noise is analyzed in this clip, rather than the Source clip. The Noise clip should have similar noise characteristics to the Source clip.
4. You can attach pre-calculated motion vectors to the Motion input. You can generate motion vectors using VectorGenerator, NukeX’s SmartVector, or an external vector generator. See VectorGenerator or SmartVector for more information.
   Denoise can generate motion vectors internally, but connecting this input may produce significantly faster results.
5. Attach a Viewer to either the Source or Noise clip, depending on which you want to use for the analysis.
6. Proceed to Analyzing and Removing Noise.
Analyzing and Removing Noise

1. In the Denoise controls, set Source to Film or Digital depending on the type of footage you’re using. Digital is the default setting and it works fine in most cases even if the footage is in a film format.

2. In general, you can also leave Noise Model set to Modulated. However, you may want to try Constant if you’re working on either:
   • film footage with lots of detail but not too much noise in dark regions, or
   • digital footage with lots of detail but not too much noise in light regions.

3. In the Viewer, scrub to a frame with a suitable analysis region. This should be a flat area free from image detail, so no textures, edges, or shadows. If this is not the case, you may get poor results, as the algorithm thinks the image detail is noise and removes it.

4. You can enable Temporal Processing to use more than one frame to denoise the analysis frame. Denoise blends frames either side of the analysis frame to improve the result. The number of frames used either side of the current frame is determined by the Frames To Blend control.

   Note: Temporal Processing requires the motion, forward, and backward channels from the Motion input. You can't enable Temporal Processing unless the Motion input is connected.

   You can increase the Frame Blending value to force Denoise to blend more regions of the frame, but higher values can lose image detail.

5. Position and resize the analysis box to cover the analysis region. Note that the minimum size for the analysis region is 80x80 pixels. If the analysis region is too small, Denoise cannot analyze the footage or remove any noise.

   A good analysis region. A poor analysis region.

The analysis area selection automatically updates not only the Analysis Region parameter in the Noise Analysis group, but also the frame from which the sample is taken (Analysis Frame).
default, whenever the analysis box is altered, the internal analysis of the noise in that region reoccurs.

6. Connect a Viewer to the Denoise node.
The output should now show the denoised frame.

7. Proceed to Reviewing the Results.

**Tip:** By default, Denoise starts analyzing the noise in your input footage when you move the analysis box. If you scrub to a new frame, don't move the analysis box, and want to reanalyze the noise from the new frame, you can force analysis by clicking the Analyze Noise button in the Noise Analysis parameter group.

If you'd like to disable analysis, you can check Lock Noise Analysis.

**Tip:** You can export the analysis profile in an external file. Note that if you've set Profile to Constant, only the controls that affect the analysis are saved in this file. By contrast, if you've set Profile to Automatic, both the analysis profile and the automatically calculated noise profile are exported.

To export the profile, use the Analysis file control under Noise Analysis to set the name and location of the file. The file extension doesn't really matter here - for example .txt is fine. Once you have named the profile, click the Export button to export it.

To import the same profile later, use Analysis File again to navigate to it and click Import. This disables any controls that are read from the profile. To re-enable them, you can uncheck Lock Noise Analysis.
Reviewing the Results

1. Zoom in to review the results.
2. To compare the denoised image with the original, press D on the Denoise node repeatedly to disable and re-enable it.

![The original image. The denoised image.]

3. It can also be useful to look at the noise that was removed from the original image. To do so, set Output to Noise.
   Only noise should be visible in this image. If you can see a lot of picture detail, it means the current settings are making Denoise work too hard and remove too much of the image, which leads to a soft result.

![If you can see picture detail in the Noise output, too much of the image is being removed.]

4. If your footage contains a lot of areas of sub-black, areas of black with a value less than 0, enable Lift Blacks to lift the blacks towards white. You can also enable Preserve Edges to attempt to sharpen the image at edges preventing over-smoothing. Preserve Edges can emphasize noise in some cases.
5. To blend the denoised luminance with the image’s original luminance, you can temporarily increase Luminance Blend. This brings back some of the image detail in the result. You might want to have this set to 1, for example, when you're working on denoising the footage, but for the final result, you'll want to decrease it. The default value is 0.7.
6. If you’re not happy with the results, you can try:
• moving the analysis box to a different, flat area of the image,
• analyzing on a different frame (by moving to a different frame and clicking Analyze Noise),
• enabling Temporal Processing to blend more than one frame to calculate the denoise effect, or
• tweaking the Denoise controls (Denoise Amount, Roll Off, and Smoothness in particular).
Proceed to Fine Tuning.
Fine Tuning

1. Set **Denoise Amount** to adjust the overall amount of noise to remove. Increase this to remove more noise, or decrease it if you want to keep more detail. A value of 0 removes no noise.

2. If the denoised image looks too sharp, use **Roll Off** to adjust the smoothness of the denoise thresholding. A value of 1 equals hard thresholding. Any other value produces soft thresholding between:
   - the **Denoise Amount** value and
   - the **Roll Off** value multiplied by **Denoise Amount**.

3. If you're not getting the correct smoothness level by adjusting the **Denoise Amount**, try setting **Smoothness** to a new value. This controls the smoothness of the denoised image, affecting the underlying image rather than the noise detected. In most cases, the default value of 1 works fine though.

4. If the results are too smooth and adjusting the above controls didn't help, try setting **Profile** to **Automatic**. Unlike the **Constant** profile, which looks at the analysis region and removes the same amount of noise across all intensities, automatic profiling looks at the entire **Profile Frame** to estimate a noise profile and removes different amounts of noise from the shadow, midtone, and highlight areas of the **Source** image.

   ![Constant profile.](image1)
   ![Automatic profile.](image2)

When you first switch to automatic profiling, Denoise uses the current frame to calculate the profile. If you'd like to use a different frame, you need to scrub to that frame and click the **Recalculate Profile** button on the **Profile** tab. This updates the **Profile Frame** control, which is read only.

---

**Note:** Denoise always bases the noise profile on your **Source** footage even if you've attached another clip to the **Noise** input.
5. You can also tweak the noise profile yourself using the controls on the **Profile** tab. This works in both the **Constant** and **Automatic** profiling mode.

   - Denoise displays the noise profile curve in the curve editor. The x axis represents image intensity, from low frequencies on the left to higher frequencies on the right. The y axis represents the relative amount of noise removed.

   ![A noise profile curve.](image)

   You can adjust the curve manually by dragging the points on the curve to a new location. To add more points to the curve, **Ctrl/Cmd+Alt+click** on the curve.

   If you are not happy with your changes, click **Reset Profile** to reset the curve to its original shape (clicking **reset** works as well).

   - Make sure you check **Tune Profile** to enable your changes. Then, adjust **Low Gain, Mid Gain,** and **High Gain** to scale the denoising threshold in the shadow, midtone, and highlight areas of the image. For example, a value of 2 multiplies the threshold by 2. Everything below the threshold is considered noise and removed, while everything above the threshold is kept.

   ![The noise profile after decreasing Low Gain.](image) ![The noise profile after decreasing High Gain.](image)

6. Back on the **Denoise** tab, the **Tune Frequencies** section lets you enable, disable, and adjust the denoising in different noise frequencies. This allows you to select the frequencies that contain
noise, process them by as much as you think is best, and leave details in the other frequencies untouched.

Normally, most of the noise occurs in the high and medium frequencies, so often you can disable the low and very low frequencies altogether. Use the Gain sliders to remove noise (by increasing the value) or to add more detail and noise (by decreasing the value).

7. Under Tune Channels, you can adjust the denoising threshold for the luma and chroma channels. Increase the Luminance Gain and Chrominance Gain values to remove more noise, or decrease them to remove less.
Keying with ChromaKeyer

This section explains how to use the blue/green screen keyer, ChromaKeyer, in Nuke. ChromaKeyer can take advantage of modern GPUs and multi-core CPUs to accelerate the keying process when used for compositing in Nuke's Node Graph. ChromaKeyer is also available as a soft effect in Nuke Studio's timeline environment.

Quick Key

The images below show a green screen foreground and the background to be composited over.

1. Start Nuke and read in the foreground and background images. From the Keyer menu, select ChromaKeyer and attach a Viewer.
2. In the ChromaKeyer Properties panel, click the color swatch next to screen color to activate the eye dropper.
3. Ctrl/Cmd+Shift+click and drag a rectangular area over the green pixels in the Viewer. This averages the pixels in the area selected to produce a better key.
In some cases, this is all you need to do to perform a key, since selecting the screen color creates a screen matte and despills the foreground.

4. Merge the foreground over the background to produce your final comp.

Picking the screen color may be enough for a lot of keys, but there are many more tools within Nuke that can be used to tackle more complicated shots. See Improving Mattes on page 429, Despilling and Color Replacement on page 432, and Multi-Pass Keying on page 434 for more information.
Picking the Screen Color

The screen color is probably the most important control in keying, and you should always pick the screen color before doing anything else. It should be set to the color of the green or blue behind the foreground object.

1. Start Nuke and read in the foreground and background images. From the Keyer menu, select ChromaKeyer and attach a Viewer.
2. In the ChromaKeyer Properties panel, click the color swatch next to screen color to activate the eye dropper.
3. Pick the screen color directly from the image by Ctrl/Cmd+Shift and dragging a rectangle over the green pixels. The average value of the pixels selected is used.

Tip: You can discard sampled pixels by Ctrl/Cmd+right-clicking in the Viewer.

It's worth picking different shades of green or blue from different parts of the screen to get the best result.

Note: Picking colors repeatedly does not add to previous selections, keying more of the image with each click. To key more of the image, pick different shades of green or blue in additional ChromaKeyer nodes downstream, and set the inside mask control to source alpha. See Multi-Pass Keying for more information.

Picking the screen color does two things:
• It creates the initial matte used to composite the foreground downstream. The matte can be improved using the matte controls in the Properties panel. See Improving Mattes for more information.
• It despills the foreground, but you can use the despill controls for greater accuracy. For more information on despill, see Despilling and Color Replacement for more information.

Adjusting Screen Gain and Balance

The screen gain controls how much of the screen color is removed to make the matte. Increasing this value keys more of the foreground. In the example image, a lower screen gain value adds too much of the background back into the output whereas a high screen gain value erodes the foreground too much and tends to tint the edges the opposite of the screen color (for green screens, edges become magenta).

A screen gain value lower than 1 adds background color back into the image. A screen gain value higher than 1 removes background color from the image.

The screen balance controls the bias toward the two non-primary colors after the screen color has been chosen. A screen balance closer to 0 balances the output toward the stronger of the two colors and a value closer to 1 toward the weaker color. In the example image, the background color is green with RGB values of 0.04, 0.7, and 0.07, so values closer to 0 affect the blue component.

A screen balance closer to 0 affecting A screen balance closer to 1 affecting
the blue component.  the red component.
Improving Mattes

The **matte** controls can improve the basic matte created after picking the **screen color**. The best way to view the matte in ChromaKeyer is to switch the Viewer to display the **alpha** channel in the **R** or **Luminance** display styles.

Adjusting the **Gain**, **White Point**, and **Black Point**

Adjusting the **chroma gain**, **white point**, and **black point** either dilates or erodes the matte to include more or less of the foreground image based on the control used. The example image, concentrating on the area near the character’s chin, shows a good example of where a matte can be improved.
Adjusting the **chroma gain** controls how much of the chroma difference between the source image and the screen color is used to correct the matte. Increasing the gain generates a matte that has less transparent areas in the foreground, but can produce harder edges.

![The matte with a low chroma gain value.](image1.png) ![The matte with a high chroma gain value.](image2.png)

Adjusting the **white** and **black point** controls the pixels that appear as foreground and background. Any pixels with values higher than the **white point** value are treated as foreground (alpha = 1) and any pixels with values lower than the **black point** value are treated as background (alpha = 0).
The chroma gain, white point, and black point controls represent a balancing act between removing too much foreground and retaining too much background. Once you’re happy with the matte, proceed to Despilling and Color Replacement.

For particularly tricky keying problems, you can improve the matte using multi-pass keying. This method uses multiple ChromaKeyer nodes with all but the first one in the node tree set to inside mask > source alpha, so that the mattes in the alpha channels are passed down the node tree. See Multi-Pass Keying for more information.
Despilling and Color Replacement

Although ChromaKeyer automatically despills the matte when you select a screen color, you may find that you can improve the matte by manually despilling the key using a despill bias or by replacing color at the edges of the matte.

Custom Despilling

The despill bias allows you to specify a color from the image, separate from the alpha bias, to improve the overall despill for the matte. Typically, you should pick skin tones or hair colors for the despill bias.

1. Enable custom despill bias to allow you to use the despill bias controls.
2. Click the color swatch next to despill bias to activate the eye dropper.
3. Ctrl/Cmd+Shift+click and drag a rectangular area over the color you want to replace the highlighted pixels in the Viewer. This averages the pixels in the area selected to produce a better result.

![Blue screen pixels present in a default matte. Selecting the hair color despills the blue pixels, producing a better matte.](image)

Color Replacement

Improving a matte by adjusting the alpha channel can remove the wrong amount of screen color from the pixels where transparency has changed. The replace controls instruct ChromaKeyer how to deal with these pixels. The replace mode controls which pixels inherit the replace color.

1. In the ChromaKeyer Properties panel, select the required replace mode using the dropdown:
   - ignore - the despilled image is left untouched if the alpha is modified. This is the default operation.
• **edge hard color** - the despilled image has a corresponding amount of the replace color added for any increase in alpha.

• **edge linear color** - the despilled image has a graded amount of the replace color, controlled by a linear curve, added for any increase in alpha. Pixels closer to the background have more background bias and pixels closer to the foreground have more foreground bias.

• **edge soft color** - the despilled image has a corresponding amount of the replace color added for any increase in alpha, however, it attempts to modulate the luminance of the resulting pixel so that it matches the original pixel. This gives a more subtle result than the edge hard color option.

2. Click the color swatch next to replace color to activate the eye dropper.

3. **Ctrl/Cmd+Shift**+click and drag a rectangular area over the color you want to replace the highlighted pixels in the Viewer. This averages the pixels in the selected area to produce a better result.

**Note:** You can enhance the despill effect by enabling **add-in matte fix** to also apply the replace color to areas eroded or dilated by the white point and black point controls.

4. Use the replace amount slider to control how much of the replace color is applied.

5. Disable the premultiply control if you don't want the output premultiplied by the alpha channel.
Multi-Pass Keying

In production situations, you may not always get a uniform color across your blue or green screen background, which can make pulling a key difficult. ChromaKeyer allows you to key different regions of the screen additively by picking shades of blue or green in multiple ChromaKeyer nodes. The example image shows a poor background containing different shades of blue as a screen color.

![Example image of poor background containing different shades of blue as a screen color.]

To perform a multi-pass key, do the following:

1. Pick an initial screen color as described in Picking the Screen Color on page 426.

   ![Initial screen color picked.]

   You can see from the image that ChromaKeyer has done a decent job on the area chosen, but the lower-half of the image still contains some noise.

2. Add a second ChromaKeyer node and in the Properties panel, set the inside mask control to source alpha.

   This takes into account the screen matte from the initial ChromaKeyer node when you pull another key.

3. Pick another screen color in the problem area using the second ChromaKeyer node.

   ![Second screen color picked.]

   The key is improved by adding the second screen matte to the results of the first.

4. You can add as many ChromaKeyer nodes as you like, as long as you remember to set the inside mask control to source alpha in all but the first ChromaKeyer node.
Keying with Keylight

This section explains how to use the blue/green screen keyer, Keylight, in Nuke.

Quick Key

Consider this shot from The Saint, pictures courtesy of CFC and Paramount British Pictures Ltd.

The figure above is the blue screen foreground that should be composited over the background shown below.

1. Start Nuke and read in both images. From the Keyer menu, apply Keylight and attach a Viewer.
2. Click the color swatch next to ScreenColor to activate the eye dropper. In the Viewer, Ctrl/Cmd+Shift+Alt+click and drag a rectangular area over the blue pixels as shown below.
Picking the screen color also sets the **ScreenBalance**.

That's it. In many cases, this is all you need to do to perform a key, since selecting the screen color creates a screen matte and despills the foreground.

3. Switch output from **FinalResult** to **Composite** to see the foreground keyed over the background. The final composite is shown below.

Picking the screen color may be enough for a lot of keys, but there are many more tools within Nuke that can be used to tackle more complicated shots. These are described later in this chapter.
Basic Keying

The following section describes the parameters you need to do basic keying. This gives you enough to tackle most simple keys. A discussion of advanced parameters to fine tune keys and tackle complex keys can be found under Advanced Keying.

Picking the Screen Color

The screen color is probably the most important parameter and you should always pick the screen color before doing anything else. It should be set to the color of the green or blue curtain behind the foreground object. Pick the screen color directly from the image by Ctrl/Cmd+Shift+Alt dragging a rectangle over the blue pixels. The average value of the pixels selected is used.

Tip: If you press Alt when sampling a color, Nuke always samples the source image regardless of what you’re looking at. This means that you can pick the blue screen color even if you are viewing the matte, status, or composite.

Tip: You can discard sampled pixels by Ctrl/Cmd+right-clicking in the Viewer.

Tip: Picking different shades of blue from the screen color can give quite different results. It’s worth picking from different parts of the screen to get the best result.

Picking the ScreenColor creates the screen matte used to composite the foreground over the background. It also sets the ScreenBalance (if this has not already been set manually) and despills the foreground.

Screen Matte

Setting the screen color pulls a key, or in other words, creates a matte - the ScreenMatte. Setting the screen color also despills the foreground, although you can also use the DespillBias to remove more spill. In some cases this is enough to get a decent key. For more information on ScreenColor, see Screen Color.

The image below shows a well-lit blue screen behind an actor.
You should note that repeatedly picking colors does not add to previous selections and key more of the image with each click. To key more of the image, try picking different shades of blue then use the screen strength parameter. See *Keying More*.

## Viewing the Key

After picking the screen color, you have created a matte (the screen matte) and despilled the foreground. The result can be displayed in a number of different ways using the **View** control. You can output the final composite of the foreground over the background as an rgba, or you can output the premultiplied or unpremultiplied foreground for compositing elsewhere in the tree. The screen matte and the status view are the other two options which are useful in fine-tuning the key rather than as an output image in their own right.

The **Status** is one of the options in the **View** dropdown menu and shows an exaggerated view of the key so that you can make a more informed decision when tuning the key. The image on the right shows the **Status** display after the screen color has been picked from the image on the left.

Three colors are displayed:
- Black pixels show areas that are pure background in the final composite.
- White pixels show areas that are pure foreground.
• Gray pixels are a blend of foreground and background pixels in the final composite. You need gray pixels around the edge of the foreground to get a good key at the foreground edge. Pixels that are a blend between the foreground and background are shown in just one shade of gray. This is done to highlight potential problems with the key. These gray pixels may represent a foreground/background blend of 50/50 or 99/1. No distinction is made as to this ratio.

You may occasionally see other colors in the Status view and these are covered under View.

Keying More

To improve the key by firming up the foreground so the background doesn't show through, you should adjust the ClipWhite parameter. To key more of the foreground so that the background is clearer, you should use the ClipBlack parameter. Look at the ScreenMatte and the Composite while you're doing this. Don't overdo either of these or the edges between foreground and background become hard.
Advanced Keying

The following section describes how Keylight works under the hood as well as the parameters you need to fine tune keys and get the most out of Keylight. Basic parameters covered previously may also be covered here in more detail.

Under the Hood

Keylight is a ‘color difference keyer’, which means that for it to figure out a key, it compares every pixel in the image against a single color, known here as the ScreenColor.

View

The View parameter allows Keylight to render the final composite of the foreground over the background, or the foreground RGBA for compositing further down the tree. Two options, ScreenMatte and Status, are for viewing the key rather than an output. The options are:

- **Source** - shows the blue/green screen foreground.
- **SourceAlpha** - shows the alpha channel on the foreground input.
- **ScreenMatte** - this is the matte created from picking the ScreenColor. It does not include any inside or outside masks.
- **InsideMask** - shows the inside input. This is used to firm up the foreground matte to stop print through.
- **OutsideMask** - shows the outside input. The outside mask is used as a garbage mask to reveal the background.
- **CombinedMatte** - the screen matte, inside mask, and outside masks added together.
- **Status** - this renders an exaggerated view of the key so that minor problems are shown clearly.
- **IntermediateResult** - use this option on shots that can only be keyed using several different keys on different parts of the image (multi-pass keying). This renders the original source image with the ScreenMatte generated in this Keylight node. In Keylight nodes down the tree, you should set the SourceAlpha in the InsideMask folder to AddToInsideMask.
- **FinalResult** - this creates a premultiplied RGBA foreground that can be composited later. There’s an UnpremultiplyResult toggle you can use if you wish.
- **Composite** - this renders the foreground composited over the background using all mattes, spill and color corrections.
**Status**

*Status* is one of the options in the View dropdown menu and shows an exaggerated view of the key so that you can make a more informed decision when fine tuning the composite. The figure on the right shows the Status after the screen color has been picked from the image shown in the figure on the left.

Three colors are displayed:
- Black pixels represent pure background in the final composite.
- White pixels are pure foreground.
- Gray pixels are a blend of the foreground and background pixels. The gray is just one color to highlight any areas that are not pure foreground or background. Gray pixels do not mean the key is poor - the final composite may be fine.

You may occasionally see other colors in the **Status** view. The figure on the left shows black, white, gray, and green pixels.
• Green pixels are a warning. They show you the parts of the alpha that have changed through processing the alpha channel (clipped, softened, or eroded). These areas have had the correct amount of spill removed, but the alpha has subsequently changed and the composite may no longer look right. This can be corrected using the `ScreenReplaceColor` to put back color in these areas. Above, the figure on the right is an extreme example to illustrate the point. The `ScreenReplaceColor` has been set to pure red and you can see that this mirrors the green pixels in the Status view.

Similarly, you may see blue pixels in the Status.

![Status showing how the inside matte affects the foreground.](image1)

![Composite showing the inside replace color.](image2)

• Blue pixels represent processed pixels in the `InsideMask` that affect the despill of the foreground. The `InsideReplaceColor` is used to modify these pixels. Another extreme example is shown above in the figure on the right. The `InsideReplaceColor` is set to pure yellow and the `InsideReplace` is `HardColor`.

• You may also see dark red pixels in the Status. Red pixels indicate areas where an outside mask has been used to reduce the transparency of the image.
View

The View parameter allows Keylight to render the final composite of the foreground over the background, or the foreground RGBA for compositing further down the tree. Two options, Screen Matte and Status, are for viewing the key rather than an output. The options are:

- **Source** - shows the blue/green screen foreground.
- **Source Alpha** - shows the alpha channel on the foreground input.
- **Screen Matte** - this is the matte created from picking the Screen Color. It does not include any inside or outside masks.
- **Inside Mask** - shows the inside input. This is used to firm up the foreground matte to stop print through.
- **Outside Mask** - shows the outside input. The outside mask is used as a garbage mask to reveal the background.
- **Combined Matte** - the screen matte, inside mask, and outside masks added together.
- **Status** - this renders an exaggerated view of the key so that minor problems are shown clearly.
- **Intermediate Result** - use this option on shots that can only be keyed using several different keys on different parts of the image (multi-pass keying). This renders the original source image with the Screen Matte generated in this Keylight node. In Keylight nodes down the tree, you should set the Source Alpha in the Inside Mask folder to Add To Inside Mask.
- **Final Result** - this creates a premultiplied RGBA foreground that can be composited later. There’s an Unpremultiply Result toggle you can use if you wish.
- **Composite** - this renders the foreground composited over the background using all mattes, spill and color corrections.

Status

Status is one of the options in the View dropdown menu and shows an exaggerated view of the key so that you can make a more informed decision when fine tuning the composite. The figure on the right shows the Status after the screen color has been picked from the image shown in the figure on the left.
Three colors are displayed:

• Black pixels represent pure background in the final composite.
• White pixels are pure foreground.
• Gray pixels are a blend of the foreground and background pixels. The gray is just one color to highlight any areas that are not pure foreground or background. Gray pixels do not mean the key is poor - the final composite may be fine.

You may occasionally see other colors in the Status view. The figure on the left shows black, white, gray, and green pixels.

• Green pixels are a warning. They show you the parts of the alpha that have changed through processing the alpha channel (clipped, softened, or eroded). These areas have had the correct amount of spill removed, but the alpha has subsequently changed and the composite may no longer look right. This can be corrected using the Screen Replace Color to put back color in these areas. Above, the figure on the right is an extreme example to illustrate the point. The Screen Replace Color has been set to pure red and you can see that this mirrors the green pixels in the Status view.
Similarly, you may see blue pixels in the Status.

- Blue pixels represent processed pixels in the Inside Mask that affect the despill of the foreground. The Inside Replace Color is used to modify these pixels. Another extreme example is shown above in the figure on the right. The Inside Replace Color is set to pure yellow and the Inside Replace is Hard Color.
- You may also see dark red pixels in the Status. Red pixels indicate areas where an outside mask has been used to reduce the transparency of the image.
Screen Color

The screen color represents the color of the pure blue (or green) screen. The first thing you should do when pulling a key is pick the Screen Color.

**Note:** If you press Alt when sampling a color, Nuke always samples the source image regardless of what you’re looking at. This means that you can pick the blue screen color even if you are viewing the matte, status or composite.

**Tip:** You can discard sampled pixels by Ctrl/Cmd+right-clicking in the Viewer.

Picking the Screen Color creates the screen matte used to composite the foreground over the background. It also sets the Screen Balance and despills the foreground.

The Screen Color is a single color. It has a primary component, blue or green, and that has a saturation. Once the screen color has been picked, Keylight analyzes all the pixels in the image and compares the saturation of the primary component in each of these pixels with the corresponding saturation of the screen color. Keylight uses this comparison to do two things.

1. It calculates the transparency of that pixel and puts it in the alpha channel.
2. It removes the screen color from the pixel, a process known as despilling.

**Tip:** It's worth sampling a selection of screen (blue or green) colors and viewing the result. Picking different colors gives different results.

Background Pixel

If the saturation of the pixel in the image is as strong or greater than the screen color, then it'll be a pixel from the blue screen background, and that pixel is set to completely transparent and black. See the figure below.
Blue screen pixel set alpha to zero.

**Edge Pixel**

If the saturation of the pixel is less than the screen color, then it’ll be the edge of the foreground object, and we subtract some of the screen color from the pixel (despilling) and set the image to semi-opaque. See the figure below.

Edge pixel gives partial alpha.

**Foreground pixel**

If the primary component in the pixel is not the same as the primary component of the screen color, we have a foreground pixel, and the alpha is set to completely opaque. The pixel color is not modified. See the figure below.

Foreground pixel gives full alpha.
Note: You should note that the **Screen Color** is a single color. You are not picking lots of colors that are keyed out.

**Biasing**

What’s biasing all about? Biasing in Keylight was originally developed for a shot in the motion picture "Executive Decision". The foreground consisted of reddish browns, but a combination of factors led to the 'green screen' being lit so that its primary component was actually slightly red.

So what happens when we pick the screen color? Well because the screen was 'red', as is the foreground, our pilot ends up being keyed out as shown below.

Not a great result, I'm sure you'll agree, and much pressure was applied to the lowly programmers to get around the problem.

A work around to this is to manually color correct the image so that the background is properly green, pull the key from this corrected image, then 'un-correct' the result of that so that the foreground colors match the original. A corrected image would look something like the one shown below. The green screen is now strongly green and distinct from the foreground colors. Notice also the red cast on the pilots mask has been removed and turned into a neutral gray.
This is effectively how the Keylight developers got around the problem. They introduced the concept of a 'bias' color, which is a color cast that is removed from the source image and screen color, then a key is pulled from this modified image, then the color cast is put back. In essence, this automates the work around described above, however, it is done in a way that does not slow Keylight down at all.

For our Executive Decision shot, an appropriate color is the red cast on the pilot's mask in the source footage. Setting our bias to this now gives us the far better result as shown below.

The Bias Colors in Everyday Use

It also turns out that the bias color is actually useful for situations without strong casts, typically where there is some color spill around the edge of keys. By setting the biases to the main color that occurs near the edge of the foreground (typically flesh tones or hair tones), you allow Keylight to better discriminate between foreground and background.

Picking a bias color

To pick a bias color, click the color swatch next to Alpha Bias to activate an eye dropper and Ctrl/Cmd+Shift+Alt+drag a box over the image foreground. The average color under the box is used for the bias you have selected.

Note: If you press Alt when sampling a color, Nuke always samples the source image regardless of what you’re looking at. For instance, you may be looking at the blue screen keyed over the background but you are picking colors from the Source image.

Tip: You can discard sampled pixels by Ctrl/Cmd+right-clicking in the Viewer.
Why are there two bias colors?

Remember that Keylight does two things, calculates a transparency and removes the screen color from the foreground. By default, one bias color, the Alpha Bias, is used for both operations. This works fine in most situations, for example, the Executive Decision shot above.

However, sometimes you can pick a bias that gives a great alpha, but performs a poor despill, and another bias that gives a great despill, but a poor alpha. Consider the blue screen from the TV series Merlin, courtesy of CFC Framestore shown below in the figure on the left.

We pick the strong blue of the background without selecting an alpha bias, and end up with the lovely alpha shown on the right, but the despill resulting from this key is poor as shown below.

![Merlin blue screen. Nice Alpha.](image)

**Tip:** There are several nodes in Nuke you can use for spill removal. For example, if you are using a greenscreen image, you can add an Expression node after your foreground image and set the expression field for the green channel to:

\[ g > \frac{(r+b)}{2}?\frac{(r+b)}{2}:g \]

Similarly, you can despill a bluescreen image by setting the expression field for the blue channel to:

\[ b > \frac{(r+g)}{2}?\frac{(r+g)}{2}:b \]
You can also use the HueCorrect node for despill. For more information, see Correcting Hue Only.

We can pick an alpha bias to get a better despill, but this destroys our nice alpha. The way around this is to turn off the **Use Alpha Bias for Despill**, which gives you a separate bias factor to use solely for despill calculations. If you then pick the **Despill Bias** to be something from Miranda Richardson's hair or skin tone, you keep the nice alpha, and get a good despill as well (see the figure on the right).

![Poor despill. Final key, using separate Despill and Alpha Biases.](image)
Clip Black and White

The clip levels are adjusted using two parameters - **Clip Black** and **Clip White**. Any alpha value at or below **Clip Black** is set to zero and any alpha value at or above **Clip White** is set to 1. The figure on the left shows the original alpha of an image, and the figure on the right shows the result of clipping it.

Notice how the gray areas in the black background have been reduced and that the gray edges have hardened up considerably. When compositing, the **Clip Black** control can be used to improve the background image if parts of the foreground are showing through. The **Clip White** control, on the other hand, can be used to firm up the center of the matte, making it less transparent to the background.

**Note:** If you choose to use **ClipBlack** and **ClipWhite**, you need to be really careful that you don't destroy the edges on your foreground. It is possible to use **ClipRollback** to compensate for this.
Screen Gain

The screen gain controls how much of the screen color is removed to make the screen matte. Increasing this value keys more. The figure on the left shows the Status after picking the Screen Color.

![Status after picking the Screen Color.](image)

You can clearly see that parts of the background are gray where they should be black. When composited, you may see faint pixels from the foreground where you should be seeing pure background. Increasing the screen gain fixes this, as shown in the figure on the right (above), but increasing it too much destroys your good work. Like many keying parameters it's a balance - not too much, not too little. Increasing the screen gain too much leads to the background showing through the foreground and edge detail can be destroyed. Below, the figure on the right shows this quite well.

![Status showing the increase in Screen Gain.](image)

**Screen Gain** = 1.05 giving a good Screen Matte.

**Screen Gain** = 1.50 giving background show-through and over eroded edges.

Note the steering wheel is black when it should be white. If you look at the composite, you can see the background showing through here. Also, some of the fine hair detail on the actor, visible in the figure on the left, has been eroded in the figure on the right.
Screen Balance

The **ScreenBalance** is set automatically after picking the **ScreenColor**.

Saturation is measured by comparing the intensity of the primary component against a weighted average of the two other components. This is where the Screen Balance control comes in. A balance of 1 means that the saturation is measured against the smallest of the other two components in the screen color.

A balance of 0 means that the saturation is measured against the larger of the other two components. A balance of 0.5 measures the saturation from the average of the other two components.

The appropriate balance point for each image sequence you key is different depending on the colors in that image. Generally speaking, blue screens tend to work best with a balance of around 0.95 and green screens with a balance of around 0.5. These values are selected automatically the first time you pick the screen color. If the key is not working too well with these settings, try setting the balance to about 0.05, 0.5 and 0.95 and see what works best.
PreBlur and Tuning

Some shots can be improved by softening the foreground image that is used to generate the key. The original image is then used in the composite and color corrections. The Screen PreBlur parameter is used to do this. DV footage or grainy shots may benefit from subtle use of this control.

Tuning

Keylight creates the screen matte after the screen color has been picked. You can make fine adjustments to this matte using the Gain controls. Increasing the gain controls makes the screen matte more transparent by increasing the amount of screen color showing through the matte. This tends to tint the edges the opposite of the screen color (for blue screens, edges become yellow). Decreasing the gain makes the main matte more opaque by reducing the amount of screen color showing through the matte.

The matte can be adjusted independently in the shadows, midtones, and highlights, giving more control than the clipping levels.

The level of the midtones can be adjusted too. For example, if you are working on a dark shot you may want to set the midtone level to a dark gray to make the gain controls differentiate between tones that would otherwise all be considered shadows.
Screen Processing

Once you have picked the screen color and got the screen matte, you may wish to process this matte using the parameters in the Screen Matte group. The matte can be adjusted using clipping levels; it can be eroded or grown, despotted, and softened.

Two-stage keying

Consider this example. Having applied Keylight and picked the screen color, you have good edges to your matte but the background is showing through the foreground. You could fix this by tweaking the Clip White, but in doing so it ruins your edges. One way round this is a two stage key (another way is using Clip Rollback). In the first key, you process the screen matte using the clipping levels to give a harsh black and white matte, then soften and erode it. Switch View to Intermediate Result to output the original green screen with the eroded matte as an RGBA. Then, use this as the input to another Keylight node. In this second node, pick the screen color to give good edges but with transparent foreground. Don't process this matte, instead use the input alpha channel to fix the transparent foreground. Just set Source Alpha in the Inside Mask folder to Add To Inside Mask.

Clip Rollback

Pulling a screen matte (the figure on the left) typically produces lots of transparency (gray) in the matte at the edges. This is good since this is what you need to key hair well. You may also get transparency in the foreground as shown in the figure on the right. This is bad as your subject appear slightly see-through, and this should be corrected.
You can do this by connecting a matte into the third (InM) input, or you can use the **Clip White** parameter to turn these gray pixels white. This cleans up the foreground (the figure on the left) but it also destroys the edge detail you want to keep. This is where **Clip Rollback** comes in. This is used to put back the edges to restore the detail that was lost. A rather exaggerated clip rollback is shown in the figure on the right to illustrate the point.

![Image](image_url)

**Clip White** has been used to remove the unwanted gray pixels in the white matte. **Clip Rollback** has been used to restore the unwanted erosion of the edge.

### Dilate

This control should not normally be used as eroding the edges can produce a very poor key. However, the **Screen Dilate** parameter allows you to grow (if greater than zero) or shrink (if less than zero) the alpha in the **Screen Matte**. These controls are sub-pixel accurate.

![Image](image_url)

**Screen Matte.** **Eroded Matte.**

### Softness

Occasionally, it is useful to be able to blur the matte. Use **Screen Softness** for this. The most common example would be to pull a very harsh matte that you would use as an inside matte further down the tree. For this, you’d soften and erode the screen matte.

![Image](image_url)
Despot

This controls how much to simplify the matte. It coagulates similar regions so that, for example, black specks in the white matte can be absorbed by the surrounding white areas. Increasing the Screen Despot Black removes isolated spots of black in the white matte. Increasing Screen Despot White removes isolated spots of white in the background up to that size.
Mattes

There are 4 mattes in Keylight.

1. Screen Matte
2. Inside Mask
3. Outside Mask
4. Alpha (Composite Alpha)

The **Screen Matte** is generated by the Keylight algorithm after the screen color has been picked. It can be processed (clipped, eroded, etc) by the screen matte processing tools.

The **Inside Mask** is the hold out matte. It is used to confirm areas that are definitely foreground. If your subject has blue eyes and is being shot in front of a blue screen, this mask can be used to put back the eyes. This mask is taken from the **InM** input to Keylight. The embedded alpha channel of the foreground input can be added to this mask using the **Source Alpha** parameter in the **Inside Mask** folder.

The **Outside Mask** is the garbage mask and is used to remove unwanted objects (lighting rigs, etc) from the foreground. The mask is taken from the **OutM** input to Keylight. The luminance or the alpha of this input is set using the **OutM Component** parameter.

The matte used to blend the foreground and background in the final composite is the alpha displayed in the alpha channel of the composite. This matte is the combination of the screen matte, inside matte, and outside matte.
Inside and Outside Masks

If you can’t adequately improve the screen matte using the clip levels, you can create a matte in Nuke round the pixels you definitely want to be foreground or background and use this as a mask input. The inside mask makes the foreground less transparent and the outside mask is used to clean up the background that might have bits of the foreground showing through. It is sometimes referred to as the hold out mask.

The outside mask (garbage mask) is often used to clean up screens that are not a constant color or have lighting rigs in shot by forcing the alpha transparent.

The inside mask can be used to keep elements in the foreground that you don't want to lose (an actor’s blue eyes in front of a blue screen). These masks should normally be softened externally to blend into the screen matte.

The below image shows the Bezier spline drawn around the lighting rig on the left side of the screen.
Connect the mask to the OutM input of Keylight and switch the parameter OutM Component to Alpha. The outside mask forces that part of the image to be in the background, thus keying out the rig.
Source Alpha

This parameter determines what to do with any embedded alpha in the original source image. You need this if you are doing multiple keys on different parts of the image with the View output set to Intermediate Result.

- Ignore - this does not add any embedded alpha to the screen matte.
- Add To Inside Mask - the embedded alpha is added to the inside mask. You should select this when multipass keying with Output View set to Intermediate Result.
- Normal - the embedded alpha is used to composite the image.
Color Replacement

Remember that Keylight does two things - it removes the screen color to despill the image and generates an alpha (Screen Matte) to composite the foreground over the background layer.

If you then process the Screen Matte, for example, by eroding the alpha or changing the clip levels, Keylight would be removing the wrong amount of screen color from the pixels whose transparency has now changed. The Screen Replace instructs Keylight how to deal with such pixels. The Status displays which pixels use a replace method. Those pixels that use a replace method because the alpha processing tools modified the transparency are green, whilst those pixels whose transparency was modified by the inside matte are blue. See the Status View under View.

There are four options to the replace method. These are:

1. **None** - the despilled image is left untouched if the alpha is modified.
2. **Source** - the image has a corresponding amount of the original pixel (screen color and all) reintroduced/removed if the alpha is changed.
3. **Hard Color** - the despilled image has a corresponding amount of the Screen Replace Color added for any increase in alpha.
4. **Soft Color** - the despilled image has a corresponding amount of the Screen Replace Color added for any increase in alpha, however, it attempts to modulate the luminance of the resulting pixel so that it matches the original pixel. This gives a more subtle result than the Hard Color option.

**Inside mask**

If the changes to the screen matte are due to an inside mask, the Inside Replace and Inside Replace Color parameters can be used to modify the color in these areas just like the Screen Replace parameters described above.

**Edges**

Built-in crop tools are included to quickly remove parts of the foreground at the edges of the image. It can also be useful in tidying up a matte at the edges where luminance changes in the blue screen are proving difficult to key out.

With X Method and Y Method set to Color and Edge Color, set to pure blue (for a blue screen), set the Left to crop out the left-hand side of the image revealing the background. The figures below show the changes to the Combined Matte with cropping.
InM component

The component (luminance or alpha channel) of the inside mask input that is used in the calculations. Typically, this is a garbage matte that covers the area you know to be 100% foreground.

Tip: To avoid having to roto the inside matte throughout the clip, you can connect another Keylight node to the InM input and use it to create a hard, dilated key (set Screen Dilate to a low value).

OutM component

The component (luminance or alpha channel) of the outside mask. Typically, this is a garbage matte that covers the area you know to be 100% background.
A bluescreen image.  

An outside matte.
Keying with Primatte

This section explains how to use the blue/green screen keyer, Primatte, in Nuke.

Quick Start

1. Connect the Primatte node to your background and foreground images. See Connecting the Primatte Node.

2. Click on the **Auto-Compute** button. Primatte attempts to automatically sample the backing screen color and perform the cleanup phases of keying with Primatte.
   
   If you get good results, jump ahead to spill removal. See Spill Removal - Method #1.
   
   If you don't get the results you wanted from **Auto-Compute**, continue to step 3.

3. Click the **operation** dropdown, select **Smart Select BG Color**, and manually sample the targeted background color by holding **Ctrl/Cmd** and clicking in the Viewer.

   **Tip:** You can discard sampled pixels by **Ctrl/Cmd**+right-clicking in the Viewer.

   See Smart Select BG Color for more information.

4. Click the **operation** dropdown, select **Clean BG Noise**, and clean up any remaining white regions in the dark, bluescreen area by sampling in the Viewer.

   See Clean BG Noise for more information.

5. Click the **operation** dropdown, select **Clean FG Noise**, and clean up any dark regions in the foreground by sampling in the Viewer.

   See Clean FG Noise for more information.

Connecting the Primatte Node

1. Start up Nuke and create a Primatte node (**Keyer** > **Primatte**).
2. Connect a foreground image to the Primatte node’s **fg** input and a background image to the **bg** input.
3. Add a Viewer node so you can see the result.

4. When you select the Primatte node, the Primatte properties panel displays.
Connecting the Primatte Node

[Image of NUKE interface showing Primatte node settings]

- **Foreground**: rgb
- **Background**: rgb
- **Crop**: x 0, y 0, r 2048, t 1556
- **Mask**: none
- **Algorithm**: Primatte
- **De-grain Type**: none
- **De-grain Tolerance**: 0.2
- **Actions**: Smart Select BG Color
- **Adjust Lighting**: On
- **Hybrid Matte**: On
- **Fine Tuning**: On
- **Spill Process**: Complement
- **Replace Color**: 0.18
- **Defocus**: 10
- **Output Mode**: Composite
- **Output**: rgba, red, green, blue, rgba.alpha
Primatte Basic Operation Tutorial

This describes the operation of the Primatte node in Nuke. A more detailed explanation of how the Primatte algorithm actually works can be found under The Primatte Algorithm.

Auto-Compute

Primatte has a feature that attempts to eliminate the first three steps of the more standard keying procedure. The **Auto-Compute** button is a good starting point and it may make your keying operation much easier.

1. Click on the **Auto-Compute** button. Primatte attempts to automatically sense the backing screen color, eliminate it, and even get rid of some of the foreground and background noise that would normally be cleaned up in the **Clean BG Noise** and **Clean FG Noise** phases of keying with Primatte.

2. If you get good results then jump ahead to the spill removal tools. See [Spill Removal - Method #1](#).

3. If you don’t get the results you wanted from **Auto-Compute**, please continue from this point on to get the basic Primatte operation procedures. See [Smart Select BG Color](#)

The basic functionality for the Primatte interface is centered around the **operation** dropdown menu and the Viewer window.
There are four main steps to using the Primatte and **Select BG Color** is the first step.
Smart Select BG Color

Ensure that the Smart Select BG Color action is selected (it should be at this time as it is the default operation mode).

Position the cursor in the bluescreen area (or whatever background color you are using), usually somewhere near the foreground object. Hold the Ctrl/Cmd key down and sample the targeted background color. Release the mouse button and Primatte starts the compositing process. If the foreground shot was done under ideal shooting conditions, Primatte has done 90-95% of the keying in this one step and your image might look like this.

However, if you have a very unevenly lit backing screen, you may not be getting the results you’re after. If this is the case, enable adjust lighting on the Primatte properties. For more information, see Actions Section.

**Note:** Primatte works equally well with any color backing screen. It does not have to be a specific shade of green or blue.

**Tip:** If you dragged the cursor in the blue area, Primatte averages the multi-pixel sample to get a single color to adjust to. Sometimes Primatte works best when only a single pixel is sampled instead of a range of pixels. The color selected at this point in the Primatte operation is critical to the operation of the node from this point forward. Should you have difficulties further along in the tutorial after selecting a range of blue shades, try the Smart Select BG Color operation again with a single dark blue pixel or single light blue pixel. You
can also switch to the alpha channel view and click around in the bluescreen area and see the different results you get when the initial sample is made in different areas.

**Tip:** If you would rather make a rectangular selection and not use the default 'snail trail' sampling method, you can do a Ctrl+Shift+drag sample.

**Tip:** You can discard sampled pixels by Ctrl/Cmd+right-clicking in the Viewer.

**Tip:** If the foreground image has a shadow in it that you want to keep it in the composite, do not select any of the dark blue pixels in the shadow and the shadow comes along with the rest of the foreground image.

The second and third steps in using Primatte require viewing the matte or alpha view in the Viewer window. Press the A key on the keyboard to change to the alpha view. The image displayed changes to a black and white matte view of the image that looks like this.
Clean BG Noise

Set operation to Clean BG Noise. If there are any white regions in the dark, bluescreen area, it is noise (or shades of blue that did not get picked up on the first sample) and should be removed. Sample through these white noise regions and release the pen or mouse button to process the data and eliminate the noise. Repeat this procedure as often as necessary to clear all the noise from the background areas. Sometimes increasing the brightness of your monitor or the screen gamma allows you to see noise that would otherwise be invisible.

Note: 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 become 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 bluescreen portions of the image. In particular, if an attempt is made to meticulously remove noise around the foreground object, 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 areas near the edge of the hair. Leave a little noise around the hair as this can be cleaned up later using the FineTuning sliders.

Before background noise removal.  After background noise removal.
Clean FG Noise

If there are dark regions in the middle of the mostly white foreground object, that is, if the key is not 100% in some portion of the targeted foreground, select **Clean FG Noise** from the operation dropdown menu. Use the same techniques as for **Clean BG Noise**, but this time sample the dark pixels in the foreground area until that area is as white as possible.

Before foreground noise removal.  
After foreground noise removal.

**Tip:** After sampling the backing screen color and finding that the edges of the foreground object look very good, you sometimes find that an area of the foreground object is transparent. This is due to the foreground containing a color that is close to the backing screen color. When this transparency is removed using the **Clean FG Noise** operation, the edge of the foreground object picks up a fringe that is close to the backing screen color and it is very hard to remove without sacrificing quality somewhere else on the image. To tackle this problem, you can enable **hybrid render**. For more information, see **Actions Section**.

These were the steps necessary to create a clean matte or key view of the image. With this key, the foreground can be composited onto any background image. However, if there is spill on the foreground object from light that was reflected off the background, a final operation is necessary to remove that background spill get a more natural looking composite.

For the fourth step in the Primatte operation, return the **RGB** view to the Viewer window by clicking again on the **A** keyboard key. This turns off the alpha channel viewing mode; the Viewer window again displays the **RGB** view with the background image (if you connected one to the Primatte node).

The sample image below has gone through the first three steps and has examples of spill. Notice the blue fringe to her hair and a blue tint on her right cheek, arm and chest.
Blue spill visible.
Spill Removal - Method #1

There are three ways in Primatte to remove the spill color. The quickest method is to select Spill Sponge from the operation dropdown menu and then sample the spill areas away. By just positioning the cursor over a bluish pixel and sampling it, the blue disappears from the selected color region and is replaced by a more natural color. Additional spill removal should be done using the Fine Tuning tools or by using the Spill(-) operation. Both are explained further on in this chapter.

**Note:** All spill removal/replacement operations in Primatte can be modified using the Spill Process replace with tools. Spill can be replaced with either the complement of the background color, a solid color you’ve selected, or by colors brought from a defocused background. Depending on the spill conditions, one of these options should provide the results you are looking for. See the information in Replacing Spill for more details.

**Note:** Primatte's spill removal tools work on color regions. In the image in Clean FG Noise, samples should be made on the light flesh tones, the dark flesh tones, the light blonde hair, the dark blonde hair, and the red blouse color regions. One sample in each color region removes spill from all similar colors in the foreground image.

If the spilled color has not been totally removed using the Spill Sponge or the result of the Spill Sponge resulted in artifacts or false coloring, the Spill(-) tool should be used instead for a more subtle and sophisticated removal of the spilled background color. This is discussed in Actions Section.

**Tip:** There are several nodes in Nuke you can use for spill removal. For example, if you are using a greenscreen image, you can add an Expression node after your foreground image and set the expression field for the green channel to:

\[ g>(r+b)/2?(r+b)/2:g \]

Similarly, you can despill a bluescreen image by setting the expression field for the blue channel to:

\[ b>(r+g)/2?(r+g)/2:b \]

You can also use the HueCorrect node for despill. For more information, see Correcting Hue Only.
Spill Removal - Method #2

1. Select the **Fine Tuning Sliders** in the **operation** dropdown menu. This activates the **Fine Tuning** sliders.

2. In the Viewer, zoom into an area that has some blue edges or spill.

3. Using the cursor, sample a color region that has some spill in it. When you let up on the pen or mouse button, Primatte registers the color selected (or an average of multiple pixels) in the current color swatch.

4. For most images, the **L-poly (spill)** slider is all that is required to remove any remaining blue spill. Move the slider to the right to remove spill color from the sampled pixels. Move it to the left to move the selected pixels toward the color in the original foreground image. When using the **L-poly (spill)** slider, spill color replacement is replaced based on the setting of the **Spill Process replacement with** control. For more information on these tools, see the section of this chapter on **Replacing Spill**.

   **Tip:** It is better to make several small adjustments to the blue spill areas than a single major one.

5. You can use the other two sliders in the same way for different key adjustments. The **S-poly (detail)** slider controls the matte softness for the color which is closest to the background color. For example, you can recover lost smoke in the foreground by selecting the **Fine Tuning Sliders** action, sampling the area of the image where the smoke just starts to disappear and moving the **S-poly (detail)** slider to the left. The **M-poly (trans)** slider controls the matte softness for the color which is closest to the foreground color. For example, if you have thick and opaque smoke in the foreground, you can make it semi-transparent by moving the **Transparency** slider to the right after selecting the pixels in the **Fine Tuning Sliders** mode.

   **Tip:** If the foreground image changed color dramatically during the fine tuning process, you can recover the original color by selecting an area of the off-color foreground image and moving the **L-poly (spill)** slider slightly to the left. This may introduce spill back into that color region. Again, use the **Fine Tuning Sliders** option to suppress the spill, but make smaller adjustments this time.
Spill Removal - Method #3

If these final spill suppression operations have changed the final compositing results, you may have to return to earlier operations to clean up the matte. If the composite view looks good, it is a good idea to go back and take a final look at the alpha channel view. Sometimes in the Primatte operation, a 100% foreground area (all white) becomes slightly transparent (gray). You can clean those transparent areas up by using the Matte Sponge tool.

1. Select the Matte Sponge tool in the operation dropdown menu.
2. Click on the transparent pixels and they become 100% foreground. All of the spill-suppression information remains intact.

Note: The Matte(+) operation also works to solve this problem. For more information, see Sampling Tools.
Sampling Tools

Primatte's spill, matte, and detail sampling tools allow you to make fine adjustments to balance between these aspects of your composite. You can find these tools in the operation dropdown menu in the Primatte properties panel.

The Spill Sampling Tools

Using the Spill(+) and Spill(-) modes, you can gradually remove or recover the spill intensity on the foreground object by sampling the referenced color region repeatedly. The difference to the Spill Sponge tool is that Spill Sponge removes the spill component in a single action at one level and does not allow sampling the same pixel a second time. Even though just a small amount of spill needs to be removed, the Spill Sponge removes a preset amount without allowing any finer adjustment. To use the Spill(+) and Spill(-) tools:

1. Select the Spill(-) sampling tool from the operation dropdown menu.
2. In the Viewer, zoom into an area that has some blue edges.
3. Click on a pixel with some spill on it.
4. Repeated clicking incrementally removes the spill. Continue this operation until you reach a desired result.
5. To add spill, select the Spill(+) tool and repeat steps from 2 to 4.

The Matte Sampling Tools

The Matte(+) and Matte(-) modes are used to thicken or attenuate the matte information. If you want a thinner shadow on a foreground object, you can use the Matte(-) mode as many times as you like to make it more transparent. On the other hand, you can use the Matte(+) mode to make the matte thicker in that color region.
The Detail Sampling Tools

The Detail(+) and Detail(-) modes are a refined version of Clean BG Noise and Restore Detail. For example, when you see some dilute noise in the backing area but don't want to remove it completely because it affects some fine detail in a different area, try using Detail(-). It attenuates the noise gradually as multiple samples are made on the pixel. You should stop the sampling when important fine details start to disappear.
Replacing Spill

The proper processing of spill on foreground objects is one of the many useful features of Primatte. You can move between four modes to see how they affect the image clip you are working with. Under Spill Process, in the replace with dropdown menu, you can select the following options:

- **no suppression** - In this mode, no suppression is applied.

- **complement** - This is the default spill replacement mode. This mode maintains fine foreground detail and delivers the best quality results. If foreground spill is not a major problem, this mode is the one that should be used. The complement mode is sensitive to foreground spill. If the spill intensity on the foreground image is rather significant, this mode may often introduce serious noise in the resulting composite.

- **solid color** - In the solid color mode, the spill component is replaced by a palette color that you can pick. While the complement mode uses only the backing color complement to remove small amounts of spill in the original foreground, the solid color mode tries to assuage the noise using the user-defined palette color. Changing the palette color for the solid replacement, you can apply good spill replacement that matches the composite background. Its strength is that it works fine with even serious blue spill conditions.

On the negative side, when using the solid color 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.
- **defocused background** - The **defocused background** mode uses a defocused copy of the background image to determine the spill replacement colors instead of a solid palette color or just the complement color. This mode can result in good color tone on the foreground object even with a high contrast background. As in the example below, spill can even be removed from frosted glass using this feature and still retain the translucency.

On the negative side, the **defocused background** mode sometimes results in the fine edge detail of the foreground objects getting lost. Another problem could occur if you wanted to 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.

Blue suppression of a frosted glass object.
Primatte Controls

On the Primatte properties panel, you can further adjust several controls.

Initialize Section

In the Initialize section, you can select which algorithm Primatte uses to calculate your keying result:

- **Primatte** - The Primatte algorithm delivers the best results and supports both the solid color and complement color spill suppression methods. It is the algorithm that uses three multi-faceted polyhedrons (as described further down in this chapter) to separate the 3D RGB colorspace. It is also the default algorithm mode and, because it is computationally intensive, it may take longer to render.

- **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 further down in this chapter) and, as a result, does not have the ability to separate out the foreground from the backing screen as carefully as the above Primatte algorithm. Other disadvantages 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+** - this is in between the above two options. It uses a six planar surface color separation algorithm (as described further down in this document) and delivers results in between the other two in both quality and performance. Other disadvantages 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.

- **Reset** - Clicking this resets all of the Primatte properties to their initial values.

- **Auto-Compute** - This can be used as the first step in the Primatte operation. Its purpose is to try and do the first three steps of the Primatte operation for you. It tries to automatically detect the backing screen color, remove it, and do some clean-up on the foreground and background noise. If the clip was shot with an evenly lit, well-saturated backing screen, the Auto-Compute button leaves you with an image that may only need some spill removal to complete your keying operation. See Auto-Compute.

- **viewer** - This opens a Primatte Viewer that displays a graphical representation of the Primatte algorithms and allows you to see what is happening as the various Primatte tools are used. It is a
passive feature that has no adjustment capabilities, but it may prove useful in evaluating an image as you operate on it. See Primatte Viewer Tools.

When you select viewer, you are presented with a window that may look similar to one of these images (depending on which Primatte algorithm you have selected).

![Primatte algorithm](image1)

Primatte RT+ algorithm.

![Primatte RT algorithm](image2)

The different algorithms are described in more detail in a later section of this chapter, see The Primatte Algorithm. For a description of the Primatte viewer tools, see Primatte Viewer Tools.
Primatte Viewer Tools

To navigate within the Primatte Viewer, you can:
• drag to rotate the polyhedrons,
• Shift+drag to zoom in and out of the polyhedrons, and
• Ctrl/Cmd+drag to pan the polyhedrons.

At the top of the Primatte Viewer window, there are also three areas that you can click on:
• Clicking and dragging on the blue center area allows you to move the window around on the screen.
• Clicking and dragging on the triangular white region in the upper right corner allows you to scale the Primatte Viewer window.
• Clicking on the square white region in the upper left of the window displays a dropdown menu that looks like this:

![Dropdown Menu]

Note: A selected feature has a solid yellow square next to it. An unselected feature has a hollow yellow square next to it.

• **Minimize** - This feature, when selected, makes the Primatte Viewer window disappear. Only the blue title bar at the top of the window remains.
• **Large Surface** - This feature, when selected, displays the large Primatte polyhedron in the Viewer window.
• **Middle Surface** - This feature, when selected, displays the medium Primatte polyhedron in the Viewer window.
• **Small Surface** - This feature, when selected, displays the small Primatte polyhedron in the Viewer window.

• **Opaque** - This feature, when selected, makes the selected polyhedrons opaque. De-selecting it makes them semi-transparent.

• **Samples** - This feature, when selected, allows you to sample color regions on the image window using the 3D Sample **operation** mode and see where those regions are in relation to the polyhedron and the algorithm. The colors are displayed as a spray of pixels in the color selected. This button only allows you to see or hide the sampled colors.

  **Note:** The **3DSample** mode must be selected in the **operation** dropdown menu for this feature to work.

• **Clear BG** - This feature changes the background color of the Primatte Viewer window from black (when unselected) to transparent (when selected).

• **Sliced** - This feature, when selected, slices open the large and medium polyhedrons so that the inner polygons can be seen. When unselected, the largest polyhedron selected becomes a completely closed polyhedron and you might not be able see the inner polyhedrons (unless the **Opaque** feature is deselected).

• **Wireframe** - This feature, when selected, changes the polyhedrons from shaded-surface objects to wireframe objects.

• **Quad view** - This feature, when selected, displays the different polyhedrons in different parts of the Primatte Viewer:
  - The upper left corner of the Viewer shows the medium polyhedron.
  - The upper right corner of the Viewer shows the small polyhedron.
  - The lower left corner of the Viewer shows the large polyhedron as a wireframe object.
  - The lower right corner of the Viewer shows the large polyhedron.
Degrain Section

The Degrain tools are 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. These tools were created to, hopefully, help when a compositing artist is faced with a grainy image.

Degrain type

The Degrain type dropdown menu gives you a range of grain removal from none 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 BG Noise operation mode. These tools allow you to clean up the grain noise without affecting the quality of the key. A short tutorial explaining when and how to use these tools is at the end of this section. Select:

- **none** - When none is selected, you get the color of the exact pixel sampled. This is the default mode.
- **small** - When small is selected, you get 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** - When medium is selected, you get 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** - When large is selected, you get the average color of a larger region of the area around the sampled pixel. This should be used when the grain is very loose.
- **tolerance slider** - Adjusting the tolerance slider should increase the effect of the Clean BG Noise tool without changing the edge of the foreground object.

Degrain Tools Tutorial

If you have a noisy image as in the example below...
...you will find that the matte is also noisy:

You can use the **Clean BG Noise** operation to remove the noisy pixels, but this can also modify the edge of the foreground object in a negative manner.

Using the **Degrain** tools in the following way may help you clean up the image and still get a good edge on the matte:

1. Use the **Clean BG Noise** operation just a small amount to remove some of the white noise in the alpha channel view but do use it so much that you affect the edge of the foreground object.
2. Then set the **Degrain type** dropdown menu to **small** as a first step to reduce the grain:
With the degrain **tolerance** slider set at 0, move it around some. This should increase the affect of the **Clean BG Noise** tool without changing the edge of the foreground object. Sometimes, this may not be enough to totally remove the grain, so by adjusting the degrain **tolerance** slider, you can tell the Primatte algorithm what brightness of pixels you think represents grain. You should try not to use too high of a value; otherwise, it affects the overall matte. For an example of an over-adjusted image, see below.

The Primatte degrain algorithm uses a defocused foreground image to compute the noise.

**Note:** The small, medium and large settings for the degrain tools all produce defocused foregrounds that have larger or smaller blurs respectively.

**Note:** It is important to make sure that the **crop** settings are correctly applied; otherwise, when the defocus image is generated, if there is garbage on the edges of the images, then that garbage is blurred into the defocused foreground.

As a review:

1. Select the **Select BG Color operation** mode and click on a backing screen color.
2. Select the **Clean BG Noise operation** mode and use it sparingly so that it has minimum affect to the edge of the foreground object.

3. If there is still grain in the backing screen area, then use the **Degrain type** functionality starting at the **small** setting to reduce the grain.

4. If the grain is still present, then try increasing the **tolerance** slider a little - not too much.

5. If grain is still a problem, then try changing the **type** to **medium** or **large** and also changing the grain tolerance until the desired effect is achieved.

---

**Note:** The grain functionality does not always remove grain perfectly, but is sometimes useful to minimize its effects.
Actions Section

In the operations dropdown menu, you can select from the following options:

**Smart Select BG Color**

This operational mode gets the sampled backing screen color, analyzes the original foreground image, and determines the foreground areas using the Primatte foreground detection routine. Then, using the newly determined foreground areas, performs a **Clean FG Noise** operation internally and determines a more desirable shape for the middle and outer polyhedrons, before finally rendering the composite using the generated polyhedrons.

For keying operations, this is the first step and should be followed by the steps described immediately below.

**Clean BG Noise**

When you select this operational mode, you sample pixels in the Viewer known to be 100% background. White noisy areas in the 100% background region become black. This is usually the second step in using Primatte.

**Clean FG Noise**

When you select this operational mode, you sample pixels on the image window known to be 100% foreground. The color of the sampled pixels is registered by Primatte to be the same color as in the original foreground image. This makes dark gray areas in the 100% foreground region white. This is usually the third step in using Primatte.

**Matte Sponge**

When this operational mode is selected, the color sampled in the Viewer becomes 100% foreground. However, if the sampled color is already keyed out and removed, it leaves the current suppressed color. It only affects the key or matte information. This tool is usually used to quickly remove stray transparent pixels that have appeared during the chromakeying procedure. It is a quick and easy way to make final adjustments to a composite.
Make FG Trans

When this mode is selected, the opaque foreground color region sampled in the Viewer becomes slightly translucent. This operation is useful for the subtle tuning of foreground objects which are otherwise 100% covered with smoke or clouds. It can only be used one time on a particular color. For a more flexible way to thin out a color region and be able to make multiple samples, you should use the Matte (-) tool. It expands the medium polyhedron slightly.

Restore Detail

With this mode selected, the completely transparent background region sampled in the Viewer becomes translucent. This operation is useful for restoring lost hair details, thin wisps of smoke and the like. It shrinks the small polyhedron slightly.

Spill Sponge

When you select this operational mode, the background color component in the sampled pixels (or spill) within the image window is keyed out and removed for the color region selected. This operation can only be used once on a particular color and the amount of spill suppression applied is not adjustable. It is the fastest way to remove spill from a composite image. For more accurate spill suppression, a Fine Tuning or Spill (+) operation should follow or be used instead. This can usually be the fourth (and final) step in using Primatte unless additional adjustments are necessary.

Spill (-)

When this operational mode is selected, color spill is removed 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 tool removes more of the color spill. Continue using this tool until all color spill has been removed from the sampled color region. This tool expands the Primatte large polyhedron in the color region sampled.

Spill (+)

When this operational mode is selected, color spill is returned 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. This tool dents the Primatte large polyhedron in the color region sampled.
Matte (-)

When this operational mode is selected, the matte is made 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. This tool expands the Primatte medium polyhedron in the color region sampled.

Matte (+)

When this operational mode is selected, the matte is made 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. This tool dents the Primatte medium polyhedron in the color region sampled.

Detail (-)

When this operational mode 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. This is 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. This tool dents the Primatte small polyhedron in the color region sampled.

Detail (+)

When this operational mode is selected, 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. This tool expands the Primatte small polyhedron in the color region sampled.
Fine Tuning Sliders

If the spilled color has not been totally removed at this point, use the **Fine Tuning Sliders** for more subtle removal of spilled background color. For more information, refer to the section on **Fine Tuning**.

3D Sample

When this operational mode is selected and **viewer** is enabled in the Primatte properties, the sampled colors are displayed as a spray of white pixels in the Primatte Viewer. This allows you to see where the selected backing screen colors reside within the 3D RGB colorspace.

Simple Select BG Color

Although using the **Smart Select BG Color** operation is preferred, the **Simple Select BG Color** operation may still be useful in some cases. This operation uses the older 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 performs a **Clean FG Noise** operation internally and creates the shape of the middle and outer polyhedrons, before finally rendering the composite using the generated polyhedron.

Current Color Chip

This shows the current color selected (or registered) by the **Fine Tuning** operational mode.

Adjust Lighting

This enables Primatte's light adjusting feature, which, based on the currently selected BG color, generates a clean, evenly lit backing screen to use in the keying operation. This can improve the results if you have a very unevenly lit backing screen. For more information, see **Adjust Lighting**.
An uneven backing screen.

The resulting matte when **adjust lighting** is disabled.

The resulting key. Areas near the sampled color on the right look good, but on the left the hair is chunky.

The resulting matte when **adjust lighting** is enabled.

The resulting key is clean on both sides.

**Hybrid Render**

To tackle this problem, you can enable **hybrid render** to have Primatte create two keys internally and combine them for the best results. For more information, see **Hybrid Matte**.
Adjust Lighting

When **adjust lighting** is enabled, Primatte first creates a grid on the foreground image and samples the grid to determine which squares contain foreground information and which contain some shade of the background color. Using the remaining shades of blue, it uses extrapolations of the existing backing screen colors, and then uses this technique to fill in the area previously covered by the foreground object. This results in a clean backing screen that has no foreground object (to see this, you can set **output mode** to **adjust lighting BG**). Primatte uses this clean backing screen as the reference data to process the original foreground image to generate light-adjusted foreground that has a much more even shade of blue with the foreground object (this is displayed if you set **output mode** to **adjust lighting FG**). This allows Primatte to perform the keying operation using the light-adjusted foreground, resulting in a clean key around all areas on the foreground object.

The default settings under **Adjust Lighting** should detect all the areas of the grid that contain foreground pixels and deliver a smooth, artificially created, optimized backing screen for the keying. Should it fail to do this, you can adjust the settings of the algorithm by moving the **threshold** slider. This determines if a grid pixel should be treated as a pure background sample, a simulated background sample, or a foreground sample. Increasing the **threshold** value brings more of the foreground into the adjusted lighting.

If necessary, you can also set the **grid size** used in the **adjust lighting** algorithm. Increasing this value increases the grid resolution used in the adjusted lighting calculation.
Hybrid Matte

After sampling the backing screen color and finding that the edges of the foreground object look very good, you sometimes find that an area of the foreground object is transparent. This is due to the foreground containing a color that is close to the backing screen color. When this transparency is removed using the Clean FG Noise operation, the edge of the foreground object picks up a fringe that is close to the backing screen color. This is very hard to remove without sacrificing quality somewhere else on the image.

When hybrid render mode is enabled, Primatte internally creates two keys from the same image:

- **core** - This matte has the transparency removed, but suffers from the bad edges on the foreground object.

  ![Core Matte](image1)

- **edge** - This matte has a clean edge on the foreground, but suffers from transparency within the foreground object.

  ![Edge Matte](image2)

The core matte with the bad edges is then blurred and eroded before it is composited over the edge matte that has the transparency, resulting in a composite with the best of both options.

The controls under Hybrid Matte allow you to adjust the operations performed on the core matte:

- **erode** - Adjust the amount of erosion performed on the core matte. To view the results, set output mode to hybrid core and view the alpha channel.

- **blur radius** - Adjust the blur radius used when blurring the core matte. To view the results, set output mode to hybrid edge and view the alpha channel.
Fine Tuning

When the Fine Tuning Sliders operational mode is selected, the color of the sampled pixel within the Viewer window is registered as a reference color for fine tuning. It is displayed in the current color chip in the Actions section. To perform the tuning operation, sample a color region on the image, select a Fine Tuning slider and move the slider to achieve the desired effect. See the Fine Tuning Sliders tool descriptions further on in this section for more details on slider selection.

Spill or L-poly slider (Spill Removal)

When in the Fine Tuning operation mode, this Spill slider can be used to remove spill from the registered color region. After selecting the Fine Tuning Actions mode and registering a color region, this slider can be moved to remove spill from the registered 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 all the way to the right does not remove all the spill, re-sample the color region and again move the slider to the right. These slider operations are additive. This result achieved by moving the slider to the right can also be achieved by clicking on the color region using the Spill(-) operational mode. This slider bulges the Primatte large polyhedron near the registered color region.

Transparency or M-poly slider (Adjust Transparency)

When in the Fine Tuning operation mode, this Transparency slider can be used to make the matte more translucent in the registered color region. After selecting the Fine Tuning operation mode and selected a color region, moving this slider to the right makes the registered color region more transparent. Moving the slider to the left makes the matte more opaque. If moving the slider all the way to the right does not make the color region translucent enough, re-sample the color region and again move the slider to the right. These slider operations are additive. This result achieved by moving the slider to the right can also be achieved by clicking on the color region using the Matte(-) operational mode. This slider bulges the Primatte medium polyhedron near the registered color region.

Detail or S-poly slider (Add/Restore Lost Detail)

When in the Fine Tuning operation mode, this Detail slider can be used to restore lost detail. After selecting the Fine Tuning operation mode and selected a color region, moving this slider to the right makes the registered color region more visible. Moving the slider to the left makes the color region...
less visible. If moving the slider all the way to the right does not make the color region visible enough, re-sample 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. This shrinks the small polyhedron (which contains all the blue or green background colors) and releases pixels that were close to the background color. The **S-poly** slider in the **Fine Tuning** mode is useful for restoring pixels that were lost because they were so similar to the background color. This slider dents the Primatte small polyhedron near the registered color region.
Spill Process Section

The controls in the **Spill Process** section of the Primatte properties panel let you specify how to replace the spill color.

Replace with

This allows you to select between the three methods of spill replacement:

- **no suppression** - No spill suppression is applied.
- **complement** - Replaces the spill color with the complement of the backing screen color.
- **solid color** - Replaces the spill color with a solid color of your choice.
- **defocused background** - Replaces the spill color with colors from a defocused version of the background image.

Replace color slider

When **solid color** is selected, this sets the color to use to replace the spill.

Defocus slider

When **defocused background** is selected, this area allows you to adjust the amount of defocus applied to the background buffer image.
Output Section

These are the formats for the output of the node (output mode):

- **composite** - outputs the composite result of the Primatte node, along with the calculated matte.

- **premultiplied** - outputs the premultiplied result of the Primatte node, along with the calculated matte.

This can be useful if you want to do your compositing using a Merge node (with operation set to over) rather than Primatte. This allows you to color correct, transform, and otherwise process your image before compositing it over the background. Note, however, that Primatte works within the sRGB colorspace, whereas Nuke works within a linear colorspace. This means you need to add a Colorspace node after both Primatte and your original background image to convert their colorspace to sRGB, then do your color corrections or transforms, merge the images together, and, finally, use another Colorspace node to convert the result back to linear.
• **unpremultiplied** - outputs the unpremultiplied result of the Primatte node, along with the calculated matte.

This can be useful if you want to do your compositing using a Merge node (with operation set to **matte**) rather than Primatte. This allows you to color correct, transform, and otherwise process your image before compositing it over the background. Note, however, that Primatte works within the sRGB colorspace, whereas Nuke works within a linear colorspace. This means you need to add a Colorspace node after both Primatte and your original background image to convert their colorspace to sRGB, then do your color corrections or transforms, merge the images together, and, finally, use another Colorspace node to convert the result back to linear.

• **not premultiplied** - outputs the unpremultiplied original foreground (rather than the result of the Primatte node), along with the calculated matte.

• **adjust lighting FG** - outputs the light-adjusted foreground that the **adjust lighting** mode creates (this has a more even shade of the backing color with the foreground object). If **adjust lighting** is disabled, this option simply displays the un-optimized, original foreground image. For more information, see **Actions Section**.
• **adjust lighting BG** - outputs the optimized artificial backing screen that the **adjust lighting** mode creates (a clean backing screen that has no foreground object). For more information, see Actions Section.

• **hybrid core** - outputs the internally generated **core** matte, used when **hybrid render** is enabled. For more information, see Actions Section.

• **hybrid edge** - outputs the internally generated **edge** matte, used when **hybrid render** is enabled. For more information, see Actions Section.
The Primatte Algorithm

There are three Primatte algorithms. Here is a chart that shows the main differences between them.

<table>
<thead>
<tr>
<th>Number of Separating Surfaces</th>
<th>Primatte</th>
<th>Primatte RT Plus</th>
<th>Primatte RT</th>
</tr>
</thead>
<tbody>
<tr>
<td>128 (one for each color vector)</td>
<td>6</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Saturated FG Support</th>
<th>Primatte</th>
<th>Primatte RT Plus</th>
<th>Primatte RT</th>
</tr>
</thead>
<tbody>
<tr>
<td>OK</td>
<td>Not Supported</td>
<td>Not Supported</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Color Suppression Model</th>
<th>Primatte</th>
<th>Primatte RT Plus</th>
<th>Primatte RT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Replacement/Complement</td>
<td>Replacement</td>
<td>Replacement</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Pixel Calculation Cost</th>
<th>Primatte</th>
<th>Primatte RT Plus</th>
<th>Primatte RT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Heavy</td>
<td>Light</td>
<td>Very Light</td>
<td></td>
</tr>
</tbody>
</table>

For a description of the Primatte algorithm, see Explanation of How Primatte Works.

For a description of the Primatte RT+ algorithm, see Explanation of How Primatte RT+ works.

For a description of the Primatte RT algorithm see Explanation of How Primatte RT works.
Explanation of How Primatte Works

The Primatte chromakey algorithm is a sophisticated method of color space segmentation that can be easily explained to help a user achieve maximum effectiveness with the tool. Basically, Primatte segments all the colors in the foreground image into one of four separate categories. The result is a spill suppressed foreground image and a matte which is used to apply the modified foreground to a suitable background.

Primatte works in 3D RGB color space. Here is a visual representation of the Primatte algorithm after an image has been processed.

By operating the Primatte interface, you essentially create three concentric, multi-faceted polyhedrons. These can be pictured as three globes (or polyhedrons or polys), one within the other, which share a common center point. The creation of these polyhedrons separates all possible foreground colors into one of four regions; inside the small polyhedron (1), between the small and medium polyhedrons (2), between the medium and the large polyhedrons (3) and outside the large polyhedron (4).
The four regions created are described as follows:

**Region 1** (inside the small polyhedron) - This region contains all of the foreground image colors that are considered 100% background. These are the green or blue or whatever colors that were used as the backing color of the foreground image.

**Region 2** (between the small and medium polyhedrons) - This region contains all the foreground colors that are at the edges of the foreground object(s), in glass, glass reflections, shadows, sheets of water and other transparent and semi-transparent color regions. These color regions also have spill suppression applied to them to remove color spill from the backing screen.

**Region 3** (between the medium and large polyhedrons) - This region contains all the foreground image colors that are 100% foreground but have spill suppression applied to them to remove color spill from the backing screen. Otherwise they are 100% solid foreground colors.
Region 4 (outside the large polyhedron) - This region contains all the 100% foreground image colors that are not modified from the original foreground image. There is no spill suppression applied to these colors.

In the first step in using Primatte (Smart Select BG Color), you are asked to indicate the backing color on the original foreground image. The sample should usually be taken from a medium shaded area near the foreground object. By medium shaded area, it is meant that if green is the backing color and the green area of the foreground image has many shades of green ranging from very pale green to almost black, a shade of green in-between these extreme ranges should be chosen. If you’re not getting good results using this sample, you should reset Primatte and take another sample using a slightly darker or lighter shade of green. The first sample of Primatte often determines the final result as the center point of all three polyhedrons is created based on this first sample.

A single pixel may be selected or a range of pixels (snail trail or rectangular sample). If a range of pixels is taken, the sample is averaged to get a single color sample. This single pixel or averaged color sample then becomes the center of the small polyhedron. A few other shades around that color are included in the original small polyhedron.
**Note:** It is recommended that a single pixel be selected as the first sample as you then have some idea where the center point of the polyhedrons is located. If a box sample or a long snail trail sample is made, you can only guess at the average color that ends up being the center point. You can get an idea how this sample affects the algorithm by resetting the Primatte node, viewing the alpha channel, and clicking around on the green or blue screen area while in the **Smart Select BG Color** operation mode. You can immediately see the results of the initial settings of the polyhedrons in this way.

The second step in using Primatte is to clean up the backing color area by adding additional shades of green or blue to the small poly. This second step (**Clean BG Noise**) is usually executed while viewing the black and white alpha channel.

![Before background noise removal.](image1.jpg) ![After background noise removal.](image2.jpg)

While in the **Clean Bg Noise** sampling mode, you sample the white milky regions as shown in the left-hand image above. As you sample these regions, they turn to black as shown in the right-hand image above.

What is happening in the Primatte algorithm is that these new shades of green (the white milky areas) are added to the small poly where all the shades of green or blue are moved. The advantage of this technique is that the polyhedron distorts to enclose only the shades of green that are in the backing screen. Other shades of green around these colors are left undisturbed in the foreground.

Now that you have created a small polyhedron, you need to shape the medium and large polys. A default medium and large poly are both automatically created and are then modified based on the next couple of Primatte operations. The third Primatte step (**Clean FG Noise**) is to sample and eliminate gray areas in the 100% foreground area of the image.
Again, you make several samples on the dark, grayish areas on the foreground object until it is solid white in color. Primatte is shaping the large polyhedron with each color region that is sampled. Care should be taken in both this and the previous steps to not sample too close to the edges of the foreground object. Getting too close to the foreground object’s edges results in hard edges around the foreground object. Primatte uses these samples to modify and shape the medium and large polys to the desired shape. At this point, the matte or key has been created and would allow the foreground objects to be composited into a new background image.

If you now view the RGB channels, there is usually color spill on the edges (and sometimes the center) of the foreground objects. When on the edges of the foreground object, this spill comes from where the edges of the foreground object blended into the backing color. If it is on the center of the foreground object, it usually results from reflected color from the backing screen. The next Primatte step, either Spill Sponge, Fine Tuning or Spill(-), can now be used to eliminate this spill color.

Let's take a look at what is happening in the Primatte algorithm while this next step is performed. Here is what the various tools in Primatte do to the Polyhedrons when they are used:
As you can see above, the **Spill Sponge** bulges the large polyhedron in the color region specified. A color region is specified by clicking on the image in a particular area with spill present. For example, if you click on some spill on the cheek of a foreground person, Primatte goes to the section of the large polyhedron closest to that particular flesh tone and bulges the polyhedron there. As a result, the flesh tones move from outside the large poly to in-between the medium and large polys. This is **Region 3** and, if you remember, is 100% foreground with spill suppression. As a result of the suppression, the spill is removed from that cheek color and all other shades of that color on the foreground. You would then continue to sample areas of the image where spill exists and each sample would remove spill from another color region.

When all spill has been removed, you should have a final composite. As a last step, you should view the alpha channel and make sure that gray, transparent areas have not appeared in the foreground area. If there are any, you should select the **Matte Sponge** operation mode and sample those gray pixels until they have all turned white again.

The **Matte Sponge** and **Spill Sponge** tools bulge or dent the polyhedrons a pre-selected amount. If the desired results are not achieved or the results are too extreme for the image, a manual method can be applied. You should select the **Fine Tuning** sliders, select a color region of interest and then move the appropriate slider to get the desired results.

For example, to remove spill, select a region of the composite image with spill on it. Move the spill or large poly slider to the right a little bit, the large poly bulges and the spill should disappear. Move it a little more, if necessary. Moving this slider to the right removes spill (moves the colors from outside...
the large poly to between the medium and large polyhedrons) and moving it to the left, dents the large poly and moves that color region to outside the large poly.

If you sample a foreground object shadow and then move the M-poly (trans) slider to the right, the shadow becomes more transparent. This is useful for matching composited shadows to shadows on the plate photography. It can also be used to make clouds or smoke more transparent.

If some foreground detail disappears during the composite, you can select where the detail should be and then move the S-poly (detail) slider to the left. This dents the small poly in that color region and releases the detail pixels from the small poly into the visible region between the small and medium polyhedrons.

The Spill Sponge and Matte Sponge tools are shortcut tools that automatically move the sliders a pre-selected amount as a timesaving step for you. Other shortcut tools include the Make FG Trans. tool and the Restore Detail tool.

These shortcut tools are one-step operations where you click on a color region of interest and Primatte performs a pre-calculated operation. Hopefully, most operations using Primatte would only require these tools, but the manual operation of the sliders is always an option.

The Spill(-) tool bulges the large poly a small amount incrementally in the color region that is clicked on and the Spill(+) tool dents it a small amount with each click. The Matte(-) and Matte(+) tools do the same to the medium poly and the Detail(-) and Detail(+) do it to the small poly.
Explanation of How Primatte RT+ works

The Primatte RT+ algorithm differs from the Primatte algorithm in that it has a six surface color separator instead of the 128-faceted polyhedrons. This makes the Primatte RT+ algorithm much simpler and, therefore, faster to calculate. The results and performance of Primatte RT+ falls in between the Primatte and Primatte RT options. Where the Primatte RT+ algorithm might not work well is with less saturated backing screen colors and it also does not support the complement color spill suppression method (which is the spill suppression method that delivers the best detail). For a well-lit and photographed image or clip, this algorithm produces good results and render quickly.

Here is what a visual representation of the Primatte RT algorithm looks like after an image has been processed:

![Visual representation of Primatte RT algorithm](image)

Explanation of How Primatte RT works

Primatte RT is the simplest algorithm and, therefore, the fastest. It uses only a single planar surface to separate the 3D RGB colorspace and, as a result, does not have the ability to separate out the foreground from the backing screen as carefully as the above Primatte algorithm. Like the Primatte RT+ algorithm, Primatte RT might not work well with less saturated backing screen colors and it too does not support the complement color spill suppression method (which is the spill suppression method that delivers the best detail). For a well-lit and photographed image or clip, this algorithm produces good results and render very quickly.

Here is what a visual representation of the Primatte RT algorithm looks like after an image has been processed:
Contact Details

Here are the contact details for the Photron main office and Primatte office.

Main Office

Photron USA, Inc., 9520 Padgett Street, Suite 110, San Diego, CA 92126, USA

Primatte Office

Photron USA, Inc., 3113 Woodleigh Lane, Cameron Park, CA 95682. Phone: 1-530-677-9980, FAX: 1-530-677-9981, Cell: 1-530-613-3212, E-mail: sgross@photron.com, Website: http://primatte.com

Proprietary Notices

Primatte is distributed and licensed by Photron USA, Inc., San Diego, CA, USA. Primatte was developed by IMAGICA Corp., Tokyo, Japan. Primatte is a trademark of IMAGICA Co IMAGICA Digix Inc., Tokyo, Japan.
Keying with Ultimatte

This section explains how to use the blue/green screen keyer, Ultimatte, in Nuke.

Ultimatte Quick Start

Here's a quick overview of the workflow:

1. Sample the screen (backing) color. For more information, see Sampling the Screen Color.
2. Refine the overlay using the overlay tools and, if needed, adjust the controls on the Ultimatte tab. For more information, see Using Overlay Tools and Screen Correct.
3. Next refine the matte density using the matte tools and, if needed, adjust the controls on the Density tab. For more information, see Adjusting the Density of the Matte.
4. If necessary, activate Shadow processing, and use Shadow tool and, if needed, adjust the controls on the Shadow tab. For more information, see Retaining Shadows and Removing Noise.
5. If necessary, improve Spill Suppression using spill tools and, if needed, adjust the controls on the Spill tab. For more information, see Adjusting Spill Controls.
6. If necessary, adjust the Cleanup controls, but in general you'll get better results by using ScreenCorrect and MatteDensity controls. For more information, see Retaining Shadows and Removing Noise.
7. If necessary, you can adjust the controls on the Color tab to match your blacks, whites and gammas between the foreground and the background. For more information, see Adjusting Color Controls.
8. If necessary, activate film processing and adjust its settings on the Film tab. For more information, see Adjusting Film Controls.
Connecting the Ultimatte Node

Start up Nuke, create an Ultimatte node by clicking **Keyer > Ultimatte** and connect a foreground and a background image (and any additional inputs you want) to it. Add a Nuke Viewer node so you can see the result.

To use the Ultimatte inputs

In Ultimatte, you have a number of inputs you can use to connect images and mattes that you need to key your footage. Connect the following inputs to your images and/or mattes as necessary:

- **foreground (fg)** - Connect your foreground image to this input.
- **background (bg)** - Connect your background image to this input.
- **clean plate (cp)** - Connect your clean plate to this input. For more information on clean plates, see [Using Overlay Tools and Screen Correct](#).
- **garbage matte (gm)** - Connect your garbage matte to this input. A garbage matte is often used to clean up screens that are not a constant color or have lighting rigs in shot by forcing the alpha transparent.
- **holdout matte (hm)** - Connect your holdout matte to this input. A holdout matte is used when foreground objects are the same color or very close to the backing color. The holdout matte is used to let Ultimatte know that the pixels in the holdout matte region should be considered 100% opaque. Note that, unlike the other inputs, this input is hidden and appears as a small triangle on the left hand side of the node.
Sampling the Screen Color

The first step when keying with Ultimatte is to sample the screen color, or tell Ultimatte what color your blue or green screen is. Do the following:

1. When you've connected the Ultimatte node, the foreground image is displayed in the Viewer.
2. Click Screen in the Ultimatte toolbar above the Viewer, and select the screen color by holding down Alt+Ctrl/Cmd and clicking in the Viewer.

   You should select an area on the green screen near important subject detail that is not obscured in any way. The image is rendered and a composite is displayed. If further adjustments are needed, use the controls described below.

Adjusting the Controls on Ultimatte Tab

After you've sampled your screen color, you can start adjusting your result. With the controls on the Ultimatte tab you can select different sets of controls you want to enable on other properties panel tabs. There's also an enable checkbox on each tab that you can use to activate the controls on that tab and the corresponding tools in the Viewer.

- **Film** - check this to reduce the effects of cyan undercutting on the Film tab.
- **screen correct** - check this to use the controls on Screen Correct tab.
- **shadow** - check this to use the controls on the Shadow tab.
- **spill suppression** - check this to use the controls on the Spill tab.
- **cleanup** - check this to use the controls on the Cleanup tab.
- **color conformance** - check this to use the controls on the Color tab.
Using Overlay Tools and Screen Correct

Screen Correct compensates for anomalies in the backing area such as uneven lighting, smudges, seams, variation in the backing color, green set pieces, and unwanted shadows cast by set pieces, boom arms, etc. This technique assumes that an exact copy of the problematic green screen element with the subject matter omitted, usually called a Clean Plate, is supplied.

Although this technique gives great results by differentiating between foreground elements and backing flaws, you often haven't got the necessary reference materials. In that case, you can create a synthetic Clean Plate with Ultimatte.

To achieve the best results, use a reference Clean Plate as an input to Ultimatte in addition to allowing Ultimatte to generate a synthetic plate. In this way, the reference plate allows for the best shadow separation, while the synthetic plate is used to reduce artifacts such as film grain or video noise (which is rarely accurately reproduced when the clean plate is shot during the time of principal photography). Switch the view to screen by clicking the overlay dropdown menu and selecting screen.

With screen correct selected, use the add overlay tool to scrub on areas that represent the green screen, including shadows. Usually, it's best to press Alt+Ctrl/Cmd while scrubbing, so that you are picking the color in the input image at that pixel, instead of the color of the processed image.

The overlay is used in the final process to fully eliminate these areas from the final composite. Continue in this manner until the foreground subject and its shadows are surrounded by an overlay. Make sure the overlay does not cover any foreground detail that is to be retained. If the overlay does cover foreground subject matter, then use the remove overlay tool to scrub on those areas that should not be covered by the overlay. Repeat these two processes until the overlay covers the green screen areas and no subject matter. View the composite in the Viewer to see the screen-corrected composite with anomalies in the backing area removed. To learn which controls were automatically set by Ultimatte, click the Screen Correct tab and note the position of the controls.

Note: When scrubbing on the image using add overlay, the RGB value of the points selected are accumulated in a keep list and remove overlay points are accumulated in a drop list. If both lists have equivalent values, a conflict may occur, resulting in no visible change in the overlay. If a conflict occurs, try using Undo (Ctrl/Cmd+X) which flushes the last
set of points added to the appropriate list. If that doesn't help, you can also press the **Reset** button on the **ScreenCorrect** tab to clear the lists and start over.

Under some circumstances, it may be difficult to completely cover the screen area with the overlay. This does not mean that the screen cannot be removed in that area, but that other controls may need to be adjusted to help reduce anomalies in those areas that were not covered by the overlay. It may be advantageous to resample the screen using the **Screen** tool above the Viewer in the area where covering the screen with the overlay is unsuccessful, but be careful about selecting in shadow areas if the final composite requires the retention of shadows.

Additionally, it may be difficult in some circumstances to remove the overlay from very dark areas of foreground detail that may be close to values in dark or shadow areas of the screen. If the overlay does include some dark foreground areas, these areas may be recovered by enabling the **Shadows** controls.

**Note:** Since Ultimatte produces a synthetic clean plate using an interpolation process, it is important to exclude non-video areas of the image, such as the black blanking lines that may appear in video or the black areas in letterbox images. You can use other nodes, such as **Crop**, to exclude these areas from the overlay and final composites. Otherwise the synthetic clean plate may contain black (or other colors) which can result in an inaccurate clean plate being generated.

**Adjusting overlay controls**

You can choose your overlay view, color and output mode in the properties panel, on the **Ultimatte** tab.

- **overlay** - The overlay control is used to show the calculated overlay on the Viewer and it's helpful for debugging purposes. It helps tune the screen correct controls and also the add/remove overlay tools. In this way you can see immediately if something is wrong and needs to be adjusted. In the overlay dropdown menu, change
  - **off** - to not view the overlay.
  - **screen** - to view the subject in clear, and the preliminary matte area blended with the overlay color.
  - **subject** - show the subject blended with overlay color, and the preliminary matte area in clear.
  - **show image as monochrome** - check this to make the input image appear in grayscale so that the overlay areas can be more easily distinguished.
• overlay color - use this to change the color of the overlay. You can adjust the alpha channel to modify the opacity of the overlay.

Adjusting the screen correct controls

• screen tolerance - This is used to adjust the color range or tolerance to be included or excluded from the screen overlay.
• shrink - Use this control to shrink the screen overlay.
• darks(reds smaller) - Use this control to include or exclude dark areas from the screen overlay in those areas where the blue value (when using green screen) is greater than the red value in the original foreground image.
• darks(reds larger) - Use this control to include or exclude dark areas from the screen overlay in those areas where the red value is greater than the blue value (when using green screen) in the original foreground image.
• brights(reds smaller) - Use this control to include or exclude bright areas from the screen overlay in those areas where the blue value (when using green screen) is greater than the red value in the original foreground image.
• brights(reds larger) - Use this control to include or exclude bright areas from the screen overlay in those areas where the red value is greater than the blue value (when using green screen) in the original foreground image.
• orphans - Use this to include or exclude neighboring "rogue" pixels from the screen overlay.
Adjusting the Density of the Matte

The controls on the **Density** tab are used to adjust the density or opacity of the foreground objects. The density of a foreground object is determined by its matte (alpha) value. A completely opaque object's matte is white, a completely transparent object's matte is black, and a partially transparent object's matte is gray. Use the **add matte** dropper to scrub on areas in the matte that appear gray, but should be white (fully opaque) in the matte. These areas are described as "print-through", meaning that the opacity of the subject is too low in this area and the background is visible through the foreground in this area. Be careful not to select those objects that should have a gray matte value such as fine hair, smoke or partially transparent objects, as they become opaque. To learn which controls were automatically set by Ultimatte, click the **Density** tab in the properties panel and note the position of the controls.

**Note:** If there are sections of the matte which should be opaque but are exhibiting gray values that don't look like typical transparency, then there is a chance that there is a remainder of overlay in this area. If overlay exists on subject matter, return the **Overlay** dropdown menu to **Screen** and use the **removeoverlay** tool to scrub on that area. Check on **show image as monochrome** on the **Ultimatte** tab to aid in determining the extent of the overlay.

Adjusting Density controls

Use these controls to adjust the density of your matte:

- **brights** - Use this control to adjust density in bright foreground objects. Advancing this control too far can cause hard, dark edges around foreground subjects.
- **darks** - Use this control to adjust density in black glossy or dark foreground objects.
- **edge kernel** - Use this control to adjust number of pixels to use as a kernel to reduce dark edges that may exist in transition areas due to an over-dense matte. Advancing this control too far, may cause too much print-through at the edges.
- **warm** - Use this control to adjust density in warm colors (flesh tones). Note that reducing this control too much can cause print-through in reddish foreground objects.
- **cool** - Use this control to adjust density in cool colors. Note that reducing this control too much can cause print-through in bluish foreground objects.
Adjusting Spill Controls

Ultimatte automatically suppresses spill from the backing onto foreground subject matter. The spill controls are used to suppress excessive spill or to retain color similar to spill that has been over-suppressed. To adjust the spill controls, check the spill suppression box on the Ultimatte tab and click Spill tab:

- **cool** - Use this control to adjust the amount of spill in cool colored foreground objects. Used to reproduce blue, green or cyan colors that changed through the spill suppression algorithms.
- **warm** - Use this control to adjust the amount of spill in warm colored foreground objects. Used to reproduce pink, purple and magenta colors for bluescreen, or yellow and orange colors for green screen that changed through the spill suppression algorithms.
- **midtones** - Use this control to adjust the amount of spill in mid-range foreground objects.
- **brights** - Use this control to adjust the amount of spill on bright foreground objects.
- **darks** - Use this control to adjust the amount of spill on dark foreground objects.
- **ambience** - Use this control to select a color to subtly influence the foreground objects in areas that may have contained spill.
- **strength** - Use this control to adjust the intensity of the selected ambience color.
- **background veiling** - This control is used to override the automatic suppression of the backing color. Ultimatte uses the selected backing color to suppress the backing to black. An indication that the automatic settings did not suppress enough backing is "veiling" or a colorized haze in some background areas. An indication that the automatic settings suppressed too much backing is darkened or mis-colored foreground edges and transparencies. In most cases this control should be left at the default setting.

**Tip:** There are several nodes in Nuke you can use for spill removal. For example, if you are using a greenscreen image, you can add an Expression node after your foreground image and set the expression field for the green channel to:

```
g>(r+b)/2?(r+b)/2:g
```

Similarly, you can despill a bluescreen image by setting the expression field for the blue channel to:

```
b>(r+g)/2?(r+g)/2:b
```
You can also use the HueCorrect node for despill. For more information, see Correcting Hue Only.
Retaining Shadows and Removing Noise

Use the **hold shadow** dropper (only available when **screen correct** and **shadow** are enabled) to scrub on the shadows that you'd like to preserve. These shadows may best be seen in the foreground image.

If unwanted shadows remain, then use the **remove noise** dropper to reduce or eliminate those shadows. If the area that you scrub does not reside under the overlay, then erasing adjusts the appropriate **Cleanup** controls to reduce anomalies in the screen area. This might result in losing some fine detail. Repeat using these two tools until the shadows you want are retained and any shadows or noise you don't want to keep are removed.

The density, sharpness, and tint of the shadow may be altered by manually adjusting the controls in **Shadow** controls. To learn which controls were automatically set by Ultimatte, click the **Shadow** and/or **Cleanup** tab in the properties panel and note the position of the controls.

### Adjusting Shadows controls

Check the **shadow** box on the **Ultimatte** tab and adjust the following controls:

- **high** - Decrease this to reduce or eliminate unwanted shadows (which must be lighter than those shadows that are to be retained). All preserved shadows are lighter.
- **low** - Increase this to restore the density of the darkest part of the shadows that are to be retained.
- **density** - Use this to change the density of the shadows that are retained to better match shadows in the background scene.
- **blur** - Use this to blur the shadows.
- **tint** - Use this to tint the shadows.

### Adjusting Cleanup controls

The following controls are used to adjust the black and gray areas of the matte channel. This dramatically affects the nature of foreground objects' edges, the opacity of transparent objects, and the noise in the foreground image. Use these controls sparingly as they might result in the loss of foreground detail. Using **Screen Correction** for dealing with imperfections in the screen is a good alternative for using **Cleanup**.
• **cleanup** - Use this control to reduce imperfections or small amounts of noise in the backing. Note that adjusting this control too far results in a "cut and paste" look with a hard, unnatural edge. Background noise (as well as some foreground detail) is reduced. An alternative method for dealing with green screen imperfections is to use the **Screen correct** controls.

• **shrink** - Use this control to choke or reduce the size of the cleaned-up matte.

• **blur** - Use this control to soften the cleaned-up matte.

• **recover** - Use this control to set a threshold below which the **Cleanup** control does not have influence.
Adjusting Color Controls

Check the **color conformance** box on the **Ultimatte** tab, and set these controls on the **Color** tab. A good keying result requires matched blacks, whites, and gammas between the foreground and background elements. With color conformance you can select blacks, whites, and gammas and can automatically match the foreground to the background (or vice versa).

- **darks** - You can use this control to influence the darkest parts of the image. This is a global control that affects the entire image, but the greatest effect is seen in the darkest areas.
- **midtones** - You can use this control to influence the mid-range parts of the image. This is a global control that affects the entire image, but the greatest effect is seen in the mid-range areas.
- **brights** - You can use this control to influence the brightest parts of the image. This is a global control that affects the entire image, but you can see the greatest effect in the brightest areas.
- **hue** - This control changes the color contents of the image without changing its brightness or color intensity (purity) values. At default setting (0), the image hue is not altered. The range of the control extends from -300 to +300.
- **saturation** - This control changes the color intensity or purity values of the image without altering its color contents or brightness values. At default setting (0), the image saturation is not altered. At minimum setting (-200), the color intensity is reduced to zero and the image is monochrome, or shades of gray.
- **brightness** - This control changes the overall intensity of the image. At the default setting, the image brightness is not altered.
Adjusting Film Controls

Use the controls on the **Film** tab to reduce the effects of cyan undercutting. These controls are only available when **Film** box is checked on **Ultimatte** tab. You can view the results better when you select **subject** in the **overlay** dropdown menu on the **Ultimatte** tab.

Due to the nature of film's emulsion layers, a phenomenon known as cyan undercutting exists that reveals itself as a smeared red edge in areas of sharp transitions, which can best be seen by viewing the individual RGB channels of a film image. Normally, this phenomenon is not a problem until bluescreen compositing techniques are applied. Since the red emulsion layer tracks at a slower rate than the blue and green emulsion layers, the artificial red edge it produces are retained as foreground detail resulting in an unacceptable composite.

- **transparency** - Use this control to adjust the amount of Film Correction in partially transparent foreground objects (such as hair detail).
- **correction** - Use this control to adjust the amount of correction in the individual RGB channels of the foreground image.
- **strength** - Use this control to adjust the overall amount of Film Correction that is applied to the foreground image.
- **shrink** - Use this control to shrink the subject overlay.
- **brights** - Use this control to include or eliminate bright areas from the subject overlay.
- **darks** - Use this control to include or eliminate dark areas from the subject overlay.
Choosing an Output Mode

You can output the merged foreground and background as a final composite, or you can output the premultiplied or unpremultiplied foreground for compositing elsewhere in your node tree. In the output mode dropdown menu, select:

• composite - to output a merged layer of the background input with the extracted subject.
• premultiplied - to output the extracted subject premultiplied by the final matte.
• unpremultiplied - to output the extracted subject and final matte unpremultiplied.
Using RotoPaint

Nuke features a vector-based RotoPaint node for help with tasks like rotooning, rig removal, garbage matting, and dustbusting. You can draw Bezier and B-Spline shapes with individual and layer group attributes, including per-point and global feather, motion blur, blending modes and individual or hierarchical 2D transformations. This chapter gives full instructions on its usage.

Roto or RotoPaint?

There are two similar nodes in Nuke for rotooning, Roto and RotoPaint. The main difference between these two is that you can only create and edit Bezier and B-spline shapes with Roto, while RotoPaint allows you to draw paint strokes too with various brushes. So the Roto node is an optimal choice if you're doing rotooning only, whereas RotoPaint gives you a broader scale of tools to use.

All tools and controls in the Roto node work the same way as they do in RotoPaint, so you can learn about using them in the RotoPaint instructions in this chapter. For instance, see:

- Working with the Toolbars for information about the toolbars in Roto.
- Working with the Stroke/Shape List for information about the shape list in the Roto properties panel.
- Drawing Shapes for information about using the Bezier and B-Spline Tools.
- Selecting Existing Strokes/Shapes for Editing for information about selecting shapes.
- Editing Shape-Specific Attributes for information about editing Bezier and B-Spline attributes.
- Editing Existing Stroke/Shape Timing for information about changing the timing of your shape.
- Animating Strokes/Shapes for information about editing your shapes.

RotoPaint Quick Start

Here's a quick overview of the workflow:

1. Connect the RotoPaint node to a Viewer and possible backgrounds. For more information, see Connecting the RotoPaint Node.

2. Select a stroke or a shape tool from the RotoPaint toolbar on the left side of the Viewer. For more information, see Drawing Paint Strokes or Drawing Shapes.

3. Use the RotoPaint tool settings on the top of the Viewer to adjust the stroke/shape you're about to draw. For more information, see Working with the Toolbars.
4. Draw one or more strokes/shapes in the Viewer window. For more information, see for example Using the Brush tool or Using the Bezier and Cusped Bezier Tools.

5. Select an existing stroke/shape using the Select tools or the stroke/shape list. For more information, see Selecting Existing Strokes/Shapes for Editing.

6. Use the control panel controls to adjust your existing stroke/shape. For more information, see Editing Existing Stroke/Shape Attributes.

In addition you can:

• Adjust the splines of your stroke/shape. For more information, see Editing Existing Stroke/Shape Splines.

• Animate your strokes/shapes. For more information, see Animating Strokes/Shapes.

• Use RotoPaint in stereoscopic projects. For more information, see RotoPaint and Stereoscopic Projects.

• Set your favorite RotoPaint tool as your default tool. For more information, see Setting Default RotoPaint Tools and Settings.
Connecting the RotoPaint Node

The RotoPaint node accepts one primary input. Even if you have no inputs, you can still use RotoPaint and in that case, you can use the format control to select your output format (by default, the format control is hidden, but you can display it by clicking on the black triangle above color).

To Connect the RotoPaint Node

1. Click Draw > RotoPaint to add a new RotoPaint node. You can also press P on the Node Graph. To create a Roto node, you can press O on the Node Graph.
2. Drag the bg input to the node that you want to apply RotoPaint to.
3. Connect any additional background elements you wish to use. If you plan to reveal pixels from a background element, drag the bg1 input from the left side of the node to the node whose output you wish to use.
Working with the Toolbars

In the RotoPaint node, you can use two toolbars to define the type of stroke/shape you want to start drawing. These toolbars are placed in the Viewer. The vertical RotoPaint toolbar is for selecting the tool you want to use and the horizontal one, RotoPaint tool settings, is for adjusting the currently selected tool’s settings before drawing new strokes/shapes.

**Note:** You can’t use RotoPaint tool settings to adjust an already existing stroke/shape. For any changes you want to make to a stroke/shape after you’ve created one, you can use the controls in the RotoPaint properties panel.

In the RotoPaint toolbar, you can select your stroke/shape tool. The tools are grouped under the toolbar icons. You can click any tool to make it active and view a tool group by right-clicking (or left-clicking and holding) the icon. The tool that is currently selected is highlighted.

**Tip:** When the Viewer has mouse-over focus, you can also use the $S$ keyboard shortcut to cycle through the modes of the currently selected tool.

In the RotoPaint tool settings on the top of the Viewer, you can define the settings for the tool that you’ve selected. The controls in this toolbar change depending on which tool you have selected at any given time.
**Tip:** You can hide the toolbars by clicking the hide button next to them. You can also press [ (square bracket) to hide the RotoPaint toolbar and { (curly bracket) to hide the tool settings (and the entire top toolbar of the Viewer).
Working with the Stroke/Shape List

After you've drawn strokes/shapes, you can edit their order and group them with the Stroke/Shape list in the RotoPaint properties panel. By default, the newest stroke/shape/group appears on top of the list, and your strokes/shapes are named according to their type (for example, "Bezier1" or "Smear2").

Using the list, you can also select strokes/shapes/groups and edit them with the various controls in the properties panel. Some controls can be applied to groups, some can't. Some controls also can only be applied to strokes or shapes (these are grouped under the Stroke and Shape control tabs respectively). If a control can't be applied to a group, it is grayed out if you have a group selected in the stroke/shape list.

The stroke/shape list provides an overview of existing parameters, showing, for example, whether the item is locked, set invisible, or whether it has motion blur applied to it. Some controls can also be edited directly in the overview by clicking on their icon.

To Use the Stroke/Shape List

You can edit the stroke/shape list in many ways, and use it to adjust strokes/shapes and how they're displayed in the Viewer.

- You can reorder the columns in the stroke/shape list by dragging and dropping them.
- You can create groups for sets of strokes/shapes in the stroke/shape list by clicking the Add button below the list. This creates a subfolder, named “Layer1” by default, and you can drag and drop strokes/shapes to this folder to group them. After strokes/shapes have been grouped, you can edit them as a group and they move together if you change their place in the Viewer. Every group also has its own transformation overlay, which you can use to move the group.
- You can remove a stroke/shape or a group by clicking the Remove button under the stroke/shape list.
- If you want to rename any of the strokes, shapes, or groups, double-click on the name while the item is selected, and give your item a new name. The name has to be unique, so you can't give two items the same name.
- You can also cut, copy, and paste strokes and shapes by right-clicking on them in the control panel and using the copy > curve, cut > curve, and paste > spline options in the menu that appears.
For information on copying, cutting and pasting shapes and points, see Copying, Pasting, and Cutting Stroke Positions; or for animated shapes and points, see Animating Strokes/Shapes. Alternatively, to copy, cut, or paste shape and point attributes only, see Editing Existing Stroke/Shape Splines.

**Note:** You cannot use Ctrl/Cmd+C or Ctrl/Cmd+V in the control panel to copy and paste shapes. You must use the right-click menu.

- You can duplicate strokes/shapes by right-clicking on them and selecting duplicate. A new stroke/shape is created with the same spline, animation, and attributes as the one you selected.

- You can hide a stroke, shape, or group by clicking the Visible icon in the stroke/shape list. You can still edit an invisible stroke/shape and view its position in the Viewer.

- You can lock strokes/shapes to prevent them from being edited. To lock an item in the stroke/shape list, click the Lock column in the list. A lock icon appears next to the Visible icon.

- You can select the color in which you want the outline of your stroke/shape to appear in the Viewer. Click the Overlay column and select your overlay color. To be able to see the splines for all paint strokes in the Viewer, you need to activate one of the Select tools in the RotoPaint toolbar and click in the tool settings.

- You can change the color of your stroke/shape in the stroke/shape list by clicking the Color column and using the color picker to select the color.

- You can invert a shape using the Invert column. With your shape selected, click in the Invert column to toggle between inverted and uninverted modes.

- You can select a blending mode for your stroke/shape using the Blending column. With your shape selected, click the Blending column and select the mode you want to apply.

- You can apply shape motion blur using the Motion blur column. With your shape selected, click the Motion blur column to toggle the motion blur effect.

See Adding Motion Blur for more information on shape and global motion blur.

- You can specify the frame range of certain strokes and shapes by navigating to the stroke/shape list of the RotoPaint node, and right-clicking all under the Life column. Select frame range from the pop-up menu, and specify the frame range desired in the Set frame range dialog.

By default, new shapes and strokes are set to all on creation.
Tip: To undo and redo any changes you've made with the RotoPaint node, use the **Undo** and **Redo** buttons on top of the properties panel. Undo uses a cache memory of your changes, so at any time you can undo and redo all the changes you've made since you last opened your project.
# Drawing Paint Strokes

Any given RotoPaint node can hold many paint strokes and shapes. You can apply paint strokes using any of the following tools (see also Drawing Shapes).

<table>
<thead>
<tr>
<th>Icon</th>
<th>Tool</th>
<th>Keyboard Shortcut</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Brush" /></td>
<td>Brush</td>
<td>N (toggles between Brush and Eraser)</td>
<td>Applies colors atop the current plate, or blends colors with the current plate. You can also clone from surrounding frames.</td>
</tr>
<tr>
<td><img src="image" alt="Eraser" /></td>
<td>Eraser</td>
<td>N</td>
<td>Removes pixels from existing strokes and brings background back.</td>
</tr>
<tr>
<td><img src="image" alt="Clone" /></td>
<td>Clone</td>
<td>C (toggles between clone and Reveal)</td>
<td>Applies pixels from one region of the current plate to another region of the current plate.</td>
</tr>
<tr>
<td><img src="image" alt="Reveal" /></td>
<td>Reveal</td>
<td>C</td>
<td>Applies pixels from a source plate to a destination plate in the corresponding place.</td>
</tr>
<tr>
<td><img src="image" alt="Blur" /></td>
<td>Blur</td>
<td>X (toggles between Blur, Sharpen and Smear)</td>
<td>Blurs the image in the area of the brush stroke.</td>
</tr>
<tr>
<td><img src="image" alt="Sharpen" /></td>
<td>Sharpen</td>
<td>X</td>
<td>Sharpens the image in the area of the brush stroke.</td>
</tr>
<tr>
<td><img src="image" alt="Smear" /></td>
<td>Smear</td>
<td>X</td>
<td>Smears the area of the smear brush stroke, stretching the selected pixels over their surrounding area.</td>
</tr>
<tr>
<td><img src="image" alt="Dodge" /></td>
<td>Dodge</td>
<td>D (toggles between Dodge and Burn)</td>
<td>Brightens the background color on the area of the brush stroke to reflect the brush stroke. Using this tool on black produces no change. No part of the stroke area is darkened.</td>
</tr>
<tr>
<td><img src="image" alt="Burn" /></td>
<td>Burn</td>
<td>D</td>
<td>Darkens the background color on the area of the brush stroke to reflect the brush stroke. No part of the stroke area is lightened.</td>
</tr>
</tbody>
</table>
**Tip:** You can choose to have your RotoPaint always open with a particular tool selected. If you want to open it with the Brush tool selected, for example, do the following:

1. Create a file called `menu.py` in your plug-in path directory if one doesn't already exist. For more information on plug-in path directories, see *Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts*.
2. To select the Brush tool as your default tool, save the following in your `menu.py`:
   ```python
   nuke.knobDefault('RotoPaint.toolbox','brush')
   ```

For more information on default tools, see *Setting Default RotoPaint Tools and Settings*.

**Tip:** If you are using a tablet, you can tie a new stroke's opacity, size, or hardness to pen pressure by clicking the corresponding *pressure alters* checkbox in the RotoPaint tool settings. You can also later change which attribute the pressure alters for an existing stroke on the *Stroke* tab of the properties panel.

Separate Select tools in the RotoPaint toolbar let you select strokes/shapes once they've been drawn and after that you can make changes to them using the properties panel controls (see *Selecting Existing Strokes/Shapes for Editing*, and *Editing Existing Stroke/Shape Attributes*).

These pages discuss the general steps for using each of these tools, and give an overview on editing their attributes and timing.
Using the Brush tool

The **Brush** tool lets you apply colored or blended strokes to the current plate.

![Painting with the Brush tool.](image)

**To Use the Brush Tool**

1. Click the **Brush** tool in the RotoPaint toolbar.
2. Set color, opacity, brush type, brush size, and brush hardness in the RotoPaint tool settings at the top of the Viewer. (For information on the available options, see [Editing Existing Stroke/Shape Attributes](#).) You can also change the size of the brush using `Shift`+drag directly in the Viewer as shortcut.
3. Optionally, set the lifetime of the stroke in the RotoPaint tool settings. (For information on the available options, see [Editing Existing Stroke/Shape Timing](#).)
4. Apply strokes as necessary.

**Tip**: If you need to draw a completely straight line with the brush tool, try this: Draw a freehand stroke, activate the **Select All** tool, enable `show paint stroke splines` mode in the tool settings, and marquee select and delete all but the first and last point on the stroke.
Using the Eraser Tool

The Eraser tool lets you remove pixels from existing paint strokes.

To Use the Eraser Tool

1. Set opacity, brush type, brush size, and brush hardness in the RotoPaint tool settings at the top of the Viewer. (For information on the available options, see Selecting Existing Strokes/Shapes for Editing.)

2. Optionally, set the lifetime of the stroke in the RotoPaint tool settings. (For information on the available options, see Selecting Existing Strokes/Shapes for Editing.)

3. Right-click the Brush tool in the RotoPaint toolbar and select the Eraser tool.

4. Apply strokes as necessary. You can also erase multiple strokes, if you have drawn more than one.

Unwanted paint strokes. Using the Eraser tool to remove the unwanted paint strokes.

Tip: If you're using a graphics tablet, Nuke automatically switches to Eraser mode when you use the erase end of the pen.
Using the Clone Tool

The **Clone** tool lets you remove unwanted features from the plate or from a different input by painting over them with pixels offset from the pointer or a transformation of the pointer.

Painting with the Clone tool.

**To Use the Clone Tool**

1. Click the **Clone** tool 🎨 in the RotoPaint toolbar.

2. To view all the settings for the Clone tool, enable **show clone settings** ⏰ in the RotoPaint tool settings at the top of the Viewer.

3. In the RotoPaint tool settings, set the paint source dropdown menu to the input you want to clone pixels from. (For information on the available options, see *Editing Stroke-Specific Attributes*.)

   You can also use the transform controls in the clone settings 🛡️ to transform the clone source and reset it back to original with the **reset** button.

4. Set opacity, lifetime, brush type, size, and hardness for your stroke in the tool settings. (For more information, see *Editing Existing Stroke/Shape Attributes* and *Editing Existing Stroke/Shape Timing*.) If you've got more than one view set up, you can check the **single view** box to only clone in one view, or uncheck it to clone in all of them.

5. To set the clone offset, hold down **Ctrl/Cmd** and left-click at the source location, drag to where you want to paint, and release. Alternatively, you can enter the offset numerically using the **translate** controls in the RotoPaint tool settings. If you'd like the offset amount to be rounded to an integer (whole number of pixels), check **round**. Rounding to a pixel can be useful if you don't want to soften the image by partially blending pixels together.

6. Start painting. The pointer overlay depicts the source of the offset as a crosshair within a circle and the destination as a circle (the diameter of which represents the breadth of the stroke).
You can use / (forward slash) and * (asterisk) on the numeric keypad to zoom your clone source in and out, and 0 (zero) and . (decimal point) to rotate it right and left. You can also use the number keys on the numeric keypad to nudge the clone source.

7. To reset the clone offset, you can use Ctrl/Cmd+drag to adjust the offset you set before, or Ctrl/Cmd+Shift+drag to start a new offset from the brush pointer’s location.

**Tip:** If you’re cloning from the current plate (foreground), you’re also cloning all the strokes/shapes you’ve previously drawn. If you want to clone from the original background or a different picture, you need to set the paint source dropdown menu to pull from that input.

**Tip:** To clone pixels from another frame of the input clip, you can use the time offset slider to define which frame you want to clone from. See Editing Clone or Reveal Attributes.
Using the Reveal Tool

The **Reveal** tool lets you pull pixels from background elements onto the current plate. The **Reveal** tool requires at least two inputs (see [Connecting the RotoPaint Node](#)); otherwise, your strokes draw in white. You can also view which input you are using as the source for your strokes in the **Source** column in the stroke/shape list.

![The current plate.](image1) ![The source input.](image2)

Painting with the Reveal tool.

**To Use the Reveal Tool**

1. Right-click the **Clone** tool in the RotoPaint toolbar and select the **Reveal** tool.
2. In the RotoPaint tool settings at the top of the Viewer, set the paint **source** dropdown menu to the input you want to pull pixels from. (For information on the available options, see [Editing Stroke-Specific Attributes](#).)
3. Set opacity, brush size, and brush hardness in the RotoPaint tool settings. (For information on the available options, see [Editing Existing Stroke/Shape Attributes](#).)
4. Optionally, set the lifetime of the stroke in the RotoPaint tool settings. (For information on the available options, see [Editing Existing Stroke/Shape Timing](#).)
5. You can also reveal pixels from another frame of the input clip by using the **time offset** slider to define which frame you want to reveal from. See [Editing Clone or Reveal Attributes](#).
6. If you want, you can view your revealing source image in a Viewer overlay. To do this, check the **onion** box in the RotoPaint tool settings and enter an onion skin value to adjust the opacity of the
overlay. This can help you better see what you are revealing from the source image. You can also toggle the keyboard shortcut T to enable or disable onion skin.

7. Start painting.

Onion skin disabled.  Onion skin enabled.
Using the Blur Tool

The **Blur** tool lets you blur parts of the plate.

Painting with the Blur tool.

**To Use the Blur Tool**

1. Click the **Blur** tool 🌊 in the RotoPaint toolbar.
2. Set opacity, brush type, brush size, and brush hardness in the RotoPaint tool settings. (For information on the available options, see *Editing Existing Stroke/Shape Attributes.*)
3. Optionally, set the lifetime of the stroke in the RotoPaint tool settings. (For information on the available options, see *Editing Existing Stroke/Shape Timing.*)
4. Apply strokes by clicking on the part of image you want to blur.

**Tip:** You can use the **effect** control in the RotoPaint tool settings to adjust the strength of the tool you’re using. With the **Blur** tool, it controls the amount of blur.
Using the Sharpen Tool

With the **Sharpen** tool, you can sharpen the image within the area of the brush stroke.

![Painting with the Sharpen tool.](image)

**To Use the Sharpen Tool**

1. Right-click the **Blur** tool and select **Sharpen** tool.
2. Set opacity, brush type, brush size, and brush hardness in the RotoPaint tool settings at the top of the Viewer. (For information on the available options, see *Editing Existing Stroke/Shape Attributes*.)
3. Optionally, set the lifetime of the stroke in the RotoPaint tool settings. (For information on the available options, see *Editing Existing Stroke/Shape Timing*.)
4. Apply strokes by clicking on the part of image you want to sharpen.

**Tip:** You can use the **effect** control in the RotoPaint tool settings to adjust the strength of the tool you're using. With the **Sharpen** tool, it controls how much the image is sharpened.
Using the Smear Tool

With the Smear tool, you can smear or stretch pixels over the surrounding pixels.

Painting with the Smear tool.

To Use the Smear Tool

1. Right-click the Blur tool and select Smear tool 🎨.
2. Set opacity, brush type, brush size, and brush hardness in the RotoPaint tool settings at the top of the Viewer. (For information on the available options, see Editing Existing Stroke/Shape Attributes.)
3. Optionally, set the lifetime of the stroke in the RotoPaint tool settings. (For information on the available options, see Editing Existing Stroke/Shape Timing.)
4. Apply strokes by clicking and dragging on the part of image you want to smear.
Using the Dodge Tool

With the Dodge tool, you can lighten the pixels in the area of the brush stroke. This makes the background color brighter on the area of the brush stroke to reflect the brush stroke.

Painting with the Dodge tool.

To Use the Dodge Tool

1. Click the Dodge tool.
2. Set opacity, brush type, brush size, and brush hardness in the RotoPaint tool settings at the top of the Viewer. (For information on the available options, see Editing Existing Stroke/Shape Attributes.)
3. Optionally, set the lifetime of the stroke in the RotoPaint tool settings. (For information on the available options, see Editing Existing Stroke/Shape Timing.)
4. Apply strokes as necessary.
Using the Burn Tool

With the **Burn** tool, you can darken the pixels in the area of the brush stroke. This makes the background color darker on the area of the brush stroke.

![Painting with the Burn tool.](image)

**To Use the Burn Tool**

1. Right-click the **Dodge** tool and select **Burn** tool.
2. Set opacity, brush type, brush size, and brush hardness in the RotoPaint tool settings. (For more information on the available options, see **Editing Existing Stroke/Shape Attributes**.)
3. Optionally, set the lifetime of the stroke in the RotoPaint tool settings. (For information on the available options, see **Editing Existing Stroke/Shape Timing**.)
4. Apply strokes as necessary.
Drawing Shapes

Any given RotoPaint node can hold several shapes, and you can draw them using any of the following tools.

<table>
<thead>
<tr>
<th>Icon</th>
<th>Tool</th>
<th>Keyboard Shortcut</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Bezier]</td>
<td>Bezier</td>
<td>V (toggles between Bezier, B-spline, Ellipse, and Rectangle)</td>
<td>Applies a Bezier shape. Bezier shapes are defined using control points and tangents.</td>
</tr>
<tr>
<td>![Cusped Bezier]</td>
<td>Cusped Bezier</td>
<td>V</td>
<td>Applies a Bezier shape with sharp corners and no tangents.</td>
</tr>
<tr>
<td>![B-Spline]</td>
<td>B-Spline</td>
<td>V</td>
<td>Applies a B-spline shape. Unlike Bezier shapes, B-splines are created by only using control points. The position of the points in relation to each other determines what kind of splines the shape consists of.</td>
</tr>
<tr>
<td>![Ellipse]</td>
<td>Ellipse</td>
<td>V</td>
<td>Applies an ellipse-shaped Bezier shape.</td>
</tr>
<tr>
<td>![Rectangle]</td>
<td>Rectangle</td>
<td>V</td>
<td>Applies a rectangle-shaped Bezier shape.</td>
</tr>
<tr>
<td>![Cusped Rectangle]</td>
<td>Cusped Rectangle</td>
<td>V</td>
<td>Applies a rectangle-shaped Bezier shape with sharp corners and no tangents. You can see the difference between rectangles and cusped rectangles if you move at least one of the control points.</td>
</tr>
<tr>
<td>![Open Spline]</td>
<td>Open Spline</td>
<td>V</td>
<td>Lets you draw curves in a similar way to other shapes, except that they don’t need form a closed shape.</td>
</tr>
</tbody>
</table>

Separate Select tools in the RotoPaint toolbar let you make changes to a stroke/shape once it’s been drawn (see Selecting Existing Strokes/Shapes for Editing).
Tip: You can choose to have your RotoPaint always open with a particular tool selected. If you want to open it with the Bezier tool selected, for example, do the following:
1. Create a file called menu.py in your plug-in path directory if one doesn't already exist. For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.
2. To select Bezier as your default tool, save the following in your menu.py:
   ```python
   nuke.knobDefault('RotoPaint.toolbox','bezier')
   ```

For more information on default tools, see Setting Default RotoPaint Tools and Settings.

Tip: If you begin drawing shapes and then decide you want to planar track these with the PlanarTracker node, you can do that easily by right-clicking on the shape in the stroke/shape list and selecting planar-track. This creates a new planar tracking layer for the shape and attaches the shape to it. For more information about PlanarTracker, see Tracking with PlanarTracker.
Using the Bezier and Cusped Bezier Tools

The Bezier and Cusped Bezier tools let you draw Bezier shapes. Cusped Beziers are Bezier shapes with sharp corners and no tangents.

Drawing a Bezier shape.

To Use the Bezier or Cusped Bezier Tools

1. Do one of the following:
   • Click the Bezier tool in the RotoPaint toolbar.
   • Right-click the Bezier tool in the RotoPaint toolbar and select the Cusped Bezier tool.

2. Select color, blending mode, opacity, and other settings for the shape in the RotoPaint tool settings at the top of the Viewer. (For information on the available options, see Editing Existing Stroke/Shape Attributes.)

3. Optionally, set the lifetime of the shape in the RotoPaint tool settings. (For information on the available options, see Editing Existing Stroke/Shape Timing on page 581.)

4. Draw a shape in the Viewer by clicking to create the outlines that make up your shape.

5. If you’re using the Bezier tool, you can click+drag while drawing to create a point and adjust its tangent handles. With the tangent handles, you can adjust the spline of your shape.
   • You can move individual handles to adjust their length, keeping the angle consistent.
   • Press Shift while moving the tangent handles to move both handles at the same time, keeping the angle consistent.
   • Press Ctrl/Cmd to temporarily break the angle.

6. You can also use other shortcuts to adjust your shape while drawing:
• **Shift**+click to create a sharp exit point on the previous point (this has no effect when using the Cusped Bezier tool, as all the points are sharp anyway).

![Curved exit point.](image1.jpg) ![Sharp exit point.](image2.jpg)

• **Ctrl/Cmd**+click to sketch your shape freely.

7. To close your shape, press **Return** or click the first point of your shape. Changing to a different tool also closes a shape. By default, closing a shape activates the Select tool.
   
   If you’re using the Bezier tool and close your shape by clicking its first point, you can also drag the point to create tangent handles for adjusting it.

8. With the Select tool active, you can **Shift**+click on multiple shape points to bring up the transform box, which you can use to further transform your shape or particular points in your shape.

9. You can also cusp and smooth a point in your Bezier shape to create a sharp corner on your shape or smooth it again. To do this, right-click on a point in your Bezier shape and select **cusp/de-smooth** or **smooth**.

**Tip:** You can also apply animation from a Tracker node to a point in your Bezier shape. From the Tracker node, **Ctrl/Cmd**+drag the transformation information into the Viewer, on the point you want to animate. See also **Linking Expressions** for creating links between the Tracker node and RotoPaint controls.
Using the B-Spline Tool

The B-spline tool lets you draw B-spline shapes.

Drawing a B-spline shape.

To Use the B-Spline Tool

1. Right-click the Bezier tool in the RotoPaint toolbar and select B-Spline tool.
2. Select color, blending mode, opacity, and other settings for the shape in the RotoPaint tool settings at the top of the Viewer. (For information on the available options, see Editing Existing Stroke/Shape Attributes)
3. Optionally, set the lifetime of the shape in the RotoPaint tool settings. (For information on the available options, see Editing Existing Stroke/Shape Timing.)
4. Click in the Viewer to create the splines that make up your shape.
   - Ctrl/Cmd+drag to sketch the spline freely.
   - Click+drag the cursor left or right to adjust the tension (that is, the distance between the spline line and a point) on the previous point.
   - To adjust the tension of any point in the B-spline shape you're drawing, select a point, press Ctrl/Cmd+Shift, and drag left or right: right to increase the tension, left to decrease it.
5. To close your shape, press Return. Changing to a different tool also closes a shape. By default, closing a shape activates the Select All tool.
6. With the Select All tool active, you can Shift+click on multiple shape points to bring up the transform box, which you can use to further transform your shape or particular points in your shape.
7. You can also cusp and smooth a point in your B-spline shape to create a sharp corner on your shape or smooth it again. To do this, right-click on a point in your B-spline shape and select cusp/de-smooth or smooth.
**Tip:** You can convert your B-spline shape into a Bezier shape after creating it. Select the B-spline shape using the Select All tool, right-click on it and select `convert bspline to bezier`. You can now edit your shape in the same way as other Bezier shapes.
Using the Ellipse, Rectangle, and Cusped Rectangle Tools

The Ellipse, Rectangle, and Cusped Rectangle tools let you draw an ellipse- or rectangle-shaped Bezier shape. Rectangles and cusped rectangles look exactly the same initially, but if you move one or more of the control points, you can see that cusped rectangles always have sharp corners and no tangents.

After creation, ellipses, rectangles, and cusped rectangles can be edited like normal Bezier shapes.

Here, an ellipse and a rectangle have been used to create a quick garbage matte.

To Use the Ellipse, Rectangle, and Cusped Rectangle Tools

1. Right-click the Bezier tool in the RotoPaint toolbar and select the Ellipse tool , Rectangle tool , or Cusped Rectangle tool .
2. Select color, blending mode, opacity, and other settings for the shape in the RotoPaint tool settings at the top of the Viewer. (For information on the available options, see Editing Existing Stroke/Shape Attributes.)
3. Optionally, set the lifetime of the shape in the RotoPaint tool settings. (For information on the available options, see Editing Existing Stroke/Shape Timing.)
4. Click+drag across the Viewer to draw an ellipse or a rectangle shape.

   If you want to draw a shape from the center out, press Ctrl/Cmd+Shift as you’re dragging.
If you want to create a perfect circle with the **Ellipse** tool or a square with the **Rectangle** tool, hold down **Shift** while drawing your shape.

Holding down **Shift** while drawing an ellipse creates a perfect circle.
Using the Open Spline Tool

The **Open Spline** tool lets you draw curves in a similar way to other shapes, except that they don't need form a closed shape.

After creation, you can edit points on open splines using the standard smooth and transform handles, but they also have individual thickness and feather handles. The image shows an open spline and each point's smooth (orange), thickness (green), and feather (magenta) handles.

Here, an open spline is used to roto a creature's tail.

To Use the Open Spline Tool

1. Right-click the **Bezier** tool in the RotoPaint toolbar and select the **Open Spline** tool.

   ![Tip: You can also press V, when the Viewer has focus, to cycle through the available tools.](image)

2. Select color, blending mode, opacity, and other settings applicable to all RotoPaint tool at the top of the Viewer. (For information on the available options, see Editing Existing Stroke/Shape Attributes.)

   RotoPaint has a few additional open spline-specific controls:
   - **width** - controls the overall thickness of the spline.
   - **start type** - sets the state of the first point in the spline to either rounded or square.
   - **end type** - sets the state of the last point in the spline to either rounded or square.

3. Click in the Viewer to place the starting point of the spline and then add the required number of points on the feature you're working on.
4. Press **Return** or select another tool to finish the open spline, OR

Click on the first point in the open spline to close the shape. Closed splines are not filled in the same way as other shape tools, such as **Bezier** and **Ellipse**. You can open a closed spline by right-clicking a point on the spline and selecting **open/close shape**.

![Open Spline Tool](image)

5. You can adjust the thickness and feather of individual points on the spline by selecting the required points and using the handles displayed in the Viewer.

Drag the handle and release to apply the change to the selected points.

![Spline Adjustments](image)

Decreasing and increasing spline thickness (green handle).
Decreasing and increasing feathering (magenta handle).

6. Optionally, set the lifetime of the shape in the RotoPaint tool settings. (For information on the available options, see Editing Existing Stroke/Shape Timing.)
Setting Default RotoPaint Tools and Settings

You can choose to have your RotoPaint always open with a particular tool selected, or open a particular tool with the settings you want.

To Set a Tool as Your Default Tool:

1. Create a file called `menu.py` in your plug-in path directory if one doesn't already exist. For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.
2. Select your default tool from the following list and add the Python line in your `menu.py`:
   - Select All
     ```python
     nuke.knobDefault('RotoPaint.toolbox','selectAll')
     ```
   - Select Splines
     ```python
     nuke.knobDefault('RotoPaint.toolbox','selectCurves')
     ```
   - Select Points
     ```python
     nuke.knobDefault('RotoPaint.toolbox','selectPoints')
     ```
   - Select Feather Points
     ```python
     nuke.knobDefault('RotoPaint.toolbox','selectFeatherPoints')
     ```
   - Add Points
     ```python
     nuke.knobDefault('RotoPaint.toolbox','addPoints')
     ```
   - Remove Points
     ```python
     nuke.knobDefault('RotoPaint.toolbox','removePoints')
     ```
   - Cusp Points
     ```python
     nuke.knobDefault('RotoPaint.toolbox','cuspPoints')
     ```
   - Smooth Points
     ```python
     nuke.knobDefault('RotoPaint.toolbox','curvePoints')
     ```
   - Remove Feather
     ```python
     nuke.knobDefault('RotoPaint.toolbox','removeFeather')
     ```
   - Open/Close Curve
     ```python
     nuke.knobDefault('RotoPaint.toolbox','closeCurve')
     ```
   - Bezier
     ```python
     nuke.knobDefault('RotoPaint.toolbox','createBezier')
     ```
   - Cusped Bezier
nuke.knobDefault('RotoPaint.toolbox','createBezierCusped')

• B-Spline
  nuke.knobDefault('RotoPaint.toolbox','createBSpline')

• Ellipse
  nuke.knobDefault('RotoPaint.toolbox','createEllipse')

• Rectangle
  nuke.knobDefault('RotoPaint.toolbox','createRectangle')

• Cusped Rectangle
  nuke.knobDefault('RotoPaint.toolbox','createRectangleCusped')

• Brush
  nuke.knobDefault('RotoPaint.toolbox','brush')

• Eraser
  nuke.knobDefault('RotoPaint.toolbox','eraser')

• Clone
  nuke.knobDefault('RotoPaint.toolbox','clone')

• Reveal
  nuke.knobDefault('RotoPaint.toolbox','reveal')

• Dodge
  nuke.knobDefault('RotoPaint.toolbox','dodge')

• Burn
  nuke.knobDefault('RotoPaint.toolbox','burn')

• Blur
  nuke.knobDefault('RotoPaint.toolbox','blur')

• Sharpen
  nuke.knobDefault('RotoPaint.toolbox','sharpen')

• Smear
  nuke.knobDefault('RotoPaint.toolbox','smear')


To Set Your Default Tool Properties:

1. Create a file called init.py in your plug-in path directory if one doesn't already exist. For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.

2. Define your RotoPaint tool and the default setting you want to set. To get the specific tool and setting names, you can copy your RotoPaint node in Nuke and paste it into a text editor. The lines with curved brackets after "toolbox" indicate your current tool settings.

   For example:
• to set the **brush size** for paint to **30**, and for **clone** to **280**, add this Python line in your *init.py*:

```python
nuke.knobDefault("RotoPaint.toolbox", "'clone {
{ brush bs 30 }
{ clone bs 280 }
}'')
```

**OR**

• to set the **brush hardness** for paint to **1.0** by default, add this Python line:

```python
nuke.knobDefault("RotoPaint.toolbox", "'brush {
{ brush h 1 }
}'')
```

• to set the **source transform** round on clone to **on** by default, add this Python line:

```python
nuke.knobDefault("RotoPaint.toolbox", "'clone {
{ clone str 1 }
}''")
```

3. **Restart Nuke.**

---

**Note:** As you can see from the above examples, the different RotoPaint tools have their own default tool settings. If you attempt to set a default but don’t specify for which tool, the default gets ignored as soon as you switch to a different tool. For example, `nuke.knobDefault('RotoPaint.toolbox_brush_hardness','1.0')` sets **brush hardness** to **1.0** initially, but as soon as you activate a different tool, you get the default for that tool.

---

**Note:** You cannot set up multiple defaults through `RotoPaint.toolbox`. For example, if you run the following three commands, the default is only set for the last one (sharpen):

```python
nuke.knobDefault("RotoPaint.toolbox", "'clone { clone ltt 0}'")
nuke.knobDefault("RotoPaint.toolbox", "'blur { blur ltt 0}'")
nuke.knobDefault("RotoPaint.toolbox", "'sharpen { sharpen ltt 0}'")
```
To set the defaults for clone and blur as well, you should instead use this:

```python
nuke.knobDefault("RotoPaint.toolbox", "'clone {
{ brush ltt 0 }
{ clone ltt 0 }
}'")
```
Selecting the Output Format and Channels

In the RotoPaint properties panel, you can select one output channel or many to indicate the channels where the results of your changes should be stored.

1. From the output field, select the layer containing the channels you want to use. By default, rgba is selected, and the red, green, blue, and alpha channels are checked on the right.
2. Uncheck any channels that you don’t want to process. The node processes all those you leave checked. For more information on selecting channels, see Calling Channels.
3. If you want to, you can use the output mask dropdown menu to select a channel where RotoPaint outputs a mask for what it rendered. By default, the channel is none, but if you select a channel in the menu, the output mask box is automatically checked.
   The mask can be useful, for example, if you need to apply grain to the areas you’ve painted, but you don’t want to double up the grain in other areas.
   By default, the output mask control is hidden, but you can display it by clicking on the black triangle above color.
4. If necessary, select your premultiply value.
   Premultiply multiplies the selected input channels with a mask representing the paint strokes and shapes. For example, where there are no paint strokes or shapes (the paint matte is black or empty) the input channels are set to black, and where the paint strokes or shapes are opaque (the paint matte is white or full) the input channels keep their full value.
   Note that selecting rgba premultiplies the alpha against itself (a*a). If you don’t want this to happen, set premultiply to rgb instead.

   **Tip:** You can use the Link Menu to link the channel controls with other controls. For more information, see Linking Channels Using the Link Menu.

5. From the clip to dropdown menu, select how you want to restrict the output image:
   • no clip - Do not restrict the output image.
   • bbox - Restrict the output image to the incoming bounding box.
   • format - Restrict the output image to the incoming format area (the default).
   • union bbox+format - Restrict the output image to a combination of the bounding box and the incoming format area.
• **intersect bbox+format** - Restrict the output image to an intersection of the bounding box and incoming format area.

You can also check the **Replace** box if you want to clear the channels to black before drawing into them. You might find **Replace** useful, for instance, if you're creating a mask in the alpha channel, but the incoming image already has an alpha channel which you want to throw away.

6. If you haven't connected an input to RotoPaint, select your **format** value. This is the format which the node should output in the absence of any available input format. If an input is connected, this control has no effect.

By default, the **format** control is hidden, but you can display it by clicking on the black triangle above **color**.
Selecting Existing Strokes/Shapes for Editing

If you've already drawn a stroke/shape but wish to make changes to it, you can select it or certain parts of it with the RotoPaint selection tools. You can also toggle between all of them in the RotoPaint toolbar using the shortcut Q.

When you click on a point in a stroke/shape, the point changes color to indicate whether it's in focus (green by default) and whether it has an expression set (red by default). You can change the default colors on the Viewers tab in the Preferences.

You can also use the controls in the RotoPaint tool settings to display and hide information such as points, point numbers, splines, and transform handles. See To Adjust Display Properties for Selected Shapes/Strokes below.

To Adjust Display Properties for Selected Shapes/Strokes

Whenever you have a Select tool active, the RotoPaint tool settings allow you to control what information is displayed for the visible shapes/strokes and points in the Viewer:

- To view or hide the numbers for visible shape/stroke points, toggle the label points button.
  Feather points are marked with a bracketed number corresponding to their shape point, so for example, if a shape point is marked with the number 2, its feather point is marked with [2].

- To view or hide the splines in visible shapes, toggle the hide splines button.

- To view or hide the splines in visible paint strokes, enable show paint stroke splines and toggle the hide splines button.

- To view or hide the points (and tangent handles) in visible shapes/strokes, toggle the hide points button.

- To view or hide the transform handle jack or transform box for visible shapes/strokes, toggle the hide transform handles button (or press T on the Viewer).

- To hide the transform handle jack or transform box when moving a selection, click the hide transform handles on move button. This may make it easier to correctly position your selection.
To Select an Entire Stroke/Shape

1. Click the Select All tool 🖉, or press the keyboard shortcut Q.
2. Select the stroke/shape you wish to edit either by clicking on it in the Viewer or by clicking on its name in the stroke/shape list. To select several strokes/shapes, Ctrl/Cmd+click or Shift+click (to select a range) their names in the stroke/shape list. You can have both splines and points selected simultaneously.

   When selecting strokes/shapes in the Viewer, you can invert your selection by right-clicking and selecting invert selection. All strokes/shapes you didn't have selected before are now selected.

   If you want to move the selected stroke/shape, drag a selection box around the entire shape until a box forms around the shape. Pulling on the cross at the center of the shape allows you to move it around the Viewer. A keyframe is automatically added when you move a stroke/shape, if autokey is enabled. For information on autokey and creating keyframes, see Animating Strokes/Shapes.

   **Tip:** To select points in a paint stroke and view your selection in the Viewer, you have to enable show paint stroke splines 📆 in the tool settings.

   **Tip:** By default, if you have selected a stroke/shape in the Viewer or the stroke/shape list, clicking on an empty area in the Viewer does not deselect it. If you'd like to change this behavior, you can disable constant selection mode 💡 in the RotoPaint tool settings.

To Select a Spline

1. Right-click the Select All tool and select the Select Splines tool 📆.
2. Make sure hide splines 🗑️ is disabled in the RotoPaint tool settings.
3. Select the spline you wish to edit either by clicking on it in the Viewer or by clicking on its name in the stroke/shape list. You can also click-and-drag a selection box around the shape with Select Splines active. Selecting a spline only selects the spline, not points within it.
Tip: Using the Select Splines tool, you can also duplicate the stroke/shape you've selected. Just right-click on one of the points, and select duplicate. A new stroke/shape is created with the same spline and attributes as the one you selected.

To Select Only Points on a Stroke/Shape

1. Right-click the Select All tool and select the Select Points tool. Using the Select Points tool restricts selection to one stroke/shape only.

2. Make sure hide points is disabled in the RotoPaint tool settings.

3. Select the stroke/shape you wish to edit by clicking on its name in the stroke/shape list and then select a point in the Viewer. To select several points, Ctrl/Cmd+click on them in the Viewer or use marquee selection to create a transform box.

Using the Select Points tool restricts selection to one stroke/shape only.

To Select Feather Points

1. Right-click the Select All tool and select the Select Feather Points tool. Using the Select Feather Points tool restricts selection to one stroke/shape only.

2. Make sure hide points is disabled in the RotoPaint tool settings.

3. Select the stroke/shape you want to edit by clicking on its name in the stroke/shape list and then select a feather point in the Viewer. To select several feather points, Ctrl/Cmd+click on them in the Viewer or use marquee selection to create a transform box.

Using the Select Feather Points tool restricts selection to one stroke/shape only.

To Select Points Using a Transform Handle

1. Right-click the Select All tool and select the Select Points tool. Using the Transform Handle tool, you can select points by dragging across them. A transform box appears.

2. Make sure hide transform handles is disabled in the RotoPaint tool settings.

3. Select points in a stroke/shape with Shift+click or by clicking and dragging across the points you want to select. A transform box appears.

4. You can also use shortcut T to toggle viewing the transform handle.
Editing Existing Stroke/Shape Attributes

After selecting a stroke/shape using the stroke/shape list or one of the Select tools, you can edit and animate its attributes in the properties panel. For more information on selecting strokes/shapes, see Selecting Existing Strokes/Shapes for Editing and Working with the Stroke/Shape List.

If you want to edit the attributes of a stroke/shape prior to drawing one, you should do that in the RotoPaint tool settings in the Viewer. (See Working with the Toolbars.)

Editing Color

When drawing a stroke/shape, you can set the RGBA color values of the stroke/shape using the color controls on the RotoPaint tab of the RotoPaint properties panel (for more information on the color controls, see the Using the Compositing Environment chapter). You can also adjust color directly in the stroke/shape list using the control in the color column.

![Color set to white (the default).](image1)

![Color set to green.](image2)

Editing Opacity

You can set the opacity of the stroke/shape using the opacity slider. If you set the opacity of a shape to zero, the outline for it won't be drawn unless the shape is selected. You can also temporarily make the stroke/shape invisible (that is, completely transparent) by toggling the visible box in the stroke/shape list.
A low opacity value. A high opacity value.

**Tip:** When drawing brush strokes, you can tie their transparency to pen pressure. Just check the opacity box next to pressure alters on the Stroke tab of the properties panel.

### Selecting a Source for Your Stroke/Shape

You can select a source for your stroke/shape and define whether it's a color, a background, or a foreground image. With your stroke/shape selected, choose a source for your stroke/shape from the source dropdown on the RotoPaint, Shape, or Stroke tabs.

### Editing Blending Mode

By selecting different blending modes from the blending mode dropdown in the properties panel, you can select how the colors in your strokes/shapes blend with the underlying image. You can also apply blending modes to your strokes, shapes, or groups directly in the stroke/shape list using the blending mode column.

Each of the blending modes blends the primary color, that is the color of the current stroke/shape/group you're editing with the secondary color, which is the combined color of your background image and any previously rendered strokes/shapes/groups.

The different modes are as follows:

- **Color burn** - Darkens the primary color to reflect the secondary color by increasing the contrast. No part of the image becomes lighter.
- **Color dodge** - Brightens the primary color to reflect the secondary color by decreasing the contrast. No part of the image is darkened.
• **Difference** - Subtracts either the secondary color from the primary color or vice versa, depending on which is brighter. Blending with white inverts the primary color, while blending with black produces no change. Similar colors return black pixels. Difference is a useful mode when working with mattes.

• **Exclusion** - Creates a result similar to the Difference mode but lower in contrast. Like with Difference, blending with white inverts the primary color. Blending with black produces no change.

• **From** - Subtracts the primary color from the secondary color.

• **Hard Light** - Lightens highlights and darkens shadows. If the secondary color is lighter than 50% gray, the result lightens as if it were screened. If the secondary color is darker than 50% gray, the result is darkened as if it were multiplied.

• **Max** - Selects the lighter of the two colors as the resulting color. Only areas darker than the secondary color are replaced, while areas lighter than the secondary color do not change.

• **Min** - Selects the darker of the two colors as the resulting color. Any parts that are lighter than the secondary color are substituted. Any parts of the image that are darker than the secondary color don't change.

• **Minus** - Subtracts the secondary color from the primary color.

• **Multiply** - Multiplies the primary color by the secondary color. The result is always darker. Blending with black gives black, and with white returns the color unchanged.

• **Over** - This mode is the default. The colors of the two images do not interact in any way, and Nuke displays the full value of the colors in the primary image.

• **Overlay** - Depending on the primary color, multiplies or screens the colors. The secondary color brightens the primary color while preserving highlights and shadows.

• **Plus** - The sum of the two colors. Increases brightness to lighten the primary color and reflect the secondary color. Plus is similar to the Screen blending mode, but produces a more extreme result.

• **Screen** - This is a soft Plus making everything brighter but ramping off the whites. Light colors have more of an effect than dark colors. The result is always a lighter color. Blending with black leaves the pixel unchanged, blending with white always returns white. The result is similar to projecting multiple slides on top of each other.

• **Soft Light** - Depending on the primary color, darkens or lightens the colors. Less extreme than the Hard Light mode.

**Tip:** Note that changing the stack order of your primary and secondary colors might have an impact on your result. For example, if you have two Bezier shapes overlapping each other with a blending mode active, the result depends on which shape is on top of the other. You can change the stack order of strokes/shapes in the stroke/shape list.
Transforming Strokes/Shapes/Groups

To apply spatial transformations to your strokes, shapes, or groups, you can use the controls under the Transform tab. Select a stroke/shape/group from the stroke/shape list and adjust:

- **translate** - to move the stroke/shape on x and y axis.
- **rotate** - to spin a stroke/shape around the pivot point. Use center (or Ctrl/Cmd+drag the transform jack) to position the pivot point.
- **scale** - to resize a spline. Use center (or Ctrl/Cmd+drag the transform jack) to position the pivot point.
- **skew X** - to skew the spline of your stroke/shape along the X axis from the pivot point. Use center (or Ctrl/Cmd+drag the transform jack) to position the pivot point.
- **skew Y** - to skew the spline of your stroke/shape along the Y axis from the pivot point. Use center (or Ctrl/Cmd+drag the transform jack) to position the pivot point.
- **skew order** - to set the order in which skew X and skew Y are applied:
  - XY - Skew X is applied before skew Y.
  - YX - Skew Y is applied before skew X.
- **center x, y** - to set the center for rotation and scaling, adjust the values in center x, y.
- **extra matrix** - enter values you want to get concatenated with the transformation controls above. For more information on concatenating, see How Nodes Concatenate.

**Note:** While you can drag the matrix to another node's matrix to easily use the values elsewhere, you shouldn't try, for example, to drag a 4 by 4 matrix on a 3 by 3 matrix, as doing that might have unexpected results.

Alternatively, you can also use the transform handle (shortcut T) in the Viewer to transform elements. To transform an entire stroke/shape, you need to use the transform handle jack, and to transform points in a stroke/shape, you should use the transform box.

The transform handle appears as a transform jack only when the Transform tab is active, when any of the other tabs in the RotoPaint properties panel are active, the transform handle appears as a box.
To Transform a Stroke/Shape Using a Transform Handle Jack:

1. Click one of the Select tools in the RotoPaint toolbar, with the Transform tab active.
2. Make sure hide transform handles is disabled in the RotoPaint tool settings.
3. Select a stroke/shape by clicking it in the Viewer or by selecting it in the stroke/shape list. A transform handle jack appears.
4. Use the jack to rotate, scale or skew your stroke/shape.

To Transform Points Using a Transform Box:

1. Click the Select All, Select Points, or Select Feather Points tool in the RotoPaint toolbar, with the RotoPaint tab active.
2. Make sure hide transform handles is disabled in the RotoPaint tool settings.
3. Select points in a stroke/shape with Shift+click or by clicking and dragging across the points you want to select. A transform box appears.
4. Use the box to rotate, scale or skew your stroke/shape, or points.
5. To corner pin using the transform box, press Ctrl/Cmd+Shift and drag the transform box points to move them.

**Note:** Transforming points changes the actual point position, while transforming an entire stroke/shape/group changes transformation applied to the point positions.

**Tip:** If you'd like to hide the transform box when moving a selection, enable hide transform handles on move in the RotoPaint tool settings. This may make it easier to correctly position your selection.

To Transform Onion Skin Source

When cloning or revealing, you can use the onion skin control on the RotoPaint tool settings to view and transform your source input on top of your foreground. You can also use onion skinning if you're drawing a stroke/shape with a separate input as the source. To adjust onion skin:
1. With your stroke/shape tool selected, check the **onion** box in the RotoPaint tool settings.
2. Adjust the opacity of the onion skin and transform your source by using the onion skin transform overlay in the Viewer.

Onion skin transform handle.
Adjusting Mask Controls

By default, mask is set to none, but if you want to use a mask, do the following:

1. Check the mask checkbox on the RotoPaint tab. By default, the mask control is hidden, but you can display it by clicking on the black triangle above color.
2. Select a channel to use as a mask from the dropdown menu.
3. If you are using the mask input and want the mask copied into the predefined mask channel, check inject. This way, you can use the mask input again downstream.
4. If necessary, check invert to reverse the mask and/or fringe to blur the edges of the mask.
5. If you find that the overall effect of the RotoPaint node is too harsh, you can also blend some of the original input image back in by adjusting the mix slider.
Editing Shape-Specific Attributes

The RotoPaint properties panel includes a set of controls that you mainly need when you're editing the attributes of a shape. You can find these on the **Shape** tab in the properties panel.

Adding and Removing Feather

To soften the edges of a shape, do the following:

1. With your shape selected, check the **on** box next to the **feather** slider on the **Shape** tab to apply feather to your shape. If you don't want any feather on your shape, uncheck the **on** box.
2. Use the **feather** slider to add outward or inward feather around the whole shape. With positive feather values, your feather effect is outward and, respectively, if your feather values are negative, the feather effect is inward.
3. Use the **feather falloff** slider to adjust the falloff profile. Falloff is measured in pixels. You can also change the type of the falloff using the **falloff type** dropdown menu. Select either **linear**, **smooth0**, **smooth1** and **smooth**. Each of these produces a different rate of falloff that may be helpful for example in matching the soft edge to motion blurred image content.
4. To add feather to a single point, right-click on the point in the Viewer and select **increase feather**. The shortcut for this is **E**. If you press it several times, every press adds more feather.
5. Use the feather handle on the point to add feather into the point. By default, the point angle is locked, and moving the point unlocks the angle.
   You can also select several feather points and move them together.
6. To remove feather from a point, right-click on the point and select **reset feather** or use the shortcut Shift+E.
7. If you disable **feather link** mode (enabled by default) in the RotoPaint tool settings, the feather and shape points move independently.
   - **centered** - to center the shutter around the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 29.5 to 30.5.
   - **start** - to open the shutter at the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 30 to 31.
   - **end** - to close the shutter at the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 29 to 30.
   - **custom** - to open the shutter at the time you specify. In the field next to the dropdown menu, enter a value (in frames) you want to add to the current frame. To open the shutter before the current frame, enter a negative value. For example, a value of -0.5 would open the shutter half a frame before the current frame.
Editing Stroke-Specific Attributes

The RotoPaint properties panel includes a set of controls that you mainly need when you're editing the attributes of a paint stroke. You can find most of these under the Stroke tab in the properties panel.

Selecting a Source Image

Both on the Stroke and RotoPaint tabs, you can set the source control to a specific color or the input you want to pull pixels from for Clone and Reveal brushes. Select:
- **color** - to use a specific color in your stroke/shape.
- **foreground** - to pull pixels from the RotoPaint's bg input, including any strokes/shapes drawn on it. This input is mainly used with cloning.
- **background** - to pull pixels from the bg input, not including any strokes/shapes drawn on it.
- **background 1, background 2 or background 3** - to pull pixels from the bg1, bg2, or bg3 input.

Editing Brush Type

On the Stroke tab, select the type of brush you want to use for the stroke. From the brush type dropdown, select:
- **paint** - to use a normal paint brush.
- **smear** - to use a smear brush on the plate.
- **blur** - to blur your plate with the brush stroke. You can adjust the strength of the blur effect using the effect slider.
- **sharpen** - to sharpen your plate with the brush stroke. You can adjust the strength of the sharpening effect using the effect slider.

Editing Brush Size

On the Stroke tab, you can set the size of the stroke using the brush size slider. You can also tie a stroke's size to pen pressure by checking the size box next to pressure alters.
Editing Brush Spacing

The brush spacing slider adjusts the distance between paint brush dabs. A higher setting increases the space between dabs, creating a dotted line effect when painting. A lower setting decreases the distance and creates a solid brush stroke.

Editing Brush Hardness

On the Stroke tab, you can set the hardness of the stroke using the brush hardness slider.
A low brush hardness value. A higher brush hardness value.

You can also tie a stroke's hardness to pen pressure by checking the hardness box next to pressure alters.

### Adjusting Write On

When you are animating a stroke or a part of it over a range of frames, you can use the write on sliders under the Stroke tab in the properties panel to adjust the order in which the dabs on the stroke appear over these frames. For more information on animating parameters, see the Using the Compositing Environment chapter.

- **write on start** - slide to choose where along the stroke length the paint begins. 0 is the start of the stroke, 1 is the end.
- **write on end** - slide to choose where along the stroke length the paint ends.
Editing Clone or Reveal Attributes

When you are using the Clone or Reveal tool, you can adjust the controls under the **Clone** tab to transform the input that's being cloned or revealed. Adjust:

- **translate** - to move the source image on x and y axis.
- **rotate** - to spin the source image around a pivot point.
- **scale** - to resize the source image by adding or removing pixels. Use **center** (or Ctrl/Cmd+drag the transform jack) to position the pivot point.
- **skew X** - to skew the source image along the X axis from the pivot point. Use **center** (or Ctrl/Cmd+drag the transform jack) to position the pivot point.
- **skew Y** - to skew the source image along the Y axis from the pivot point. Use **center** (or Ctrl/Cmd+drag the transform jack) to position the pivot point.
- **skew order** - to set the order in which **skew X** and **skew Y** are applied:
  - **XY** - **Skew X** is applied before **skew Y**.
  - **YX** - **Skew Y** is applied before **skew X**.
- **round to pixel** - check this to round the **translate** amount to the nearest whole integer pixel. This can help you avoid softening when using filtering. If **source** is set to **color** on the **Stroke** tab, this control is disabled.
- **filter** - to select the appropriate filtering algorithm. For more information, see Choosing a Filtering Algorithm.
- **black outside** - When rotating or translating the clone source, a part of the image area may get cropped. To fill the cropped portion with black, check **black outside**. To fill the cropped portion by expanding the edges of the image, uncheck **black outside**.
- **time offset** - to clone or reveal pixels from a different frame. Time offset is either relative to the current frame (-1 is the frame previous to the current one) or absolute (1 is the first frame in the clip).
- **view** - to select which view you want to clone from in a stereoscopic project.
Editing Existing Stroke/Shape Timing

When editing an existing stroke/shape, you can edit the range of frames during which a stroke/shape is visible in the properties Lifetime tab. The lifetime of a stroke/shape/group is also visible in the Life column in the stroke/shape list. By default, a shape is visible on all frames, whereas a stroke is only visible on a single frame, the frame it was painted on.

<table>
<thead>
<tr>
<th>Icon</th>
<th>Lifetime Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>all frames</td>
<td>Click to make a stroke/shape visible for all frames (the default).</td>
</tr>
<tr>
<td></td>
<td>frame to end</td>
<td>Click to make a stroke/shape visible from the current frame to the last frame.</td>
</tr>
<tr>
<td></td>
<td>single frame</td>
<td>Click to make a stroke/shape visible only on the current frame.</td>
</tr>
<tr>
<td></td>
<td>start to frame</td>
<td>Click to make a stroke/shape visible from the first frame to the current frame.</td>
</tr>
<tr>
<td></td>
<td>frame range</td>
<td>Click to make a stroke/shape visible during a specified range of frames using the frames dialog or from and to controls.</td>
</tr>
</tbody>
</table>
Editing Existing Stroke/Shape Stack Order

When editing strokes/shapes after you've drawn them, you can edit their foreground to background drawing order.

In the stroke/shape list, you can drag and drop strokes/shapes to change their drawing order, and to group them under folders. For more information on using the stroke/shape list, see Working with the Stroke/Shape List.
Editing Existing Stroke/Shape Spline

To edit a stroke/shape's position, you first need to select the stroke/shape in the Viewer or the stroke/shape list. You can then modify the points that make up the stroke/shape's position.

To Add a Point to a Stroke/Shape

1. Select the Add Points tool from the RotoPaint toolbar.
2. Make sure hide points is disabled in the tool settings.
3. In the Viewer, click on the selected stroke/shape to add a new point.

To Move a Point

1. With the Select All tool or Select Points tool active, select the stroke/shape in the Viewer or the stroke/shape list.
2. Make sure hide points is disabled in the tool settings.
3. In the Viewer, drag the points you want to move to a new location.
4. You can also nudge a point using the arrows on your numeric keypad. Nudging a point moves it by one pixel to the direction you choose.

Tip: If you find it difficult to select a single point, you might want to increase the 2D/3D handle size or the handle pick size in the Nuke Preferences dialog (Preferences > Panels > Viewer Handles).

To Move Several Points Together

1. With the Select All tool or Select Points tool active, select the stroke/shape in the Viewer or the stroke/shape list.
2. Make sure hide points is disabled in the tool settings.
3. In the Viewer, drag a marquee around the points (or an entire stroke/shape) that you want to move. If hide transform handles is disabled, a transform box appears.
4. Adjust the transform box as necessary.

Note: Moving a shape with Select All active moves the selected points or feather points to the new location. Scrubbing the timeline shows the shape move across the Viewer between keyframes. However, moving a shape with Select Feather Points active moves only the feather points to the new location, displaying the feather falloff of the selected points.

To Delete a Point

1. Right-click on the Add Points tool and select Remove Points tool.
2. Select the stroke/shape in the Viewer or the stroke/shape list.
3. Make sure hide points is disabled in the tool settings.
4. In the Viewer, click the point that you want to delete.
   OR
   In the Viewer, right-click on the point that you want to delete and select delete.

To Delete an Entire Stroke/Shape

1. In the stroke/shape list, or in the Viewer with the Select All tool, click on the stroke/shape you want to delete.
2. Below the stroke/shape list, click on the minus button (-),
   OR
1. Activate the Select All tool. In the Viewer, click on the stroke/shape.
2. Right-click on the shape and select delete or press the Delete/Backspace key.
Copying, Cutting, and Pasting Stroke Attributes

Copying, cutting, and pasting stroke/shape attributes is achieved in a similar way to affecting position:

To Copy Stroke/Shape Attributes

1. With the Select All, Select Points, or the Select Feather Points tool active, select the stroke/shape in the Viewer.

2. Make sure hide points is disabled in the tool settings.

3. In the Viewer, drag a marquee around the points (or an entire stroke/shape) that you want to copy. If hide transform handles is disabled, a transform box appears.

4. Right-click inside the transform box and select copy > curve (attribute values). The attribute values of the stroke/shape, including position and shape, are copied to the clipboard. Values only apply to a specific keyframe.

To Cut Stroke/Shape Attributes

1. With the Select All, Select Points, or the Select Feather Points tool active, select the stroke/shape in the Viewer.

2. Make sure hide points is disabled in the tool settings.

3. In the Viewer, drag a marquee around the points or click on the stroke/shape for the points that you want to cut. If hide transform handles is disabled, a transform box appears.

4. Right-click inside the transform box and select cut > curve (attribute values). The attribute values of the stroke/shape, including position and shape, are cut from the Viewer and placed in the clipboard. Values only apply to a specific keyframe. Note, that the shape of the cut stroke/shape does not disappear from the Viewer.
To Paste Stroke/Shape Attributes

1. With the Select All, Select Points, or the Select Feather Points tool active, select the stroke/shape in the Viewer.

2. Make sure hide points is disabled in the tool settings.

3. In the Viewer, drag a marquee around the points or click on the stroke/shape for the points that you want to paste. If hide transform handles is disabled, a transform box appears.

4. Right-click inside the transform box and select paste > spline (attribute values).
   This pastes the copied stroke/shape attribute values that were on the clipboard. If you have copied another type of curve, for example, (values) or (animations), then all options except for that one are disabled.
Point Cusping, Smoothing, Expressions, and Links

Roto and RotoPaint allow you manipulate curves, by cusping or smoothing the points on the curve, add expressions, and link controls to points.

To Cusp or Smooth Points

You can cusp points on a shape to create sharp corners, and smooth points to replace sharp corners with curved lines.

1. Select the shape you want to edit in the Viewer or the stroke/shape list.

2. Select the Smooth Points tool or Cusp Points tool in the RotoPaint toolbar, depending on whether you want to cusp or smooth your points.

3. Make sure hide points is disabled in the tool settings.

4. In the Viewer, click on the point that you want to cusp or smooth.

5. With your point selected, you can also use the shortcut keys Z and Shift+Z to smooth and cusp it respectively.
   OR
   Right-click on the point you want to smooth or cusp, and select smooth or cusp/de-smooth.

To Add Expressions to Points

1. With the Select All , Select Points , or Select Feather Points tool active, make sure hide points is disabled in the tool settings.

2. Select a point in your stroke/shape. Right-click and select add expression.

3. Enter your expression values to the fields in the Expression dialog. Alternatively, you can Ctrl/Cmd+drag expression values from another node to the point in your stroke/shape.
   You can add expressions to edit your point's location or the shape of the point’s feather effect. For more information on Expressions see Expressions.
To Link Other Controls to a Point’s Position

You can copy a point link (that is, a point’s x and y position) and use it in other nodes.

To copy a point link:

1. With the Select All, Select Points, or Select Feather Points tool active, make sure hide points is disabled in the tool settings.
2. Right-click on a point and select copy > point link.
3. Go to the field you want to use the link in, for example another node, right-click and select Copy > Paste.

For example, you can copy a point link, click on a Transform node's animation menu next to translate, and select Copy > Paste. An expression arrow appears between the RotoPaint node and the Transform node, and when you move the point, the Transform node's translate values update with your changes.

Note: When copying, cutting, or pasting from the right-click menu, numbers are displayed next to the curve, spline, or point. These numbers represent how many of each you have selected and thus, how many are copied or pasted. For example, 2 point (values) denotes that 2 points are selected. Pasting two points into a shape with many more points replaces the lowest two points in the shape, or the highest two splines in the stroke/shape list.
Animating Strokes/Shapes

All strokes/shapes that appear on more than one frame can be animated. By default, the autokey option in the RotoPaint tool settings is on, which means your changes to a stroke/shape automatically creates keyframes and animates your stroke/shape. You can also access all the curves and shapes in the Curve Editor and Dope Sheet.

To animate a stroke/shape using autokey

1. Draw a stroke/shape that appears on more than one frame. By default, the autokey option in the RotoPaint tool settings is selected and a keyframe is automatically created in the first frame your stroke/shape appears.
2. Move to a new frame.
3. With one of the Select tools, select the points or the stroke/shape you want to animate.
4. Adjust the points in your stroke/shape position or change the stroke/shape's attributes as necessary. A new keyframe is automatically set. The frame marker on the timeline turns blue to indicate the selected stroke/shape is animated.
5. Repeat steps 2, 3 and 4 for all the frames you want to set as key frames.

Tip: Note that if you are translating an entire stroke/shape using the Transform tab, RotoPaint also draws a track of the stroke/shape's animated position. You can, for example, use the track in another node (such as Tracker or CameraTracker) by Cmd/Ctrl+dragging the values from the translate animation button to an appropriate field in another node.

To animate strokes/shapes manually

If you choose to switch the autokey function off, you can still create key frames manually. You can set key frames to the entire stroke/shape, or the stroke/shape's spline, transformation or attributes.

1. Move to the frame where you want to create a keyframe and select your stroke/shape.
2. Do one of the following:
   • If you want to create a key that is set to animate the entire stroke/shape, right-click on the stroke/shape and select set key > all.
• If you want to create a key that is set to animate a position, right-click on the stroke/shape and select **set key > spline**.

• If you want to create a key that is set to animate transformation, right-click on the stroke/shape and select **set key > transform**.

• If you want to create a key that is set to animate attributes, right-click on the stroke/shape and select **set key > attributes**.

If you have **autokey** turned off, you can only adjust a point in a stroke/shape at a keyframe. In other words, in order to make changes to a point, you either have to move to an existing keyframe on the timeline, or you need to create a new keyframe first.
Viewing Spline Keyframes

You can use the **spline key** controls on the **RotoPaint** tab to view whether there are keyframes set for the spline of your stroke/shape. Do the following:

1. Select a stroke/shape on the stroke/shape list.
2. If there are spline keys set on your stroke/shape, the key boxes are highlighted blue and display where you are currently on the timeline with regard to the keyframes that are set already. You can move backwards and forwards between the keyframes using the arrow keys ⚙️ ⏯️.
3. If you want to add or remove spline keys for the stroke/shape in a selected frame, use the **Add** ⚕️ and **Delete** ⏥️ buttons.

To View Keyframes on the Timeline

You can view different types of keyframes you’ve set, either automatically or manually, on the timeline. If you’ve set keyframes on the **RotoPaint** tab, these are visible on the timeline when you have the **RotoPaint** tab open in the properties panel. Similarly, if you’ve created transformation keyframes on the **Transform** tab, you can see those keyframes on the timeline when you have the **Transform** tab open.

- If you want to delete a key that is set to animate a position, right-click on the stroke/shape and select **delete key > spline**.
- If you want to delete a key that is set to animate a transformation, right-click on the stroke/shape and select **delete key > transform**.
- If you want to delete a key that is set to animate attributes, right-click on the stroke/shape and select **delete key > attributes**.

If no other keys are set for the selected stroke/shape, the frame marker on the timeline turns from blue to the default color.
Deleting or Rippling Keyframes

You can delete keyframes individually or by stroke/shape, or use the ripple function to change a keyframe and apply that same relative adjustment to the point across a range of frames.

To Delete a Keyframe
1. Using the **Viewer** timeline, scrub to the frame where you want to delete a keyframe.
2. In the stroke/shape list, select the stroke/shape whose key you want to delete.
3. Do one of the following:
   - If you want to delete a key that is set to animate the entire stroke/shape, right-click on the stroke/shape and select **delete key > all**.
   - If you want to delete a key that is set to animate a position, right-click on the stroke/shape and select **delete key > spline**.
   - If you want to delete a key that is set to animate a transformation, right-click on the stroke/shape and select **delete key > transform**.
   - If you want to delete a key that is set to animate attributes, right-click on the stroke/shape and select **delete key > attributes**.

If no other keys are set for the selected stroke/shape, the frame marker on the timeline turns from blue to the default color.

To Delete All Keyframes for a Stroke/Shape
You can also delete all key frames you've set for a stroke/shape at one go. Do the following:
1. Select the stroke/shape from which you want to delete key frames in the Viewer.
2. Right-click on the stroke/shape and select **no animation > all, spline, transform or attribute** depending on whether you want to delete all key frames for that stroke/shape or only key frames animating shape, transform or attributes. All key frames are removed from the stroke/shape you selected.

To Ripple Keyframes
Rippling keyframes allows you to adjust the position of a stroke/shape point on one frame and have that same relative adjustment applied to the point across all frames or a specified range of frames.
This way, you can make non-animated changes to a stroke/shape which is being animated over a set of frames.

1. Activate **Select All** tool in the RotoPaint toolbar and click the **ripple edit** button in the RotoPaint tool settings. A red border displays around the Viewer to indicate that the ripple mode is active. In the dropdown menu next to **ripple edit**, select:
   - **all** - to ripple all frames in your sequence.
   - **from start** - to ripple frames from the first frame to the current frame.
   - **to end** - to ripple frames from current frame to the last frame.
   - **range** - to ripple a particular range of frames.

2. Select the stroke/shape you want to edit from the stroke/shape list.
3. Make the necessary changes to the stroke/shape in the Viewer.
Copying, Cutting, and Pasting Animations

Copying, cutting, and pasting can save you time, but can be destructive. For example, pasting over shapes destroys all points and keyframes of the target shape in order to replace it.

To Copy Animations

If you want to copy a shape, including all animation attributes, animated points, and associated keyframes, copying animations is the best way to do this. To copy animations for a stroke/shape:

1. With the Select All tool active, select the stroke/shape in the Viewer.
2. Right-click on the stroke/shape and select copy > curve (spline-key animations).
   This copies the current selected animation and keyframe information to the clipboard.

To copy only curve attribute information for animated shapes:

1. With the Select All tool active, select the stroke/shape in the Viewer.
2. Right-click on the stroke/shape and select copy > curve (attribute animations).
   This copies only the selected attributes for the animated shape to the clipboard.

To copy only the point information for animated shapes:

1. With the Select All tool active, select the stroke/shape in the Viewer. Make sure hide points is disabled in the tool settings.
2. Right-click on the point or points and select copy > points (animations).
   This copies only the selected animated points and keyframes to the clipboard.

To Cut Animations

If you want to move a stroke/shape and replace another with all the same animated points and keyframes, cutting the shape is a better option than simply copying it. To cut animations for a stroke/shape:

1. With the Select All tool active, select the stroke/shape in the Viewer.
2. Right-click on the stroke/shape and select **cut > curve (spline-key animations)**.
   The shape is cut from the Viewer and placed in the clipboard. The transform box still appears on the Viewer and allows you to amend the shape, until you click on another shape or elsewhere in the Viewer.

To cut only animation attributes for a stroke/shape:

1. With the **Select All** tool active, select the stroke/shape in the Viewer.
2. Right-click on the stroke/shape and select **cut > curve (attribute animations)**.
   This cuts the animation attributes for the selected stroke/shape and placed it them in the clipboard.

To cut only the animated points for strokes/shapes:

1. With the **Select All** tool active, select the animated point(s) of a stroke/shape in the Viewer that you want to cut. Make sure **hide points** is disabled in the tool settings.
2. Right-click on the point(s) and select **cut > point (animation)**.
   This cuts only the animated points and keyframes and places them in the keyboard.

**To Paste Animations**

If you want to paste copied or cut animated shape information, consider whether the shape you are pasting over has values that need to be saved. Pasting animated points, spline-keys, and attributes over another shape destroys all points and keyframes of the second shape in order to replace it. To paste animations for a stroke/shape:

1. With the **Select All** tool active, select the stroke/shape in the Viewer that you want to paste the animated shape over. If there are not the same number of points on the shape as on the copied shape, pasting the animated shape may not paste as expected.
2. Right-click on the stroke/shape and select **paste > spline (animations)**.
   This pastes the animation and keyframe information for the copied or cut stroke/shape over the previously selected shape. This shape may need to be moved away from the copied shape, in order to see both.

To paste only animation attributes for a stroke/shape:

1. With the **Select All** tool active, select the stroke/shape in the Viewer that you want to paste the animated attributes over.
2. Right-click on the stroke/shape and select **paste > spline (attribute animations)**.
This pastes the animation attributes for the copied or cut stroke/shape over the previously selected shape's attributes.

To paste only the animated points for strokes/shapes:

1. With the **Select All** tool active, select the point(s) of a stroke/shape in the Viewer that you want to paste the animated points over. Make sure **hide points** is disabled in the tool settings.
2. Right-click on the point(s) and select **paste > point (animation)**. This pastes only the animated points and keyframes from the clipboard.
Adding Motion Blur

After animating your shapes, you can apply motion blur to individual shapes in the shape list or to the all shapes in the current node using the controls in the properties Motion Blur tab.

Adding Blur to Shapes

Shape Blur determines the exposure for each moving shape and blends the resulting blurred shapes. This may be more efficient than the global motion blur since each shape will only be blended once.

Note: Shape motion blur may result in artifacts when shapes blur over the same region.

To apply shape motion blur:
1. Select the target shapes in the shape list or in the Viewer.
2. In the properties panel, select the Motion Blur tab and make sure that shape mode is enabled.
3. Check the on box next to the motionblur field to apply motion blur to your shape. If you don’t want any motion blur on your shape, leave the on box unchecked.

Tip: You can also toggle motion blur on and off in the shape list by clicking in the motion blur column for each shape.

4. In the Shape Blur > motionblur field, enter the sampling rate. This affects the number of times the input is sampled over the shutter time. The higher the rate, the smoother the result. In many cases, a value of 1.0 is enough.

A motionblur value of 0.5 produces

A higher value, in this case 2.0,
5. In the **shutter** field, enter the number of frames the shutter stays open when motion blurring. For example, a value of 0.5 would correspond to half a frame. Increasing the value produces more blur, and decreasing the value less.

![Shutter Value Examples](image)

A shutter value of 0.2 produces less blur. A higher value, in this case 1.0, produces more blur.

6. Adjust the shutter offset using the **shutter offset** dropdown menu. The different options control when the shutter opens and closes in relation to the current frame value. Select:

- **centered** - to center the shutter around the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 29.5 to 30.5.
- **start** - to open the shutter at the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 30 to 31.
- **end** - to close the shutter at the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 29 to 30.

![Shutter Offset Examples](image)

Centering the shutter around the current frame. Opening the shutter at the current frame. Closing the shutter at the current frame.

- **custom** - to open the shutter at the time you specify. In the field next to the dropdown menu, enter a value (in frames) you want to add to the current frame. To open the shutter before the current frame, enter a negative value. For example, a value of -0.5 would open the shutter half a frame before the current frame.
Adding Global Motion Blur

**Global Blur** correctly accounts for interaction between motion blurred shapes. This may be more expensive than the **shape** motion blur since it may blend each shape for every sample, in the same way as TimeBlur. See **Applying a TimeBlur Filter** for more information.

Global motion blur requires that shutter and sampling parameters are the same for all shapes and has been optimized for consecutive shapes with the same properties using the **over** blend mode.

**Note:** Global motion blur overrides per-shape motion blur, applying your settings to all shapes in the current node's shape list.

To add **global** motion blur:

1. In the properties panel, select the **Motion Blur** tab and make sure that **global** mode is enabled.
2. In the **Global Blur > motionblur** field, enter the sampling rate. This affects the number of times the input is sampled over the shutter time. The higher the rate, the smoother the result. In many cases, a value of 1.0 is enough.

   ![Motion Blur Samples](image)

   A motionblur value of 0.5 produces less samples.  
   A higher value, in this case 2.0, produces a smoother blur.

3. In the **shutter** field, enter the number of frames the shutter stays open when motion blurring. For example, a value of 0.5 would correspond to half a frame. Increasing the value produces more blur, and decreasing the value less.
A shutter value of 0.2 produces less blur.

A higher value, in this case 1.0, produces more blur.

4. Adjust the shutter offset using the **shutter offset** dropdown menu. The different options control when the shutter opens and closes in relation to the current frame value. Select:

- **centered** - to center the shutter around the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 29.5 to 30.5.

- **start** - to open the shutter at the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 30 to 31.

- **end** - to close the shutter at the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 29 to 30.

- **custom** - to open the shutter at the time you specify. In the field next to the dropdown menu, enter a value (in frames) you want to add to the current frame. To open the shutter before the current frame, enter a negative value. For example, a value of -0.5 would open the shutter half a frame before the current frame.
Viewing Points in the Curve Editor and the Dope Sheet

You can edit your stroke/shape points, their feather points and tangents in the Curve Editor and Dope Sheet. To read more about using the Curve Editor and Dope Sheet, see Animating Parameters.

1. Select your stroke/shape, and right-click on a point.
2. Select curve editor... and select:
   - points - to add your point to the Curve Editor
   - points+feathers - to add your point and its feather point to the Curve Editor
   - points+tangents - to add your point and its tangent to the Curve Editor
   - all - to add your point, feather point, and tangent to the Curve Editor.
3. Viewing your points in the Dope Sheet is very similar, just right-click on your point and select one of the same options under dope sheet...
Copying, Pasting, and Cutting Stroke Positions

After creating a stroke/shape, you can copy, paste, and cut its position to use the same shape in other strokes/shapes. Copying, cutting, or pasting values applies to only one keyframe.

To Copy Stroke/Shape Positions

You can copy the stroke/shape position values for an entire selected shape. This enables you to use the same values for another shape in the Viewer. Unlike cutting stroke/shape values, described below, copying position values does not delete any keys set. To copy a stroke/shape:

1. With the Select All tool active, select the stroke/shape in the Viewer that you want to copy.
2. Right-click on the stroke/shape and select copy > curve (spline-key values).
   This copies the position and point values into the clipboard.

To Cut Stroke/Shape Positions

You can cut the stroke/shape position values for an entire selected shape. This enables you to use the same values to replace another shape in the Viewer. To cut a stroke/shape:

1. With the Select All tool active, select the stroke/shape in the Viewer that you want to copy.
2. Right-click on the stroke/shape and select cut > curve (spline-key values).
   This cuts the position and point values and places it into the clipboard. Cutting the position deletes the keyframe, or, if there is no keyframe, deletes the points or spline.

Note: To cut stroke/shape position values, you must be cutting the stroke/shape when it is on a keyframe, unless it has no keyframe. You cannot cut an animated stroke/shape not currently on a keyframe.
To Paste Stroke/Shape Positions

You can paste copied or cut stroke/shape position values from the clipboard onto another stroke/shape. This enables you to create a copy, or replacement, shape in the Viewer. To paste a copied or cut shape:

1. With the **Select All** tool active, select the stroke/shape in the Viewer that you want to copy.
2. Right-click on a selected stroke/shape and select **paste > spline (values)**.
   This pastes the copied or cut position and point values from the clipboard onto another shape. In the case of copied shapes, the new shape may be created over the original. Simply select and drag to move it away from the original shape. Spline key values must have the same number of points between the copied or cut stroke/shape and the stroke/shape you are replacing.
Copying, Pasting, and Cutting Point Positions

You can copy point position values in a stroke/shape you have selected. This enables you to use the same values for another point in a stroke/shape. Unlike cutting point values, described below, copying position values does not delete any keys set.

**Note:** When copying, cutting, or pasting from the right-click menu, numbers are displayed next to the curve, spline, or point. These numbers represent how many of each you have selected and thus, how many are copied or pasted. For example, **2 point (values)** denotes that 2 points are selected. Pasting two points into a shape with many more points replaces the lowest two points in the shape, or the highest two splines in the stroke/shape list.

**To Copy Point Values**

1. In the Viewer, scrub to the frame that contains the stroke/shape whose point position you want to copy.
2. Activate the **Select Points** tool.
3. Make sure **hide points** is disabled in the tool settings.
4. Right-click on the point a stroke/shape and select **copy > point (values)**.
   Nuke copies the positions of the selected point to the clipboard. Any keys set to animate these positions are not affected.
Pasting Point Positions

You can paste any position you've copied/cut from another point to a selected point in a stroke/shape. If you have autokey turned on, this also sets a keyframe at the current frame to animate this point. The attributes of the strokes or any keys set to animate the attributes are not affected.

To Paste Point Values

1. In the Viewer, scrub to the frame that contains the stroke/shape to which you want to paste the positions on the clipboard.

2. Activate the Select Points tool.

3. Make sure hide points is disabled in the tool settings.

4. Right-click on the point and select paste > point (values).
   Nuke pastes the positions (but not any attributes) on the clipboard to the selected point and, if you have autokey turned on, sets the current frame as a keyframe.

Note: If you have copied more than one point into the clipboard but have only selected one point to paste into, paste > point (values) only displays one point and may not look as you expect. To paste both points, ensure that you have two points selected (or however many is needed for your sample).
Cutting Point Positions

You can cut the position values of a point in a stroke/shape. Cut also copies the positions at the current frame to the clipboard. The point attributes or any keys set to animate the attributes are not affected.

To Cut Point Values

1. In the Viewer, scrub to the frame that contains the stroke/shape whose point position you want to cut.
2. Activate the Select Points tool.
3. Make sure hide points is disabled in the tool settings.
4. Right-click on the point and select cut > point (values).

Nuke deletes any keys set to animate the positions of the selected point, and copies the position to the clipboard. Cutting the position deletes the keyframe, or, if there is no keyframe, deletes the points or spline.

Note: To cut stroke/shape point values, you must be cutting the stroke/shape when it is on a keyframe, unless it has no keyframe. You cannot cut an animated stroke/shape not currently on a keyframe.
RotoPaint and Stereoscopic Projects

If you need to use RotoPaint in a multi-view or stereoscopic project, you may want to draw your strokes/shapes or apply your edits to one view only (for example, the left view but not the right), or create a stroke/shape in one view and have it automatically generated for the other in the correct position.

For more information on multi-view or stereoscopic projects in Nuke, see Stereoscopic Scripts.

Selecting the View to Draw On

When you are using the RotoPaint node to draw a new stroke/shape in a stereoscopic or multi-view project, you can toggle the singleview checkbox in the RotoPaint tool settings to draw your stroke/shape on one view only or multiple views. For existing strokes/shapes/groups, you can use the view control to select the view the stroke/shape is visible. If you’re working on a stereoscopic project, the view you’re using for a particular stroke, shape, or group is also visible on the View column in the stroke/shape list.

Selecting the View to Clone From

When cloning, you can select the view to use as the clone source. Go to the Clone tab and set the view dropdown menu to the view you want to clone from. To use the view currently displayed in the Viewer, select current.
Reproducing Strokes/Shapes in Other Views

To create a stroke/shape on one view and have it automatically generated for the other:

1. Make sure there is a disparity field upstream from the image sequence you want to paint on. If the image sequence is an .exr file, the disparity field can be included in its channels. Otherwise, you can use a ShuffleCopy node or Ocula’s O_DisparityGenerator plug-in to add it in the data stream.

2. In the RotoPaint properties, check all the views in the view dropdown menu. Display the view you want to paint on in the Viewer.

3. Draw a stroke/shape in the Viewer.

4. Select the stroke/shape in the stroke/shape list in the RotoPaint node properties.

5. Right-click the stroke/shape and select either:
   - Correlate points - to use the disparity at each point of the stroke/shape and translate each point to the corresponding position in the other view.
   - Correlate average - to take the disparity at each point, calculate the average disparity of the stroke/shape and then translate it to the corresponding position in the other view.

   The Correlate dialog displays.

6. In the Correlate dialog, select how to correlate the views in the correlate from dropdown. For example, if your stroke/shape was in the correct position in the left view but not the right, you’d set correlate from to left to right.

   This adds the disparity vectors in the map to the current values, creating the corresponding stroke/shape for the other view.

7. In the Viewer, switch between the two views to compare the original and the correlated strokes/shapes.

8. If you want to adjust the stroke/shape further, you need to adjust both views independently. Adjustments you make to one view are not automatically generated for the other.
Editing Strokes/Shapes in One View Only

Splitting a view off allows you to edit the stroke/shape data in that view only, without affecting any other views that exist in your project settings.

To Edit Stroke/Shape Points
1. Display the view you want to edit in the Viewer.
2. Right-click on a spline or point and select split off [view], for example [split off left].
   Any changes you now make to the stroke/shape points are only applied to the view you chose to split off and are displaying in the Viewer.
   Views that have not been split off can still be edited together. For example, if you have views called left, middle, and right, and choose to split off the left view, any changes to the middle view are also applied to the right view and vice versa.
3. To unsplit a view, right-click on a spline or point again and select unsplit [view].
   The view is unsplit, and all changes you made after splitting it off are lost.

   **Note:** To see splines for paint strokes in the Viewer, you need to activate show paint stroke splines in the RotoPaint tool settings.

   **Tip:** You can also split shapes off by right-clicking on them in the stroke/shape list and selecting Split off [view], for example Split off left.

To Edit Stroke/Shape Attributes
1. Display the view you want to edit in the Viewer.
2. In the RotoPaint controls, click the Views menu button next to the control you want to edit and select Split off [view name]. For example, to edit opacity in a view called left, select Split off left from the Views dropdown menu next to opacity.
   Any changes you now make to opacity are only applied to the view you chose to split off and are displaying in the Viewer.
Views that have not been split off can still be edited together. For example, if you have views called left, middle, and right, and choose to split off the left view, any changes to the middle view are also applied to the right view and vice versa.

You can also click on the small arrow on the left side of the control to see the values for each view.

3. To unsplit a view, click the Views menu button and select Unsplit [view]. For example, to unsplit a view called left, you'd select Unsplit left.

The view is unsplit, and all changes you made after splitting it off are lost.
Applying a Stereo Offset

The **stereo offset** control in the RotoPaint properties allows you to move the selected stroke/shape on the x and y axes. This is an extra transform that is applied after all other transforms. Typically, you would position the stroke/shape correctly in the hero view, then split **stereo offset**, and drag the stroke/shape to its correct location in any other views. Note that you can also press **Shift** while dragging to constrain the movement to the x or y axis only.

**Stereo offset** can be useful, for example, if you have a stroke/shape that is correctly positioned in one view and you want to move it to its correct location in another view, but can't use the **translate** control on the **Transform** tab because that's being driven by Tracker data.
Where Are the Bezier and Paint Nodes?

The pre-6.0 Bezier and Paint nodes have been deprecated in favor of the RotoPaint node. With RotoPaint you have the ability to add more strokes and shapes, group them, and so on. However, Bezier and Paint are still in the application for backwards-compatibility with old scripts. Should you find the need (or just feel nostalgic), you can create the Paint and Bezier nodes in a couple of easy ways:

• Press X on the Node Graph, make sure Tcl is checked in the dialog that opens, enter Paint or Bezier in the Command field, and click OK. Your node appears in the Node Graph.

• You can add a Paint or a Bezier node to your Toolbar menu with a statement in your menu.py file like the following:

  • For Bezier:
    ```python
tb = nuke.toolbar("Nodes")
tb.addCommand("Draw/Bezier", "nuke.createNode('Bezier')", icon="Bezier.png")
    ```
  • For Paint:
    ```python
tb = nuke.toolbar("Nodes")
tb.addCommand("Draw/Paint", "nuke.createNode('Paint')", icon="Paint.png")
    ```
Tracking \text{and Stabilizing}

Nuke’s 2D Tracker allows you to extract animation data from the position, rotation, and size of a pattern. You can then apply the data directly to transform or match-move another element. Or you can invert the data values and apply them to the original element to stabilize the image.

Before Tracking

Before you track, it's important to play back the image several times. This will help you identify the best features for the process, as well as any problems with motion blur, occlusions, or features moving out of frame.

For some images, you may need to apply filters or color-correction to boost the visibility of features before you attempt to track them. Because of the procedural nature of Nuke, you can disable these extra nodes after you get a successful track, or simply reconnect the Tracker node at the appropriate place to apply the transform.

Quick Start

Here's a quick overview of the workflow:

1. Connect a Tracker node to the image you want to track. See Connecting the Tracker Node.
2. Place the required number of track anchors on features in the image. See Adding Track Anchors.
3. Use Automatic Tracking for simple tracks or, for greater accuracy or difficult tracks, use Keyframe Tracking.
4. Apply your tracking data depending on requirements. See Applying Tracking Data.
Connecting the Tracker Node

To connect the Tracker node:
1. Read in and select the clip you want to track.
2. Click Transform > Tracker.
3. Click Image > Viewer to insert a Viewer node and connect it to the Tracker node.
Adding Track Anchors

You can add as many track anchors as required, depending on which transformational components you wish to track. For example, when tracking in areas of distortion or noise, it's a good idea to add a lot of tracking anchors and then average the results to get a better overall track.

1. Enable the fast-add button and click in the Viewer to add tracking anchors, or click add track on the Tracker tab in the properties panel to create the required number of anchors.

   **Note:** Holding Ctrl/Cmd+Alt and clicking in the Viewer also enables fast-add mode.

New anchors always appear in the center of the current Viewer. You'll also notice an anchor zoom window in the top left of the Viewer. This allows you to accurately position tracking anchors without zooming the entire Viewer.

2. Temporarily disable tracks using the checkbox on the left of the panel or remove tracks by selecting them in the Tracks list and clicking delete tracks.

Positioning Track Anchors

Each track anchor consists of a pattern and search area:

- **Pattern** - the target pixels tracked across multiple frames. This pattern should be as distinct as possible from the surrounding frame and remain visible throughout the majority of the sequence.
- **Search** - the area of the current frame in which the pattern is likely to be found in the next frame. Large search areas can affect performance.
1. Drag the crosshair in the center of the anchor over the pattern to be tracked or manually adjust the track_x and track_y fields in the Tracks list. In the example, the corner of a building is directly under the crosshair.

   **Note:** You can enable Settings > snap to markers to assist you when placing anchors.

   **Note:** Whether you select blobs or corners, a guide displays in the Viewer as you drag the anchor, highlighting likely patterns.

   ![Pattern area](image)

   Release the mouse to snap the anchor to the marker.

2. Adjust the pattern and search areas as required using the following methods:
   - Click and drag the boundaries to adjust on a single axis. For example, dragging a vertical boundary adjusts the area on the x axis.
   - Click and drag the corners of the boundaries to adjust on both axes. Hold Shift, click and drag to scale the area.

   **Note:** Hold Cmd/Ctrl to adjust boundaries asymmetrically, that is, irrespective of the anchor position.
3. Reset the pattern and search areas by clicking in the Viewer tools.
Tracking Preferences and Viewer Tools

Before tracking, use the Tracker’s properties panel and Viewer tools to control Viewer output and determine tracking behavior.

1. Click the Settings tab to display the General controls.
2. Select the channels in the image to track using the track channels dropdown.
   If this is set to anything other than all or none, use the checkboxes to select the required channels.
3. Select the pre-track filter type to apply before pattern comparison using the dropdown:
   - none - no filter is applied.
   - adjust contrast - the default filter, stretches the image contrast to better suit the tracking algorithm. This is the recommended setting and shouldn't need changing in most circumstances.
   - median - helps to remove noise from the pattern.
4. Check adjust for luminance changes to force Tracker to calculate some extra pre-filtering to compensate for changes in brightness. This option slows the tracking process and can reduce the accuracy of tracks, so only enable this control if there are known changes in brightness.

   Note: Enabling adjust for luminance changes can occasionally produce better tracks on shots with no differences in luminance, particularly on jittery shots where sub-pixel accuracy is vitally important.

5. Clamp super-white, sub-zero footage is enabled by default, restricting the pixel values within the pattern between 0-1.
   If you want to track the sequence using its full dynamic range, disable this control and increase max_error to a value large enough to cover the range. For example, a sequence with pixel values of 40 might require a similar max_error value.
6. Enable show error on track paths to color code tracks showing their pattern matching error values, green for a good match through to red for a poor match.

   Note: The Viewer toolbar button also toggles this control on and off.

7. Enable hide progress bar to free up Viewer real-estate while tracking.
8. Enable **snap to markers** to assist you when placing tracking anchors. See **Positioning Track Anchors** for more information.

9. Set when the zoom window is displayed using the **show zoom window** dropdown:
   - **always** - the window is always displayed.
   - **on track move** - the window is only displayed when a track anchor is moved.
   - **when tracking** - the window is only displayed during tracking.
   - **when tracking or track change** - the window is displayed during tracking and when a tracking anchor is moved.
   - **never** - the window is never displayed.

10. Set the zoom widget's size and magnification using the **zoom size/mag.** dropdowns. You can select **custom** and enter values manually for greater flexibility.

11. A filter is applied to the zoom window on playback by default, but you can enable the filter permanently, or disable it, using the **zoom window filter** dropdown.

    ![Note:](image) The filter applied is the same as that selected on the **Transform** tab, and can produce a more visually stable track. It can make track positioning more difficult, however.

12. Proceed to **Automatic vs. Keyframe Tracking** to determine which tracking method suits your needs.
Automatic vs. Keyframe Tracking

After placing track anchors and setting your preferences, you’re ready to calculate your tracks. Nuke’s Tracker provides two calculation methods:

• **Automatic Tracking** - ideal for simple tracks, there are no extra preparation steps once you’ve set your tracking anchors and preferences.
• **Keyframe Tracking** - a more involved method, requiring you to set keyframes on the sequence in order to calculate tracks. Keyframe tracking may be the better option for more complicated patterns and movement.
Automatic Tracking

Calculating tracks automatically uses the tools above the Viewer to control the direction and range of frames to track. Tracking backwards can produce a better track than going forwards if the pattern is clearly visible later in the clip. By default, Auto-Tracking grabs the reference pattern on the first frame of the sequence, from within the pattern anchor, using this pattern throughout the track for comparison between frames.

To help avoid clutter in the Viewer, you can enable or disable the Tracker overlay by right-clicking in the Viewer and selecting **Overlay**, or by pressing **Q** to toggle between the available states:

- overlay off
- overlay on
- overlay on, no animation path

**Calculating Auto-Tracks**

1. In the Tracker properties panel, select each track you wish to calculate or click **select all** to use all tracks.
2. For each track, select the type of movement the track is intended to output: translation, rotation, or scaling. For example, tracking a feature across the sequence and toward the camera may involve translation and scale.

   ![Note: These controls deal with output from tracking data using the Transform controls and are not the same as the Settings > warp type control, which deals with pattern recognition.]

3. Using the tool bar above the Viewer, click either the frame backward (X) or forward (C) buttons to move to the previous or next frame. Move through a few frames in this manner to ensure that all enabled track anchors are “sticking” to their patterns.

   If a particular track anchor doesn’t stick, experiment with a different position.

4. Once all track anchors stick, click the track backward (Z) or track forward (V) buttons to analyze the whole sequence.

5. To track only a certain frame range, use the range buttons to enter the required frames.

6. Click stop (Esc) to cease tracking in either direction.
Tip: When calculating multiple tracks simultaneously, you may find that some tracks stick accurately to the pattern, while others require resetting and reanalysis. When you're happy with a given track, deselect it in the Tracks list. This protects it from recalculation, and lets you experiment with better placement for the wayward tracks.

See Troubleshooting Auto-Tracks for help with troublesome tracks.

Troubleshooting Auto-Tracks

No matter how sophisticated tracking becomes, some sequences are inevitably going to cause problems. There are a number of pre-tracking checks you can perform to assist Auto-Tracking:

• Play through the sequence before placing your tracking anchors
• Look for features that are consistent throughout the majority of the sequence
• Avoid occluded features where possible - see Dealing with Occlusions.

You can also adjust the way patterns are tracked and how often they are re-sampled, or grabbed, using the Settings tab and Auto-Tracking controls:

1. Try adjusting the max iterations, epsilon, and max_error controls to improve the track accuracy:
   • max iterations - the maximum number of iterations before the tracking algorithm stops searching for features.
   • epsilon/resolution - the error level at which Tracker is assumed to have found the feature - no further search for a better match is performed. Higher values may result in a faster but less accurate track.
   • max_error - the error level at which Tracker stops searching for features.

2. In the Auto-Tracking sub-menu, enable predict track to use the animation path to predict where the pattern may appear in the next frame.

Note: If the track fails when prediction is enabled, click the Clear Forward button, or re-tracking follows the same erroneous path.

3. Set the type of movement Tracker should expect in the pattern using the warp type dropdown:
   • Translate
   • Translate/rotate
   • Translate/Scale
   • Translate/rotate/Scale
   • Affine
Translation only is the easiest to calculate, but can lose the pattern if it rotates or scales. Affine can be used to attempt to preserve straight lines and relative distances, compensating for sheering.

4. Try adjusting the pattern grabbing behavior, when or how often a new pattern should be grabbed from the sequence:
   - **on first frame** - the comparison pattern is grabbed on the first frame from with the pattern anchor. You might select this option if the feature translates but doesn't rotate, scale, or shear.
   - **every frame** - the comparison pattern is grabbed at every frame in the sequence. This option takes longer to process, but can produce a smoother track.
   - **every n frames** - allows you to set the frame interval between pattern grabs using the every n frames control.
   - **if error above** - the comparison pattern is grabbed when the error value is greater than that specified by the when error > control. Setting this control to a low value grabs the pattern more often.
   - **if error below** - the comparison pattern is grabbed when the error value is less than that specified by the when error < control.
   - **custom** - this option enables all the pattern grab behavior controls, allowing you to fine-tune when the comparison pattern in grabbed through out the sequence.

5. Enable **when tracking is stopped** to cause Tracker to re-grab the pattern at the current frame when tracking stops.

6. Enable **when tracker is moved** to cause Tracker to re-grab the pattern at the current frame when a tracking anchor is moved.

**Dealing with Occlusions**

Tracker's offset capability allows you track an obscured feature using the relative position of another feature, providing that the distance between the two points remains constant.

1. Track the pattern normally until the occlusion causes Tracker to fail.
   The zoom window helps to identify the failure point.
2. Play though the sequence to identify a likely offset point - a pattern that remains equidistant from the original pattern grab.

3. Hold down Ctrl/Cmd and drag the tracking anchor to the offset position.

   The offset amount is recorded in the Tracks list and highlighted in yellow in the Viewer.

4. Continue tracking as normal by clicking the track backward (Z) or forward (V) button. Tracker combines the two tracks into a single continuous track.

5. Use the clear backward and forward buttons to clear poor keyframes. Click clear all to remove all selected tracks and keyframes, excluding manually placed keyframes.

   Note: You can reset tracking anchor pattern and search areas by clicking .
Keyframe Tracking

Calculating tracks using keyframes can be the better option for more complex patterns and movement. It requires a little more work to set up, but can produce more reliable tracks.

Unlike auto-tracks, keyframe tracking compares the current pattern anchor to the patterns of the two nearest keyframes.

To help avoid clutter in the Viewer, you can enable or disable the Tracker overlay by right-clicking in the Viewer and selecting Overlay, or by pressing Q to toggle between the available states:

- overlay off
- overlay on
- overlay on, no animation path

Calculating Keyframe Tracks

To calculate keyframe tracks:

1. In the Tracker properties panel, select each track you wish to calculate in the Tracks list or click select all.
2. For each track, select the type of movement the track is intended to output: translation, rotation, or scaling. For example, tracking a feature across the sequence and toward the camera may involve translation and scale.
3. Scrub through the sequence a few frames and adjust the position of the tracking anchor by dragging the anchor to the location of the pattern. You can use the zoom window to fine-tune your positioning. Continue on through the sequence as required.
Tip: You can change the magnification of zoom windows by holding Shift and dragging the magnifying glass cursor away from the center of the window.

At each frame, a new keyframe window is added to the right of the zoom window. The keyframe closest to the current playhead frame is highlighted in orange. It's a good idea to place more keyframes around areas of complexity or greater movement and fewer on straight forward translation. Generally speaking, a greater number of keyframes produces a better track, but at the expense of processing time.

4. When you're satisfied with your keyframes, make sure all your tracks are selected in the Tracks list and then click to track all keyframes.

Tip: When calculating multiple tracks simultaneously, you may find that some tracks stick accurately to the pattern, while others require resetting and reanalysis. When you're happy with a given track, deselect it in the Tracks list. This protects it from recalculation, and lets you experiment with better placement for the wayward tracks.

5. You can also force the selected tracks to recalculate between the two nearest keyframes by clicking in the Viewer toolbar.

See Troubleshooting Keyframe Tracks for help with troublesome tracks.

Importing Tracking Data

You can import tracking data from third-party software using a plain text file containing three values in each line, frame, x, and y coordinates. Tracker can read files with values separated by spaces, commas, or colons and any blank lines or lines starting with #, ;, and / are ignored. For example, importing a file containing:

```
# Tracking Data
1,1,1
10,240,240
```
50,1000,700
90,1800,1200
100,2048,1556

Produces five keyframes at frames 1, 10, 50, 90, an 100 with the relevant x,y coordinates.

**Tip:** If you use a particular file format on a regular basis, you might want to create your own importer to parse the .txt file. You can use the `<install_dir>/plugins/import_discreet.tcl` file as a guide on how to do this.

To import tracking data:
1. Add a tracking anchor as described in Adding Track Anchors.
   This adds a track to the tracks table in the Properties panel.
2. Right-click the new track and select File > Import Time+value Ascii.
   The Import discreet dialog displays.

3. Enter the file path and file name in the File field.
4. Set the required column containing the x and y coordinate data.
5. Click OK to import the track.

The points from the file are converted into keyframes and displayed in the Viewer. The track between keyframes is interpolated as normal. The five keyframe example file described earlier produces a track similar to the following image.
Troubleshooting Keyframe Tracks

Again, even with preset keyframes, some sequences are inevitably going to cause problems. There are a number of pre-tracking checks you can perform to assist Auto-Tracking:

• Play through the sequence before placing your tracking anchors,
• Look for features that are consistent throughout the majority of the sequence,
• Avoid occluded features where possible - see Dealing with Occlusions.

Keyframe tracking won't generally stop when a problem is encountered. Tracker attempts to continue using the next keyframe as a reference, which is why placing a lot of keyframes around problem areas is recommended.

Tip: Tracking areas of distortion or noise can produce unreliable results due to the movement of the pixels in the pattern matching box. One way to deal with this is to seed multiple tracks in and around the problem area and then average the resulting tracks together, producing a single more reliable track, by clicking average tracks in the properties panel.

1. First, turn on the color-coded error indicator of by clicking the traffic light Viewer tool. Each keyframe is colored on a sliding scale from green (good match) to red (poor match).
Bear in mind that a red keyframe doesn't necessarily mean that the tracking result is poor, only that Tracker couldn't reliably match the pattern from one keyframe to the next.

2. Move the tracking anchor to the first of the poor frames, just about the center of the image in the example.

3. Tracker defaults to adding and deleting keyframes automatically when certain conditions are met, but you can toggle these features on and off in the Properties Tracker > Settings tab Keyframe Tracking controls:
   - **re-track when keyframe is moved** - disable this control if you plan to manually position multiple keyframes before re-tracking.
   - **re-track on creation of a new keyframe** - disable this control when placing multiple new keyframes, such as when the track encounters problem areas.
   - **create new key when track is moved** - you could disable this control if you wanted to use the zoom window to examine the sequence more closely without triggering a re-track.
   - **auto-tracks delete keyframes** - when this control is enabled, custom keyframes are deleted during automatic re-tracking.

4. Using the zoom window, drag the anchor to the correct location of the grabbed pattern. Tracker attempts to recalculate the track by including your correction.

5. Advance the playhead to the next poor keyframe and repeat until the track is complete.

Dealing with Occlusions

Tracker's offset capability also applies to keyframe tracking, allowing you to track an obscured feature using the relative position of another feature, providing that the distance between the two points remains constant.

1. Place keyframes normally until you reach the occlusion.
2. Play though the sequence to identify a likely offset point - a pattern that remains equidistant from the last keyframe.

3. Hold down **Ctrl/Cmd** and drag the tracking anchor to the offset position.

The offset amount is recorded in the Tracks list and highlighted in yellow in the Viewer.

4. Continue tracking as normal by clicking the track backward (Z) or forward (V) button. Tracker combines the two tracks into a single continuous track.

5. Use the clear backward and forward buttons to clear poor keyframes. Click clear all to remove all selected tracks and keyframes, excluding manually placed keyframes.

**Note:** You can reset tracking anchor pattern and search areas by clicking .
Applying Tracking Data

You can apply your tracking data to the input image using either the Tracker node's controls, linking expressions, or other Nuke nodes.

Applying Tracking Data Using Tracker’s Controls

The simplest way to apply tracking data to the input image or other nodes is to use the controls of the Tracker node itself. Here, we look at using these controls to stabilize, match-move, and remove or apply jitter.

Stabilizing Elements

The Tracker node's controls let you remove motion, such as unwanted camera shake, from the node's input clip. You can use a single track to stabilize horizontal and vertical motion across the 2D plane, or two or more tracks to eliminate rotation and scale.

1. Create the track(s) you want to use for stabilizing the footage:
   - If you're using a single track, ensure T is selected in the Tracks list, so that Tracker only calculates translation.
     On the Transform tab, select transform > stabilize 1-pt to lock the Filter to Keys. This filter produces the best results when using a single track.
   - If you're using more than one track in the Tracks list, select the transformations that you want Tracker to calculate when stabilizing the image, Translation, Rotation, and/or Scale.
     On the Transform tab, select transform > stabilize.
2. Set the reference frame if you don't want to use the first frame as the transform control frame.
3. Use the smooth controls to average frames together to smooth the transforms applied.
   For example, if you're stabilizing using more than one track, you might average frames together for translation and rotation by entering the number of frames in the t and r fields.
4. Select the required filter. See Choosing a Filtering Algorithm for more information.
   Nuke stabilizes the footage, locking its elements to the same position within the composite.

Note: You can export the transform information to a linked or baked Transform node by selecting the required type from the Export dropdown and clicking create.
**Match-moving Elements**

You can use the Tracker node’s controls to apply the tracked motion to another image, that is, to match-move an image.

1. Use a Tracker node to track the feature you intend to match.
2. Copy the Tracker node and paste it after the footage you want to match-move.
3. In the second Tracker node’s controls, go to the Transform tab.
4. From the transform dropdown menu, select **match-move**.
5. Set the **reference frame** if you don't want to use the first frame as the transform control frame.
6. Use the **smooth** controls to average frames together to smooth the transforms applied.
   For example, if you’re using more than one track, you might average frames together for translation and rotation by entering the number of frames in the **t** and **r** fields.
7. Select the required filter. See **Choosing a Filtering Algorithm** for more information.

Nuke applies the tracked movement to the footage you want to match-move. A simple script might appear as follows, where Tracker2 is a copy of Tracker1:

```plaintext
A simple match-move script.
```

**Note:** You can export the transform information to a linked or baked Transform node by selecting the required type from the **Export** dropdown and clicking **create**.
Removing or Adding Jitter

Tracker can be used to remove high frequency jitter from a sequence, exaggerate existing jitter, or add jitter to another sequence for a consistent look.

To remove jitter:
1. Create the tracks you want to use for jitter removal.
2. In the Tracker node's controls, go to the Transform tab.
3. From the transform dropdown menu, select remove jitter.
4. Set the reference frame if you don't want to use the first frame as the transform control frame.
5. Use the jitter period to average together frames, adjusting the jitter to achieve the required stability.
6. Use the smooth controls to average frames together to smooth the transforms applied.
   For example, if you're removing jitter using more than one track, you might average frames together for translation and rotation by entering the number of frames in the t and r fields.
7. Select the required filter. See Choosing a Filtering Algorithm for more information.
8. Nuke removes jitter from the footage, locking its elements to the same position within the composite.

To exaggerate or add jitter:
1. Create the tracks you want to use for the required jitter operation.
2. In the Tracker node's controls, go to the Transform tab.
3. From the transform dropdown menu, select add jitter.
4. Set the reference frame if you don't want to use the first frame as the transform control frame.
5. Use the jitter period to average together frames, adjusting the jitter to achieve the required instability.
6. Use the smooth controls to average frames together to smooth the transforms applied.
   For example, if you're adding jitter using more than one track, you might average frames together for translation and rotation by entering the number of frames in the t and r fields.
7. Select the required filter. See Choosing a Filtering Algorithm for more information.
   Nuke exaggerates the tracked jitter in the footage.

Note: If you want to transfer jitter to another sequence, copy the Tracker node and paste it after the footage to which you want to add jitter, then follow steps 3-7 above.
Applying Tracking Data Using Linking Expressions

Nuke's CornerPin2D and Stabilize2D nodes are specifically designed to receive tracking data by linking expressions, but you can apply tracking data in this manner to virtually any Nuke node. For example, you might animate a Bezier or a B-spline shape with tracking data by entering linking expressions into the RotoPaint node's transformation parameters. You can also apply tracking data to individual points.

This section explains the basic procedure for applying tracking data to any node via linking expressions.

Creating Linking Expressions

The Tracker node's Tracker panel displays data related to the position of each track anchor over time (tracks' x and y fields). This is the data that you most typically apply to other nodes.

To Drag and Drop Tracking Data:
1. Display both the Tracker parameters and the parameters to which you wish to apply the tracking data (the destination control - for example, a RotoPaint node's translate control).
2. Select the required track from the Tracks list. Only one track can be linked per control.
3. Ctrl+drag (Cmd+drag on a Mac) from the source control animation button to the destination control animation button.

When you release, the destination control will turn blue, indicating an expression has been applied. In this case, the drag and drop action has created a linking expression resembling the following example:
Tracker1.tracks.1.track_x

Tip: You can also apply tracking (or other transform) data to individual RotoPaint, SplineWarp, or GridWarp points (this is sometimes called per vertex tracking). To do so, Ctrl/Cmd+drag and drop the track's animation button on a RotoPaint, SplineWarp or GridWarp point in the Viewer.

You can add other components to this linking expression as necessary. For example, you might add a spatial offset to the linking expression by subtracting out the initial frame's tracking values, in which case the final expression would resemble the following:
Tracker1.tracks.1.track_x-Tracker1.tracks.1.track_x(1)
See **Expressions** for more information. Once you enter the linking expression, the destination parameter turns blue.

### To Link Animated Parameters with a Tracker Node

You can also link controls with the Tracker node if you use the **Link to** option in the Animation menu. For example, to link the **translate** control of the Roto node with a Tracker node, do the following:

1. Create the Tracker node you want to link to.
2. Go to Tracker's Transform tab and enable **live-link transform**.
   This control enables Tracker to update dynamically as the expression link changes.
3. On the **Transform** tab of the Roto node's properties panel, click on the **translate** animation menu.
4. Select **Link to > Tracker linking dialog**.
5. Select the Tracker node you want to use in the **tracker node** dropdown and in the **link to** dropdown, select whether you want to link to the **position** of the track, the **translate** values of the track, or treat the translate value as an offset.
6. Select which tracks you want to use by checking the **track** boxes. The Expression field updates with the appropriate expression syntax. If you select more than one track, the tracks are averaged, for example:
   \[ (\text{Tracker1.tracks.1.track}_x + \text{Tracker1.tracks.2.track}_x)/2 \]
7. Then click **OK**, and your linking is done.
   Your Bezier shape's translate value now changes when the Tracker value is changed.

### Transforming Masks with Tracking Data

Creating animated masks using Roto and keyframes can be a very time-consuming process, but Nuke's Tracker node can do some of the initial work for you, especially with garbage mattes.

Once you have some solid tracking data, you can drive a roto shape without keyframing individual points:

1. Track a feature in the area you intend mask. In the example, the figure's head serves as the driving point for the matte.
2. Add a Roto node to the script (keyboard shortcut O) and draw the shape you intend to drive with the tracking data. In this case, we don't need to be too accurate as we're creating a garbage matte.

3. In the Roto properties panel, click the Transform tab and select the matte in the shapes list.

4. Right-click the translate control's animation icon and then select Link to > Tracker 1 > track 1. The Tracker's track_z and track_y keyframes are copied into the Roto's translate control, applying the same translate and offset to the matte shape.

5. To compensate for this, select the Root item in the shape list and reposition the Roto shape correctly using the transform handle in the Viewer.
6. Scrub the playhead to see the matte following the tracked path.

Using the CornerPin2D Node

The CornerPin2D node is designed to map the four corners of an image sequence to positions derived from tracking data. In practice, this node lets you replace any four-cornered feature with another image sequence. For example, suppose you needed to replace the monitor image in the fast-panning shot shown below.

Fast-panning shot requires four-corner tracking.

First, use the Tracker to calculate four separate tracks, one for each corner of the feature.

Generating the four tracks.
Next, attach a CornerPin2D node to the image sequence you want to use as the replacement for the feature, and apply the tracking data. This remaps the image sequence’s corners to the correct positions over time. You can create the node manually, or by using Tracker’s Export dropdown.

The final step is to layer the result over the original element.

The compositing image.

The steps below summarize the use of Tracker’s Export CornerPin2D workflow.

**To Use the CornerPin2D Node**

1. Generate four tracks, one per corner, on the feature requiring replacement.
2. Use the current frame or reference frame field to specify the frame to use as the starting point. You can also decide whether the CornerPin2D node is expression linked or baked using the Export dropdown:
   - CornerPin2D (use current frame)
   - CornerPin2D (use transform ref frame)
   - CornerPin2D (use current frame, baked)
   - CornerPin2D (use transform ref frame, baked)
3. Click create to add the CornerPin2D node to the script.
4. Attach the image or sequence to replace the feature tracked to the input of the CornerPin2D node.
5. If necessary, select a different filtering algorithm from the filter dropdown menu. See Choosing a Filtering Algorithm for more information.
6. When filtering with Keys, Simon, or Rifmen filters, you may see a haloing effect caused by the pixel sharpening these filters employ. If necessary, check clamp to correct this problem.
7. In most cases, you will keep black outside checked. This renders black pixels outside the image boundary, making it easier to layer the element over another. (If you uncheck this parameter, the outside area is filled with the outermost pixels of the image sequence.)
8. The final step is to layer the result over the original element.

A simple script might appear as follows:
Transforming Elements

These pages explain how to perform a range of 2D and 2.5D spatial transformations. You learn how to apply geometric transformations (including translations, rotations, scales, and skews) to elements, and how to add motion blur using the nodes in the Transform menu.
Transforming in 2D

This section describes how to apply 2D transformations including translations, rotations, scales, and skews to elements using a number of Nuke nodes. All transforms, except translations, change the size of the image bounding box.

Tip: You can enable a warning to indicate when the bounding box is greater that the format in Nuke's Preferences. See Bounding Box Warnings for more information.

Using the 2D Transformation Overlay

Several of the nodes discussed in this section display a Viewer overlay for executing spatial transformations. This overlay is often a faster alternative to the properties panel. The figure below shows you how to use Nuke 2D transformation overlay.

Transformation overlay.

- A) Drag to skew the frame (see Skewing Elements).
- B) Drag to scale the frame uniformly - simultaneously on x and y (see Scaling Elements).
- C) Drag to translate the frame (see Translating Elements).

Shift+drag to constrain the translation to x or y.

Ctrl/Cmd+drag to reposition the pivot point (the point that acts as the center to transformation operations).
- D) Drag to scale the frame on x.
• E) Drag to rotate the frame around the pivot point (see Rotating Elements). The transform overlay snaps to typical values. To prevent the snapping, press Shift while dragging.
• F) Drag to scale the frame on y.

Translating Elements

To translate an element is to slide it on x or y.

Using the Transform Node

The Transform and TransformMasked nodes let you not only translate elements, but also rotate, scale, and skew them from a single properties panel.

TransformMasked is identical to Transform except that it offers controls for assigning a mask to protect certain areas of the frame from translations. For the sake of brevity, this chapter only discusses the use of Transform, but keep in mind you can use TransformMasked any time you need to process a transformation through a mask. The mask controls work in the same fashion as those described in Masking Color Corrections.

To translate an element using the Transform node:
1. Click **Transform > Transform** to insert a Transform node at the appropriate place in your script.
2. Connect a Viewer to the output of the Transform node so you can see the effect of your changes.
3. In the Transform properties panel, increment or decrement the **translate x** and **y** fields to slide the element along either axis.
   Or drag on the center of the transformation overlay.
Using the Position Node

The Position node gives you just bare-bones parameters for translating an element.

To translate an element using the Position node:
1. Click Transform > Position to insert a Position node at the appropriate place in your script.
2. Connect a Viewer to the output of the Position node so you can see the effect of your changes.
3. In the Position properties panel, increment or decrement the translate x and y fields to slide the element along either axis.

Rotating Elements

To rotate an element is to spin it around the pivot point.

To rotate an element using the Transform node
1. Click Transform > Transform to insert a Transform node at the appropriate place in your script.
2. Connect a Viewer to the output of the Transform node so you can see the effect of your changes.
3. In the Transform properties panel, select the appropriate filtering algorithm from the filter dropdown menu (see Choosing a Filtering Algorithm).
4. Position the pivot point as necessary:
   • Increment or decrement the center x and y fields to move the axis in either direction.
   • Or press Ctrl (Cmd on a Mac) while dragging on the center of the transformation overlay.
5. Increment or decrement the rotate field.
   Or drag on the horizontal bar of the transformation overlay.
Scaling Elements

To scale an element is to resize it by adding (upsampling) or removing (downsampling) pixels.

Nuke offers several nodes for scaling elements. Transform is designed primarily for scaling the background plate up or down in a composite. The scaling functions for transform are described below.

Reformat is designed for writing out elements with specific resolutions and pixel aspect ratios. Adding Motion Blur describes the use of this node.

To scale an element using the Transform node
1. Click Transform > Transform to insert a Transform node at the appropriate place in your script.
2. Connect a Viewer to the output of the Transform node so you can see the effect of your changes.
3. In the Transform properties panel, select the appropriate filtering algorithm from the filter dropdown menu (see Choosing a Filtering Algorithm).
4. Position the pivot point as necessary:
   - Increment or decrement the center x and y fields to move the axis in either direction.
   - Or press Ctrl (Cmd on a Mac) while dragging on the center of the transformation overlay.
5. To scale the frame uniformly (on both x and y):
   - Increment or decrement the Transform node’s scale field.
   - Or drag the circle-portion of the transformation overlay.
6. To scale the frame asymmetrically (on x or y):
   - Click scale parameter’s channel chooser \[2\] to reveal the x and y fields, then increment or decrement each individually.
• Or drag any of the four points on the circle-portion of the transformation overlay. The top and bottom points scale on y; the left and right points, on x.

Skewing Elements

To *skew* an element is to rotate its pixel columns around the pivot point.

To skew an element using the Transform node
1. Click **Transform > Transform** to insert a Transform node at the appropriate place in your script.
2. Connect a Viewer to the output of the Transform node so you can see the effect of your changes.
3. In the Transform properties panel, select the appropriate filtering algorithm from the **filter** dropdown menu (see [Choosing a Filtering Algorithm](#)).
4. Position the pivot point as necessary:
   • Increment or decrement the **center x** and **y** fields to move the axis in either direction.
   • Or **Ctrl+drag (Cmd+drag on a Mac)** on the center of the transformation overlay.
5. Increment or decrement the **skew** field to rotate the pixel columns around the pivot point.
   Or drag the vertical bar of the transformation overlay.

To Invert a Transform Effect

You can invert the effect you've created with the Transform node by checking the **invert** box in the Transform properties panel. This uses the inverse values of your translate, rotate, scale and skew values. When the box is checked, a small transform handle appears next to the standard transform handle in the Viewer.
Bounding Box Warnings

Zooming into the Viewer to work on a shot means that you can't always see the extent of the bounding box in relation to the format, which can result in unnecessary processing.

To make it easier to see the state of your bounding box, Nuke can display visual warnings on the nodes that affect the bounding box. To enable the warnings, in Nuke's Preferences under Panels > Node Graph, enable Bounding Box Warning:

- **red rectangle with dotted stroke** - the indicated node creates a bounding box greater than the format.

- **dotted stroke without the red rectangle** - the bounding box size is greater than the format at the indicated node, but the bounding box size has been set by an upstream node.
The bbox warning **threshold** controls how far past the edge of the format the bounding box can expand before the warning is displayed in the Node Graph. For example, if you're working with UHD_4K footage and the default 10% threshold, you can expand the bounding box horizontally by 384 pixels before the warning is displayed.

**Tip:** You can set the color of the warning rectangle in the **Preferences** under **Panels > Node Graph > Bounding Box Warning.**
Choosing a Filtering Algorithm

Spatial transformations involve remapping pixels from their original positions to new positions. The question arises as to what values to assign remapped pixels. In the simplest case, they retain their original values, but this can create problems with image quality, particularly in high contrast areas of the frame. For example, the figure below shows a close up of a high-contrast feature that has been rotated clockwise by 45 degrees. The remapped pixels have retained their original values, but the result is a highly aliased, or jagged, edge:

The solution is to apply a more sophisticated *filtering algorithm* to determine the values of remapped pixels - one that takes into account, in some fashion, the values of neighboring pixels.

For example, applying Nuke’s cubic algorithm to the above rotation, results in a softer, less jagged edge:

When executing spatial transformations, Nuke lets you select from the filtering algorithms described in the table below.

Note that the curves shown in the table plot the manner by which each algorithm samples from neighboring pixels. The center of each curve represents the value of the remapped pixel itself, and the rising and falling portions of each curve represent the amount of sampling that occurs across a five pixel radius.

**Tip:*** When using filters that employ sharpening, such as Rifman and Lanczos, you may see a haloing effect. Some nodes, such as Transform and Tracker, include a clamp control to correct this problem.
<table>
<thead>
<tr>
<th>Filter</th>
<th>Description</th>
<th>Sampling Curve and Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impulse</td>
<td>Remapped pixels carry original values.</td>
<td></td>
</tr>
<tr>
<td>Cubic</td>
<td>Remapped pixels receive some smoothing.</td>
<td></td>
</tr>
<tr>
<td>(default)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Keys</td>
<td>Remapped pixels receive some smoothing, plus minor sharpening (as shown by the negative -y portions of the curve).</td>
<td></td>
</tr>
<tr>
<td>Filter</td>
<td>Description</td>
<td>Sampling Curve and Output</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>Simon</td>
<td>Remapped pixels receive some smoothing, plus medium sharpening (as shown by the negative -y portions of the curve).</td>
<td><img src="sampling_curve_simon.png" alt="Sampling Curve and Output" /></td>
</tr>
<tr>
<td>Rifman</td>
<td>Remapped pixels receive some smoothing, plus significant sharpening (as shown by the negative -y portions of the curve).</td>
<td><img src="sampling_curve_rifman.png" alt="Sampling Curve and Output" /></td>
</tr>
<tr>
<td>Mitchell</td>
<td>Remapped pixels receive some smoothing, plus blurring to hide pixelation.</td>
<td><img src="sampling_curve_mitchell.png" alt="Sampling Curve and Output" /></td>
</tr>
<tr>
<td>Filter</td>
<td>Description</td>
<td>Sampling Curve and Output</td>
</tr>
<tr>
<td>----------</td>
<td>-----------------------------------------------------------------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>Parzen</td>
<td>Remapped pixels receive the greatest smoothing of all filters.</td>
<td></td>
</tr>
<tr>
<td>Notch</td>
<td>Remapped pixels receive flat smoothing (which tends to hide <em>moiré</em> patterns).</td>
<td></td>
</tr>
<tr>
<td>Lanczos4</td>
<td>Remapped pixels receive minor sharpening (as shown by the negative -y portions of the curve), good for scaling down. The filter number at the end of the filter name denotes the width of the filter (4 pixels).</td>
<td></td>
</tr>
<tr>
<td>Filter</td>
<td>Description</td>
<td>Sampling Curve and Output</td>
</tr>
<tr>
<td>------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>Lanczos6</td>
<td>Remapped pixels receive some sharpening (as shown by the negative -y portions of the curve), good for scaling down. The filter number at the end of the filter name denotes the width of the filter (6 pixels).</td>
<td><img src="image1" alt="Sampling Curve" /> <img src="image2" alt="Output" /></td>
</tr>
<tr>
<td>Sinc4</td>
<td>Remapped pixels receive a lot of sharpening (as shown by the negative -y portions of the curve), good for scaling down. The filter number at the end of the filter name denotes the width of the filter (4 pixels).</td>
<td><img src="image3" alt="Sampling Curve" /> <img src="image4" alt="Output" /></td>
</tr>
</tbody>
</table>

**Note:** Lanczos and Sinc filters exhibit haloing as the filter becomes sharper.

**Note:** The numbers appended to the filter name denote the width of the filter, similar to the naming scheme used by PRMan. See [http://www.renderman.org/RMR/st/PRMan_Filtering/Filtering_In_PRMan.html](http://www.renderman.org/RMR/st/PRMan_Filtering/Filtering_In_PRMan.html) for more information.
How Nodes Concatenate

Concatenation is behavior that some Nuke nodes perform when you have several nodes transforming your image one after another. When nodes concatenate, they pass on these adjacent transformation operations to the last transforming node in the row, and the last node then performs all the transformations at once.

For example, if you have three Transform nodes in a row, instead of performing each transformation separately and filtering the input image three times, Nuke combines the transformations into one operation and only filters the image once. As filtering unavoidably destroys some image detail, it’s important to concatenate as many nodes as possible in order to preserve image quality.

Three Transform nodes that concatenate. The image is only filtered once.

A Crop node between the Transform nodes breaks concatenation. The image is filtered twice, resulting in poor image quality.
In order to concatenate, the concatenating nodes have to be adjacent. So, if you have a node that doesn't concatenate (a Crop node for example) between two concatenating nodes (for example Transform nodes), no concatenation occurs.

If you're using more than one filtering method in the nodes that concatenate, the last filtering method in the series of concatenating nodes is applied on the result.

When nodes concatenate, Nuke only uses the filtering method set on the last concatenating node (in this case, the filtering method used is Mitchell).

As a rule of thumb, the only nodes that concatenate are usually transform nodes. In addition to these, Dot, NoOp, Switch nodes and disabled nodes do not break concatenation.

Color nodes do not concatenate because Nuke works in a 32-bit float, which is enough to avoid banding and visible round-off errors in color.

Please be aware that some transform nodes do not concatenate and should not be interspersed with those that do. There are also transform nodes that receive concatenation, but don't pass anything on.

<table>
<thead>
<tr>
<th>Nodes that concatenate</th>
<th>Only concatenate upstream</th>
<th>Do not concatenate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Card3D</td>
<td>BasicMaterial</td>
<td>AdjustBbox</td>
</tr>
<tr>
<td>CornerPin</td>
<td>Diffuse</td>
<td>BlackOutside</td>
</tr>
<tr>
<td>Nodes that concatenate</td>
<td>Only concatenate upstream</td>
<td>Do not concatenate</td>
</tr>
<tr>
<td>------------------------</td>
<td>---------------------------</td>
<td>--------------------</td>
</tr>
<tr>
<td>Reconcile3D</td>
<td>DisplaceGeo</td>
<td>Crop</td>
</tr>
<tr>
<td>Reformat</td>
<td>Displacement</td>
<td>Mirror</td>
</tr>
<tr>
<td>Stabilize</td>
<td>Emission</td>
<td>PlanarTracker</td>
</tr>
<tr>
<td>Tracker</td>
<td>Environment</td>
<td>PointsTo3D</td>
</tr>
<tr>
<td>Transform</td>
<td>Flare</td>
<td>Position</td>
</tr>
<tr>
<td>Dot</td>
<td>GridWarp</td>
<td>TVIScale</td>
</tr>
<tr>
<td>NoOp</td>
<td>IDistort</td>
<td></td>
</tr>
<tr>
<td>Switch</td>
<td>LensDistortion</td>
<td></td>
</tr>
<tr>
<td>Disabled nodes</td>
<td>MotionBlur2D</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Phong</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Sparks</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Specular</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SphericalTransform</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SplineWarp</td>
<td></td>
</tr>
<tr>
<td></td>
<td>STMap</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Tile</td>
<td></td>
</tr>
<tr>
<td></td>
<td>TransformMasked</td>
<td></td>
</tr>
</tbody>
</table>
Applying Core Transformations in 2.5D

Nuke’s Card3D node lets you apply the same geometric transformations possible with the Transform node, but gives you an additional axis of operation, z.

Just to be clear, the Card3D node’s transformations are not truly 3D, but rather what is sometimes called “2.5D” - meaning that you can move an element back on the z axis, but doing so does not convey the sense that it is behind or in front of another element. 2.5D transformations are useful for tasks like “cheating” the perspective of an element or “faking” a camera zoom.

Remember, however, that Nuke doesn’t limit you to 2.5 dimensions. If you need true 3D capabilities, you can construct a 3D scene. See 3D Compositing for more information.

Using the 3D Transformation Handles

You’ll note when viewing the output of a Card3D node that it displays an overlay for executing spatial transformations. This overlay is often a faster alternative to the properties panel. The figure below shows you how to use it.

- A) Drag to translate the frame on the y axis (see Translating Elements).
- Press Ctrl/Cmd while dragging to rotate the frame on any axis (see Rotating Elements).
- B) Drag to translate the frame on the z axis.
- Press Ctrl/Cmd while dragging to rotate the frame on any axis.
- C) Drag to translate the frame on the x axis.
Press **Shift** while dragging to constrain the translation to x.

Press **Ctrl/Cmd** while dragging to rotate the frame on any axis.

Adding a Card3D Node

To add a Card3D node:

1. Click **Transform > Card3D** to insert a Card3D node at the appropriate place in your script.
2. Connect a Viewer to the output of the Card3D node so you can see the effect of your changes.

Translating Elements

When using the Card3D node, you can translate elements on z in addition to the other axes.

To Translate an Element Using the Card3D Node

In the Card3D properties panel, increment or decrement the **translate x**, **y**, and **z** fields to slide the element along any axis,

**OR**

Drag on any axis or transformation overlay.

Rotating Elements

When using the Card3D node, you can rotate elements around the x and y axes, in addition to the z. This is useful for cheating the perspective.
To rotate elements using the Card3D node
1. Position the pivot point as necessary by incrementing or decrementing the \textit{pivot x, y,} and \textit{z} fields to move the axis in any direction.
   Alternatively, you can position the pivot point by pressing Ctrl/Cmd+Alt while dragging.
2. Increment or decrement the \textit{rotatex, y,} and \textit{z} fields to spin the element around the pivot point.
   Alternatively, you can press Ctrl/Cmd while dragging on any axis on the transformation overlay.

**Scaling Elements**

To scale an element using the Card3D node:
1. Position the pivot point as necessary by incrementing or decrementing the \textit{pivot x, y,} and \textit{z} fields to move the axis in any direction.
   Alternatively, you can position the pivot point by pressing Ctrl/Cmd+Alt while dragging.
2. To scale the frame simultaneously on \textit{x, y,} and \textit{z,} increment or decrement the \textit{uniform scale} field.
3. To scale the frame asymmetrically, increment or decrement the \textit{scale x, y,} and \textit{z} fields.

**Skewing Elements**

Whereas the Transform node lets you rotate pixel columns only around the \textit{z} axis, Card3D permits you to do so around all three axes.

**To Skew an Element Using the Card3D Node**
1. Position the pivot point as necessary by incrementing or decrementing the \textit{pivot x, y,} and \textit{z} fields to move the axis in any direction.
Alternatively, you can position the pivot point by pressing Ctrl/Cmd+Alt while dragging.

2. Increment or decrement the skew x, y, and z fields to rotate the pixel columns around the corresponding axes.

Specifying the Order of Operations

The order by which Nuke executes operations can affect the outcome. The Card3D node lets you select the order by which Nuke executes scales, rotations, and translations, as well as the order by which it executes rotation on individual axes.

To Choose the Operation Order for Scales, Rotations, and Translations

In the Card3D properties panel, select an option from the transform order dropdown menu, which displays all possible combinations (S signifies scale, R, rotation; and T, translation).

To Choose the Operation Order for Rotations

Select an option from the rotation order dropdown menu, which displays all possible axial combinations.

Choosing a Filtering Algorithm

Filtering algorithms let you specify the degree of smoothing and sharpening that remapped pixels receive during transformation. The Card3D node offers the same filter algorithms as the Transform node. See Choosing a Filtering Algorithm for more information.

To choose a filter algorithm, select the desired algorithm from the filter dropdown menu.
Adding Motion Blur

The following nodes under the Transform menu have their own controls for adding motion blur to transformations:

- Transform
- TransformMasked
- Card (3D)
- CornerPin2D
- Reconcile3D
- Tracker
- Stabilize2D

These controls allow you to create motion blur without adding a separate node for it. The output is similar to a TimeBlur node (see Applying a TimeBlur Filter), but rather than averaging the results of several whole images computed at steps over the shutter period, a number of samples are taken at many random times over the shutter period. This effectively gives many more "steps" and thus a smoother-looking result for a smaller total number of computations.

When using several of these nodes in a row, the motion blur is concatenated, and the last transform in the chain defines the motion blur applied.

To Add Motion Blur

1. Open the transform node’s controls.
2. Create a transform and animate it. For instructions on how to do this, see the Using the Compositing Environment chapter.

3. In the **motionblur** field, enter the sampling rate. This affects the number of times the input is sampled over the shutter time. The higher the rate, the smoother the result. In many cases, a value of 1.0 is enough. Setting the value to 0 produces no motion blur.

4. In the **shutter** field, enter the number of frames the shutter stays open when motion blurring. For example, a value of 0.5 would correspond to half a frame. Increasing the value produces more blur, and decreasing the value less.

5. From the **shutteroffset** dropdown menu, select when the shutter opens and closes in relation to the current frame value:
   - to center the shutter around the current frame, select **centered**. For example, if you set the **shutter** value to 1 and your current frame is 30, the shutter stays open from frame 29.5 to 30.5.
   - to open the shutter at the current frame, select **start**. For example, if you set the **shutter** value to 1 and your current frame is 30, the shutter stays open from frame 30 to 31.
   - to close the shutter at the current frame, select **end**. For example, if you set the **shutter** value to 1 and your current frame is 30, the shutter stays open from frame 29 to 30.
   - to open the shutter at the time you specify, select **custom**. In the field next to the dropdown menu, enter a value (in frames) you want to add to the current frame. To open the shutter before the current frame, enter a negative value. For example, a value of -0.5 would open the shutter half a frame before the current frame.

   ![](image.png)

**To Add Motion Blur to an Image Rendered in a Third-party Application**

Another way to add motion blur to your image is to use the VectorBlur node. VectorBlur takes each of your image’s pixels and blurs them in a straight line, using the u and v channels to determine the blur direction.

VectorBlur expects the values from your input plates to be pixel space screen units, in other words one unit should be equal to one pixel. Nuke uses this information to calculate the distance that one pixel travels between two frames. So, in order to get a working motion blur result, you should make sure Nuke is getting correct values to work with. You might have files using varying values, particularly if you've used a third party application to create your input files. The following is an example of creating motion blur with the VectorBlur node using files written from a third party application.
To Create Motion Blur with the VectorBlur Node

1. Read in your footage and motion blur files, for example an .exr file with a spinning donut and a .sgi file with motion blur vectors that are normalized to have values between 0 and 1.

2. Using the ShuffleCopy node, select which channels VectorBlur should read from your motion vector file (node input 1) and color image file (node input 2). In this case, you would use the motion vector file's red and green channels as the motion u and v channels, and its alpha channel as the alpha channel. Meanwhile, the image file would output the red, green and blue channels for the main color image.

With this setup, your ShuffleCopy node controls would look like this:

3. Connect the VectorBlur node to the ShuffleCopy node. You also need to tell VectorBlur which motion vector channels to use, so change the uv channels control to motion.

4. If your motion vectors have been normalized to be between 0 and 1, you can set the u and v values in the add control to -0.5 to offset the motion blur center. This is usually necessary for any motion vectors stored in an integer file format like 16 bit TIFF or TARGA. Vectors that go to negative x or y directions use half the numbers in the range and vectors that go positive use the other half.

5. With the multiply and offset controls, you can further adjust the amount of motion blur you want to produce. The offset value allows you to correct for normalization of your vector values, and the multiply value controls the magnitude of them.

6. If the vectors have been premultiplied with the alpha channel, their value is not accurate in places where the alpha is not 1.0. You'll want to check the alpha checkbox to use the input image's alpha channel to help VectorBlur to deal with motion vectors that have been premultiplied by this alpha channel.
Adding Motion Blur | To Create Motion Blur with the VectorBlur Node

7. Getting a good, even motion blur result largely depends on selecting the right calculation method. In the method dropdown, select:
   - **backward** - backward method is effective and fast but may not be accurate if you don't have motion vector values at all pixels throughout the whole frame
   - **forward** - the forward method is slower, but it gives you a more accurate result, especially in cases where the vectors don't cover the whole frame. In this case, we know the motion vectors are not continuous, so selecting forward is a good option.

[Images of Alpha disabled and enabled]
Replicating the Input Image Across the Output

The Tile node produces an output image that contains scaled-down, tiled copies of the input image. The output image is the same format as the input.

![Before the Tile node.](image1) ![After the Tile node.](image2)

To Use the Tile Node

1. Select the image you want to replicate and select **Transform > Tile**.
   A Tile node is inserted in the Node Graph.
2. Attach a Viewer to the Tile node.
3. In the Tile node properties, use the **rows** field to define how many times the image is replicated vertically. Note that the value can be fractional.
4. In the **columns** field, enter the number of times you want to replicate the image horizontally. Note that the value can be fractional.

![The original input image.](image3) ![The output of the Tile node with rows set to 4 and columns set to 5.](image4)

If you want to flip adjacent tiles vertically to form mirror images, check **mirror**.
The output of the Tile node without mirroring.  
The same image with vertical mirroring.

If you want to flip adjacent tiles horizontally to form mirror images, check mirror.

The output of the Tile node without mirroring.  
The same image with horizontal mirroring.

5. From the filter menu, select an appropriate filtering algorithm. For more information, see Choosing a Filtering Algorithm.
Warping Images

Nuke's GridWarp and SplineWarp nodes allow you to distort elements in an image and morph one image into another. There are other types of warp available in Nuke, but in this chapter, we focus on the GridWarp and SplineWarp nodes.

Quick Start

Here's a quick overview of the workflow:

1. For both warping and morphing, you can use either GridWarp node (Transform > GridWarp) or the SplineWarp (Transform > SplineWarp). Working with the GridWarp is sometimes slightly faster, whereas the SplineWarp node allows for more fine adjustment. With both nodes, you first connect your source image to the src or A input and, if morphing, the destination image to the dst or B input. GridWarp also allows you to connect an additional background image to the bg input.

2. Depending on which node you're using, you can then go on to set your source and destination points, and tweak them until you're happy with your results. For more information, see Warping and Warping an Image Using the SplineWarp Node.

3. If you're looking to morph an image into another, you can do this with both warper nodes as well. For more information, see Morphing and Morphing One Image into Another Using the SplineWarp Node.

4. If you are warping or morphing moving images, rather than stills, have a look at Transforming and Animating Warps and Morphs.
Warping

Warping refers to manipulating an image so that elements in the image are distorted. Unlike many of the transformations described under Transforming Elements, warps are transformations that only affect some of the pixels in an image rather than all of them. For example, you might make an animal’s eyes bigger or a person’s smile wider without affecting the rest of their features.

This is not to say that the pixels around the area you are moving do not move with the area. They do, because accommodating the change this way often produces more realistic results. However, the distortion lessens the further you get from the moved pixels. You also have some control over which pixels are moved and which are not, and can isolate the warp to a small area. Still, in an ideal situation, the subject you are going to warp is a subject you can key out or rotoscope to isolate it from its background before you create the warp. This way, you can be sure that the background stays intact.

In addition to performing creative manipulations on the shapes of the subjects in your images, you can also use warping to simulate different types of film or video lenses or to remove unwanted lens distortions.

Below, we discuss how to warp images, first using the GridWarp node and then the SplineWarp node. Finally, we also teach you to animate the warps. Again, we start with the GridWarp node and then show you how to do the same with the SplineWarp node.

Warping Images Using the GridWarp Node

The GridWarp node allows you to warp images by transferring image information from one Bezier grid onto another. When using this node, you first position the source grid, which defines where to warp from. Next, you position the destination grid, which defines where to warp the image to. This grid can be a duplicate of the source grid, or you can define it separately. When you manipulate the destination grid, the corresponding warp is applied to the source image.

The GridWarp node also includes controls for animating the warp and selecting the level of filtering used to remove any artifacts the warp may have caused.

To Warp an Image Using the GridWarp Node

1. Select Transform > GridWarp to insert a GridWarp node after the image you want to warp.
2. Connect the src input and a Viewer to the image.
3. When the GridWarp properties panel is open, by default the destination grid overlay appears in the Viewer. You can show or hide the source and destination grids using the \[\text{[s]}\] and \[\text{[p]}\] buttons in the Viewer toolbar or the \text{visible} checkbox in the \text{GridWarp} tab. Use the destination grid to define the warp areas.

\[\text{Note:}\] GridWarp automatically attempts to resize the grids to the image size as long as you have not modified any of the control points.

If the grids are not the same size as the input image, click the \text{Resize to image} buttons under both \text{Source Grid} and \text{Destination Grid}.
4. Use the Viewer tools to control the following aspects of the grids:

Note: You can use the copy and paste buttons \(\text{Copy} \) \(\text{Paste} \) in the grid controls to copy control point keyframes between the source and destination grids.

<table>
<thead>
<tr>
<th>Control</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>output</td>
<td>Controls the output displayed in the Viewer:</td>
</tr>
<tr>
<td></td>
<td>• source - the source image and source grid.</td>
</tr>
<tr>
<td></td>
<td>• sourcewarped - the source image and destination grid.</td>
</tr>
<tr>
<td></td>
<td>• destination - the destination image and destination grid.</td>
</tr>
<tr>
<td></td>
<td>• destinationwarped - the destination image and source grid.</td>
</tr>
<tr>
<td></td>
<td>• morph - the morphed image, controlled by the warp and mix parameters, and both grids.</td>
</tr>
</tbody>
</table>

When enabled, the grids shown in the Viewer depends on the output setting. For example, if output is set to source warped, only the destination grid appears in the Viewer.

When enabled, the source grid is displayed in the Viewer. This control can be overridden by the button.

When enabled, the destination grid is displayed in the Viewer. This control can be overridden by the button.

When enabled, changes to points in a grid are automatically keyed. You can disable this and set your keyframes manually, which is particularly useful if you intend to use the Curve Editor. See Animating Warps.

ripple

Rippling keyframes allows you to adjust the position of a stroke/shape point on one frame and have that same relative adjustment applied to the point across all frames or a specified range of frames.

• off - ripple edit is disabled.
• all - ripple all frames in the sequence.
• from start - ripple frames from the first frame to the current frame.
• to end - ripple frames from current frame to the last frame.
• range - ripple all frames within the from and to fields.

label points

When enabled, points selected on the grid are labeled \(x,y\) measured from the
<table>
<thead>
<tr>
<th>Control</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Control Icon]</td>
<td>origin.</td>
</tr>
<tr>
<td>![Control Icon]</td>
<td>When enabled, the transform handle overlays all selected points.</td>
</tr>
<tr>
<td>![Control Icon]</td>
<td>When enabled, a low resolution preview displays when points are moved on a grid. Once the render is complete, the low-res image is updated.</td>
</tr>
<tr>
<td>divisions</td>
<td>Enter the number of divisions required in the field to modify the grid. The number of divisions must be between 3 and 20. You can also click and hold the slider to overlay a preview of the subdivisions. GridWarp attempts to modify the grid based on the current control point distribution.</td>
</tr>
<tr>
<td>![Note Icon]</td>
<td><strong>Note:</strong> When using the preview, accuracy may suffer at lower divisions and additional smoothing may be required at higher divisions.</td>
</tr>
<tr>
<td>![Control Icon]</td>
<td>Click to enable <strong>Edit</strong> mode. Select individual points, multiple points using shift-click, or marquee groups of points. Edit mode also allows you to adjust the curve between points to produce distortion.</td>
</tr>
<tr>
<td>![Control Icon]</td>
<td>Click to enable <strong>Insert</strong> mode. Click on a horizontal line to add a vertical line to the grid and vice versa.</td>
</tr>
<tr>
<td>![Control Icon]</td>
<td>Click to enable <strong>Delete</strong> mode. Click on a grid line to remove it from the Viewer.</td>
</tr>
<tr>
<td>![Control Icon]</td>
<td>Click to enable <strong>Draw Boundary</strong> mode. The cursor changes to a crosshair and you can drag a marquee in the Viewer to create a custom grid.</td>
</tr>
<tr>
<td>![Control Icon]</td>
<td>Click to subdivide the grid <strong>columns</strong> across the currently selected area.</td>
</tr>
<tr>
<td>![Control Icon]</td>
<td>Click to subdivide the grid <strong>rows</strong> across the currently selected area.</td>
</tr>
<tr>
<td>![Control Icon]</td>
<td>Click to subdivide the grid <strong>columns</strong> and <strong>rows</strong> across the currently selected area.</td>
</tr>
</tbody>
</table>
5. Modify the grid around the area you want to warp. Usually, you want the grid to conform to the subject of the source image. For example, if you are warping an animal's eyes, you need to create grid lines that follow the edges of the eyes.

**Note:** If you have both grids visible when you move a point, and the same point for both grids are on top of each other, both points are moved and you won't see any distortion.

You can use the grid lines to isolate the areas you do not want to warp. You do this by adding grid lines between the area you intend to warp and the area you don't want to change.

![An unlimited warp.](image1)

![The same warp limited to a small area with grid lines.](image2)

When you select a point, four tangent handles appear around it. You can use these handles to modify the curves connecting the points.

![To move several points together, draw a marquee around them and use the transformation overlay that appears.](image3)
You can also use the **Draw Boundary** tool in the Viewer to quickly set a user defined grid. Click **Draw Boundary** and draw a marquee over the required area of the image.

![The Draw Boundary marquee.](image)

![The resulting user-defined grid.](image)

**Note:** You can also use the Curve Editor to edit curves by right-clicking a control point and selecting **Curve Editor > points, tangents, or both.**

The curves appear in the Dope Sheet as long as the GridWarp **Properties** panel is open.

6. In the areas where you want to warp the image, drag the points on the grid to a new position. When you click on a point, the point changes color to indicate whether it's in focus (green by
default) and whether it has an expression set (red by default). You can change the default colors on the **Viewers** tab in the Preferences.

The pixels in these areas are moved in the direction you moved the points. Pixels in the nearby areas are also moved to accommodate the change, but the distortion lessens the further you get from the repositioned points. If you don't want a nearby area distorted, add more grid lines between the area and the points you want to move before you drag the points to a new location.

**Tip:** You can nudge selected control points by a single pixel using the numeric pad 4 and 6 to nudge left and right, or 8 and 2 to nudge up and down.

Holding down **Shift** and using the numeric pad moves the selected points by 10 pixels, for example, **Shift+6** moves the selected points 10 pixels to the right.

7. To better see what the warped image looks like, press **Q** on the Viewer to toggle the overlay off.
To compare the original and warped images, press D on the GridWarp node to disable and enable it. If you see changes in the areas you don't want to warp, go back to modifying the grid.

8. If necessary, animate the grid to match any movement in the source image. For more information on how to do this, see Animating Warps.

9. You can adjust the controls described in the following table to enhance your results.

<table>
<thead>
<tr>
<th>Control</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>GridWarp Tab</td>
<td></td>
</tr>
<tr>
<td>channel</td>
<td>Sets the channels affected by the distortion.</td>
</tr>
<tr>
<td>mask</td>
<td>Connect a mask input and set the channel to use as a mask. By default, the</td>
</tr>
<tr>
<td></td>
<td>mask is limited to the non-black areas of this channel.</td>
</tr>
<tr>
<td></td>
<td>Use the checkboxes to modify the mask properties:</td>
</tr>
<tr>
<td></td>
<td>• inject - copies the mask input to the predefined mask.a channel. Injecting</td>
</tr>
<tr>
<td></td>
<td>the mask allows you to use the same mask further downstream.</td>
</tr>
<tr>
<td></td>
<td>• invert - inverts the use of the mask channel so that the mask is limited</td>
</tr>
<tr>
<td></td>
<td>to the non-white areas of the mask.</td>
</tr>
<tr>
<td></td>
<td>• fringe - blurs the edges of the mask.</td>
</tr>
<tr>
<td>background</td>
<td>The warped image is rendered on top of an unwarped background. This control</td>
</tr>
<tr>
<td></td>
<td>sets what to use as that background:</td>
</tr>
<tr>
<td></td>
<td>• on black - render the warped image on top of a constant black image.</td>
</tr>
<tr>
<td></td>
<td>• on src - render the warped image on top of the image connected to the src</td>
</tr>
<tr>
<td></td>
<td>input of the GridWarp node.</td>
</tr>
</tbody>
</table>
### Control | What it does

<table>
<thead>
<tr>
<th>Control</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>on dst</strong></td>
<td>render the warped image on top of the image connected to the <strong>dst</strong> input of the GridWarp node.</td>
</tr>
<tr>
<td><strong>on bg</strong></td>
<td>render the warped image on top of a background image connected to the <strong>bg</strong> input of the GridWarp node.</td>
</tr>
<tr>
<td>background mix</td>
<td>Blends between the output of the GridWarp node (at 0) and whatever you have selected from the <strong>background</strong> dropdown menu (at 1).</td>
</tr>
<tr>
<td>Set bbox to</td>
<td>Sets the boundary box properties.</td>
</tr>
</tbody>
</table>

### Render Tab

- **submesh resolution**
  - Sets the number of subdivisions that are created between Bezier curves in the grid. The higher the value, the more accurate the distortion between the grid lines, but rendering time increases.

- **filter**
  - Select the appropriate filtering algorithm (see [Choosing a Filtering Algorithm](#)).

### Options Tab

- **source color**
  - Sets the source grid color.

- **destination color**
  - Sets the destination grid color.

---

**Warping an Image Using the SplineWarp Node**

The SplineWarp node deforms an image based on multiple Bezier or B-spline curves that you create. Source curves define where to warp from, while destination curves define where to warp the source image to. Unlike the GridWarp node, you can draw these curves anywhere on the image, rather than only adding points on the existing grid lines, then join them to create the source and destination. The controls for adding and modifying points are similar to the RotoPaint node controls.

**To Warp an Image Using the SplineWarp Node**

1. Select **Transform > SplineWarp** to insert a SplineWarp node after the image you want to warp.
2. Connect the **A** input to the image and attach a Viewer to the SplineWarp node.
3. Use the Viewer tools to control the following:

<table>
<thead>
<tr>
<th>Control</th>
<th>Name</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="autokey.png" alt="autokey" /></td>
<td>autokey</td>
<td>When enabled, changes to points and shapes are automatically keyed. You can disable this and set your keyframes manually, which is particularly useful if you intend to use the Curve Editor. See Animating Warps for more information.</td>
</tr>
<tr>
<td><img src="label_points.png" alt="label points" /></td>
<td>label points</td>
<td>When enabled, points selected in the Viewer are numbered sequentially.</td>
</tr>
<tr>
<td><img src="hide_curves.png" alt="hide curves" /></td>
<td>hide curves</td>
<td>When enabled, curve lines are hidden.</td>
</tr>
<tr>
<td><img src="hide_points.png" alt="hide points" /></td>
<td>hide points</td>
<td>When enabled, points are hidden.</td>
</tr>
<tr>
<td><img src="hide_transform_handle.png" alt="hide transform handle" /></td>
<td>hide transform handle</td>
<td>When enabled, the transform handle is hidden when a selection is made.</td>
</tr>
<tr>
<td><img src="hide_transform_handle_when_moved.png" alt="hide transform handle when moved" /></td>
<td>hide transform handle when moved</td>
<td>When enabled, the transform handle is hidden when a selection is moved.</td>
</tr>
<tr>
<td><img src="constant_selection.png" alt="constant selection" /></td>
<td>constant selection</td>
<td>When enabled, clicking in the Viewer does not deselect the current shape(s).</td>
</tr>
<tr>
<td><img src="ripple.png" alt="ripple" /></td>
<td>ripple</td>
<td>Rippling keyframes allows you to adjust the position of a stroke/shape point on one frame and have that same relative adjustment applied to the point across all frames or a specified range of frames.</td>
</tr>
<tr>
<td>Control</td>
<td>Name</td>
<td>What it does</td>
</tr>
<tr>
<td>---------</td>
<td>-----------------------------</td>
<td>------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>add/remove keyframe</td>
<td>Add and remove keyframes at the current playhead position using these buttons.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>output A</td>
<td>When enabled, output the image attached to the A input and any shapes in the curves list labeled with A.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>output A warped</td>
<td>When enabled, output the image attached to the A input warped by any shapes in the curves list labeled with A.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>output AB morph</td>
<td>When enabled, output the morphed A and B inputs and any shapes in the curves list labeled with AB.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>output B</td>
<td>When enabled, output the image attached to the B input and any shapes in the curves list labeled with B.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>output B warped</td>
<td>When enabled, output the image attached to the B input warped by any shapes in the curves list labeled with B.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>show source curves</td>
<td>When enabled, joined source curves are displayed in the Viewer.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>show destination curves</td>
<td>When enabled, joined destination curves are displayed in the Viewer.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>show correspondence points</td>
<td>When enabled, all correspondence points are displayed in the Viewer when a Correspondence tool is selected.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>show boundaries</td>
<td>When enabled, all boundary curves are displayed in the Viewer.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>show hard boundaries</td>
<td>When enabled, all hard boundary curves are displayed in the Viewer.</td>
</tr>
<tr>
<td>Control</td>
<td>Name</td>
<td>What it does</td>
</tr>
<tr>
<td>---------</td>
<td>-----------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Note:</strong> Hard boundary is only useful on closed curves.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>visibility</strong> Controls visibility for the selected shape.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>lock</strong> Controls the lock state of the selected shape. Locked shapes cannot be modified.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>toggle boundary</strong> Toggles the Boundary state of the selected shape(s). Clicking <strong>toggle boundary</strong> on a standard shape defines it as a Boundary and vice versa. See Using Boundary Shapes and Pins to Limit Warp for more information.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>update mode</strong> Sets the Viewer update mode:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• <strong>off</strong> - no updates are applied when points are moved. Use this option for complex warps where the Viewer update can be slow.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• <strong>on</strong> - the Viewer updates in real-time to help you position points. With complex scripts, this mode maybe prohibitively slow.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• <strong>persistent</strong> - similar to <strong>on</strong>, but displays a preview until the render is complete, rather than showing real-time scanlines.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Select tool</strong> Click to enable <strong>Select</strong> mode allowing you to select points and shapes, or marquee select multiple points and shapes.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Points tool</strong> Click to enable <strong>Add</strong> mode or toggle between <strong>Add</strong>, <strong>Remove</strong>, <strong>Cusp</strong>, <strong>Smooth</strong>, <strong>Open/Close</strong>, and <strong>Remove Destination</strong> modes.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Click on existing shapes to add or modify points.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Draw tool</strong> Click to enable <strong>Draw</strong> mode or toggle between <strong>Bezier</strong>, <strong>Cusped Bezier</strong>, <strong>B-spline</strong>, <strong>Ellipse</strong>, <strong>Rectangle</strong>, and <strong>Cusped Rectangle</strong> mode.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Click on the Viewer to draw the selected shape.</td>
</tr>
<tr>
<td>Control</td>
<td>Name</td>
<td>What it does</td>
</tr>
<tr>
<td>---------</td>
<td>-----------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
</tbody>
</table>
| ![Correspondence tool](image) | Correspondence tool   | Click to enable **Correspondence** mode or toggle between **Add**, **Modify**, and **Remove** mode:  
- **Add** - clicking on a joined curve creates a correspondence point. These points are used to improve the warp accuracy in isolated areas, but overuse can cause performance issues.  
- **Modify** - allows you to move an existing correspondence point along its curve, in either direction, as far as the next point on the shape.  
- **Remove** - click to remove an existing correspondence point. |
| ![Pin tool](image)          | Pin tool              | Click to enable **Pin** mode. You can use pins as single point joined curves to create warp or to secure specific points in place to isolate them from warping.                                                      |
| ![Join tool](image)         | Join tool             | Click to enable **Join** mode or toggle between **Join**, **Reverse**, and **Unjoin** mode.                                                                                                                    |

4. Click ![A](image) to select source warped mode.  
5. Select the **Draw** tool ![Draw](image) and draw a source, or warp from, curve as shown below.

**Note:** If you're creating multiple open splines, draw your first spline and then press **Esc** or select another tool to let SplineWarp know you're done. Reselect **Draw** to begin drawing another spline.
6. You can modify shapes by adding points using the Add tool and adjusting the tangent handles that appear around it.

When you mouse over a point, it turns green. Clicking on a point changes its color to indicate it's selected (white by default) and whether it has an expression set (red by default). You can change the default colors on the Viewers tab in the Preferences.

To move several points together, draw a marquee around them and use the transformation overlay that appears.

To close an open curve or open a closed curve, right-click on a point and select open/close curve.

To remove a point, right-click on the point and select delete.

**Note:** You can also use the Curve Editor to edit curves by right-clicking a control point and selecting curve editor > points or points+tangents.

The curves appear in the Dope Sheet as long as the SplineWarp Properties panel is open.

7. Once you're happy with the source curve, either:
   
   • draw a destination curve and use the Join tool to connect the two curves.
   
   OR
   
   • right-click a point on the source curve and select duplicate in A and join to produce the destination curve.

8. Select the destination curve and drag points or the entire shape to a new position. The pixels in these areas are warped in the direction of movement.
9. To better see what the warped image looks like, press Q on the Viewer to toggle the overlay off. To compare the original and warped images, press D on the SplineWarp node to disable and enable it. If you see changes in the areas you don't want to warp, go back to modifying your shapes.

10. Using the curves list, you can quickly build up a complex warp hierarchy using the add/remove buttons.

Using the root, layer, and pair warp controls, you can affect different elements as required:

- **root warp** - affects warp globally, irrespective of layer and pairing.
- **layer warp** - affects warp on the selected layers only, irrespective of pairing. This control is disabled if anything other than layers is selected.
- **pair warp** - affects warp on the selected pairings only. This control is disabled if anything other than pairs is selected.

**Note:** The pair warp value is multiplied by the layer warp value, which is in turn multiplied by the root warp value.

11. If necessary, animate the source warped shapes to match any movement in the source image. For more information on how to do this, see Animating Warps.

12. You can adjust the controls described in the following table to enhance your results.
<table>
<thead>
<tr>
<th>Control</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>SplineWarp tab</strong></td>
<td></td>
</tr>
</tbody>
</table>
| crop to format | When disabled, the input image is not cropped to the Project Settings > full format size. As a result, warping can introduce pixels from the image outside the format size, rather than black.  

**Note:** Disabling crop to format can affect performance as Nuke has to calculate the warp for the entire image, not just the area inside the format size. |
| bbox boundary curve | When enabled, a boundary curve is added around the input format, limiting warp at the edges of the image.  

**Note:** In AB morph mode, the format of input A is taken as the boundary. |

| **Render Tab** | |
| curve resolution | Adjusts the accuracy of the warp/spline match. Higher values increase accuracy, but sacrifice speed and vice versa. For example, with open splines at low curve resolution, image tearing may occur. You can raise the curve resolution value to compensate for tearing, but the render times increase.  

**Note:** Correspondence points may be used to improve the warping accuracy in a specific part of the curve if turning this value up too high causes performance problems. |
| boundary curve resolution | Adjusts the number of interpolated points on boundary and hard boundary curves. Higher values stop the warp from filtering through the boundary curves but sacrifice speed, and vice versa. |
| preview resolution | Improves the accuracy of the preview at higher values and the rendering speed at lower values. |
| Classic warping | Set the type of warp function to use:  

- **disabled** - uses an updated quadratic warp function that copes well with |
<table>
<thead>
<tr>
<th>Control</th>
<th>What it does</th>
</tr>
</thead>
<tbody>
<tr>
<td>filter</td>
<td>Select the appropriate filtering algorithm (see Choosing a Filtering Algorithm).</td>
</tr>
<tr>
<td></td>
<td>overlapping control points, but has a more local warp effect.</td>
</tr>
<tr>
<td></td>
<td>• <strong>enabled</strong> - employs a warping function from previous versions of Nuke that has a more global warp effect, but doesn't cope well with overlapping control points.</td>
</tr>
</tbody>
</table>

**Using Boundary Shapes and Pins to Limit Warp**

You may notice that pixels in areas around warping are also moved to accommodate the change, but the distortion lessens the further you get from the repositioned points. If you don't want a nearby area distorted, you can:

• Designate a curve or shape as a boundary to control or eliminate warp.
• Place pins in the areas you want to limit warp.

**Note:** Boundaries only affect the input they are drawn on, A or B, not both.

• **Boundary** - generally open splines helping to control warp rather than eliminate it.

To define a boundary for an existing curve, enable **boundary** in the curves list, as shown below.

Using the example curves from Mr. Lion above, you could control the warp by adding a curve and designating it as a boundary:
Warping | Warping an Image Using the SplineWarp Node

The source and destination curves.
The same curves, but with a boundary above the destination to limit warp.

- **Hard Boundary** - closed shapes that eliminate warp outside the area that they enclose.

To define a hard boundary for an existing shape, enable **hard boundary** in the curves list, as shown below.

The effect of a hard boundary on a warp is quite distinct - the hard boundary, colored white by default, limits the warp effect to entirely within the closed shape:

- **Pins** - placing a pin creates a single point source and destination curve, effectively limiting warp in a very localized way.

The effect of a pin on a single joined curve is shown below. The warp direction is shown on the left and the pin's effect on the right.
Morphing refers to dissolving two images together so that the subject of one image seems to change shape and turn into the subject of the other through a seamless transition. A morph can be easily noticeable or very subtle. An example of a noticeable morph would be a man turning into a woman or one animal turning into another, whereas a transition from an actor to his stunt man would result in a much more subtle morph.

An image of a monkey turning into an image of a lion.

Morphing can be a time-consuming task, but it can be made easier by good advance planning of the shots. The more similar the characteristics, position, and movement of the subjects you want to morph are, the easier it is to morph them together. If the position and the movement of the subjects do not match, however, you can try to reposition and retime the clips before morphing them together. To do so, use the nodes described in Transforming Elements and Temporal Operations. You can also use the Tracker and PlanarTracker nodes to track the features you want to morph over time or to stabilize your clips before morphing them together. See Transforming Warps for more information.

Below, we first discuss morphing images using the GridWarp node and then using the SplineWarp node.
Morphing One Image into Another Using the GridWarp Node

To morph one image into another using the GridWarp node:
1. Select **Image > Read** to import the two images you want to morph together.
2. If the images do not have the same resolution, insert Reformat nodes after them. For more information, see Reformatting Image Sequences or Reformatting Elements.
3. Select **Transform > GridWarp** to insert a GridWarp node into your script.
4. Connect the source image (the image you want to turn into another image) to the **src** input of the GridWarp node, and the destination image (the image you want the source image turned into) to the **dst** input. Connect a Viewer to the GridWarp node.

![GridWarp node diagram]

5. To make the grids the same size as the input images, click the **Resize to Image** buttons under both **Source Grid** and **Destination Grid**.

![GridWarp settings]

6. In the GridWarp **Settings** controls, set **output** to **source** to display the source image and grid.
7. Adjust the grid to follow key features on the **source** image. For example, if you’re morphing an animal, you might consider the eyes, nose and mouth.
8. In the GridWarp **Settings** controls, set **output** to **destination** to display the destination image and grid.

**Note:** If the **source** and **destination** images are similar, you can copy the **source** grid onto the **destination** grid using the copy and paste buttons.

9. Adjust the grid to follow key features on the **destination** image.

10. In the GridWarp **Settings** controls, set **output** to **morph** to display both grids and activate the **warp** and **mix** sliders.
11. Scrub to the frame where you want the morph to begin. Bring the **warp** slider down to 0 (the source image). Click on the animation button and select **Set key**. Then, scrub to the frame where you want the morph to end and set **warp** to 1 (the destination image).

12. Animate the **mix** control in the same way. Scrub to the frame where the morph starts, and set **mix** to 0 (the source image). Click the animation button and select **Set key**. Scrub to the frame where the morph ends, and enter 1 (the destination image) as the **mix** value.

Play through the sequence in the Viewer and you'll see that the **source** image is morphed into the **destination** image.

---

**Morphing One Image into Another Using the SplineWarp Node**

To morph one image into another using the SplineWarp node:

1. Select **Image > Read** to import the two images you want to morph together.
2. If the images do not have the same resolution, insert Reformat nodes as required. For more information, see [Reformatting Image Sequences](#) or [Reformatting Elements](#).
3. Select **Transform > SplineWarp** to insert a SplineWarp node into your script.
4. Connect the source image (the image you want to turn into another image) to the A input of the SplineWarp node, and the destination image (the image you want the source image turned into) to the B input. Attach a Viewer to the SplineWarp node.

5. In the SplineWarp node's controls, switch **output** between A and B or click A and B in the Viewer tools. You should see both images in the Viewer.

6. Identify some features that are similar in the source and the destination images. For example, if you are morphing together images of people or animals, these features might include their eyes, noses and mouths as well as the outlines of their faces and heads.

7. Switch to input A to draw curves around the key features you identified in the previous step. For more information on how to create the curves, see [Warping an Image Using the SplineWarp Node](#).
You may find it useful to name the entries in the curves list, left_eye, right_eye, etc. Meaningful names can help you later on when you're using the Join tool to link the source and destination shapes.

8. Switch to input B and draw curves in a similar fashion. Draw the same amount of curves, on similar features, to make the morph as accurate as possible.

9. Switch the Viewer to output ABmorph, which displays both sets of curves, and activate Join in the Viewer tools.

10. Work through the A input shapes, linking them to their B input counterparts. Try to link similar points, such as the corners of the eyes or the side of the mouth.

Note: You can invert the source and destination shapes using the Reverse tool.
As you join shapes, the curves list updates showing the linked pairs.

11. Scrub to the frame where you want the morph to begin. Bring the mix slider down to 0 (input A).

   Click on the animation button \(\text{\textbullet}\) and select Set key.

12. Set keyframes for root warp in the same way.

13. If you’re using stills, as in this example, scrub the playhead to the number of frames required for the morph. Otherwise, scrub to the end of the sequence.

14. Set the mix and root warp sliders to 1 (input B).

15. Press Q in the Viewer to disable the spline overlay and press play.

   Input A is morphed into input B.

Using the cookie-cutter to create traveling masks

The curves list cookie-cutter is primarily designed to overlay a morph between two images, which may not include an alpha channel, on top of a sequence by creating a “traveling mask” automatically between closed cookie-cutter shapes.

For example, the monkey-lion morph demonstrated previously could be merged with a sequence, replacing a face with a monkey face that is morphed into a lion over time.
Note: The cookie-cutter only works with closed shapes from the curves list.

1. Designate a pair of closed shapes as cookie-cutters by clicking the cookie-cutter icon in the curves list.

   ![Curves List with Cookie Cutters](image)

   In this instance, enabling cookie-cut for the monkey's head, m_head, also designates the lion's head, l_head, as a cookie-cutter shape.

2. Use the output mask dropdown to select the channel where the traveling mask is stored, or use the mask_splinewarp.a default.

   ![Output Mask Dropdown](image)

3. Animate the morph to follow the required background element as described in Transforming and Animating Warps and Morphs.

4. Finally, merge the output of the SplineWarp node with your sequence. A simple script might appear as follows:
You can confirm the presence of the traveling mask by selecting the **output mask** channel in the Viewer dropdown and then pressing M to display the **Mat** channel. You should see an automatically created mask that matches the morphed shape.
Transforming and Animating Warps and Morphs

Unless you are warping or morphing a still image, you’ll probably need to transform or animate warp operations. The Tracker and PlanarTracker nodes can help automate transformations, and you can use Expressions and the GridWarp and SplineWarp Properties panel controls as well. See Transforming Warps and Animating Warps.

Transforming Warps
You can transform grids and splines in the same way as any other element in Nuke (see Temporal Operations), but you can also link control points to the Tracker and PlanarTracker nodes to automate any transforms you require. Once you have a suitable track, you can use the animation data to control points on grids and curves.

Using the Tracker Node
1. To generate tracking data, see Tracking and Stabilizing.
2. Make sure both the Tracker and the GridWarp/SplineWarp properties panels are open, then either:
   • Ctrl/Cmd+drag the animation button from the Tracker node properties on top of the individual grid/curve point in the Viewer. When you release the mouse, the point follows the animation from the Tracker node,
   OR
   • Right-click on the required control point and select Link to > Tracker1 and select the required transform type.

Using the PlanarTracker Node
1. To generate PlanarTracker data, see Tracking with PlanarTracker.
2. Use the Draw Boundary button to create a source grid roughly the same shape as the tracked plane, or draw similar splines.
3. On the GridWarp/SplineWarp and RotoPaint Transform tabs, open up the Transform tab extra matrix.
4. On the RotoPaint Transform tab, select PlanarTrackerLayer1 in the curves list.
5. Copy and paste the **extra matrix** animation from RotoPaint to the GridWarp/SplineWarp **extra matrix**.

6. Play through the sequence to check that the grid or splines follow the plane.

### Using Expressions with GridWarp

As well as using other nodes to control points, you can do the inverse, using expressions to link control points and their tangents to other operations.

The expressions take the following form:

- `GridWarp3_1.destination_grid_col.1.2.pos.x`
- `GridWarp3_1.destination_grid_col.2.2.tangent.1.x`

**Note:** Control points are indexed starting from the bottom left 1.1, in x.y order, and tangent points are labelled 0, 1, 2, and 3 corresponding to north, south, east, and west respectively.

It's worth mentioning that in order to get the true position of a tangent, not its offset value, you need to concatenate the `pos` and `tangent` expressions. For example:

```
GridWarp3_1.destination_grid_col.2.2.pos.x + GridWarp3_1.destination_grid_col.2.2.tangent.4.x
```
Note: You can only modify tangent positions using the Curve Editor.

For more information, refer to Expressions.

### Animating Warps

In the GridWarp and SplineWarp node's properties panels, there are controls for animating both the source and the destination grids or shapes. Here, we first look at the GridWarp node and then the SplineWarp node. The instructions assume you know how to warp a still image using these nodes (if not, refer to Warping and Warping an Image Using the SplineWarp Node).

#### To Animate a Warp Using the GridWarp Node

1. While viewing the source image, adjust the source grid as necessary (see Warping for information on how to do this).
   - If **autokey** is enabled, key frames are added every time you adjust the grid. Otherwise, click the **add key** button \( \text{瓒} \) under **Source Grid**. This saves the current grid as a key shape.
2. Move to a new frame and adjust the source grid accordingly. A new key shape is set automatically.
3. Repeat the previous step as necessary. If you need to delete a key shape, scrub to the frame where you set it and click **delete key** \( \text{删} \) under **Source Grid**.
4. Hide the source grid and display the destination grid.
5. While viewing the output of the GridWarp node, adjust the destination grid until you are happy with the warp.
   - If **autokey** is enabled, key frames are added every time you adjust the grid. Otherwise, click the **add key** button \( \text{瓒} \) under **Destination Grid**. This saves the current grid as a key shape. Click the **set** button under **Destination Grid**. The current grid is saved as a key shape.
6. Move to a new frame and adjust the destination grid again. The modified grid is automatically set as a key shape.
7. Repeat the previous step until you are happy with the animated warp.

#### To Animate a Warp Using the SplineWarp Node

1. Create the required curves as described under Warping an Image Using the SplineWarp Node. If **autokey** is enabled, key frames are added every time you adjust a shape. Otherwise, click the **add key** button \( \text{瓒} \) with **A warped** selected. This saves the current shape as a key frame.
2. Move to a new frame and adjust the shapes accordingly. New key shapes are set automatically.
Repeat the previous step as necessary. If you need to delete a key shape, scrub to the frame where you set it and click delete key with A warped selected.

**Linking Transforms Using the SplineWarp Node**

Transforms applied in the properties panel Transform tab can be expression linked with other shapes in the curves list using the transform linked toggle.

1. Create the required transform on the source shape.
2. Click the transform linked toggle in the curves list.
   A menu displays the available shapes and transform types.

![Image of menu displays the available shapes and transform types.](image)

You can link all transform types associated with the source shape or be selective, such as all but extra matrix.

3. In the example, any scale keyframes added to the l_left_nose shape are applied equally to the l_head shape.
4. If you want to remove a link, click the transform linked toggle for the required shape, select the link, and click remove link.
Temporal Operations

This chapter explains the temporal or time-based operations in Nuke. Learn how to distort time (that is, slow down, speed up, or reverse clips), apply motion blur, and perform editorial operations like slips, cuts, splices, and freeze frames.

Quick Start

Here's a quick overview of the workflow:

1. If you want to speed up or slow down your clip:
   • You can use the Retime node (**Time > Retime**). If you want, you can also add a FrameBlend (**Time > FrameBlend**) node before your retime to create smoother retiming results. For more information, see Simple Retiming and Interpolation.
   • For more advanced retiming operations and adding motion blur, you can use the OFlow node (**Time > OFlow**), or if you have a license for NukeX, the Kronos node (**Time > Kronos**). For more information, see OFlow Retiming and Kronos.
   • Sometimes you'll want to create speed-up and slow motion within a clip without altering its actual length. This is called warping and you can use the Retime node (**Time > Retime**) to do it. For more information, see Warping Clips.

   After retiming your clip, you may need to adjust the script's global frame range and playback speed (frames-per-second). For more information, see Global Frame Range and Speed.

2. If necessary, you can use the TimeBlur node to apply motion blur to garbage masks that are tracked to a fast moving feature. For more information, see Applying a TimeBlur Filter.

3. If you're looking to perform editorial operations:
   • You can use the TimeOffset (**Time > TimeOffset**) and TimeClip (**Time > TimeClip**) nodes to slip clips. For more information, see Offsetting and Slipping Clips.
   • The FrameRange (**Time > FrameRange**) node allows you to cut clips. For more information, see Cutting Clips.
   • You can use the AppendClip (**Time > AppendClip**) node to splice your clips. For more information, see Splicing Clips.
   • Using the FrameHold (**Time > FrameHold**) node allows you to set the output of a clip or scene to a particular frame or frame interval. For more information, see Freeze Framing Clips.
**Note:** Most temporal operations in Nuke can only be applied to 2D compositions, but the FrameRange, TimeOffset, and FrameHold nodes are applicable to 3D nodes, such as geometry.
Distorting Time

Time distortion changes the length of time required to playback a clip in your composite. These operations generally fall under one of two categories: *retiming* and *warping*.

Retiming is the process of slowing playback by adding frames, or accelerating playback by subtracting frames.

Warping is the process of slowing down, speeding up, or even reversing playback on a clip without necessarily altering the overall length.

**Tip:** When working with temporal operations, it helps to attach a Viewer to the retiming or warping node so you can see the effect of your changes.
Simple Retiming

Nuke’s Retime node lets you change the playback time for all the frames in a clip or for range of frames within the clip. You can also use it to reverse the clip playback. It does this by dropping or duplicating frames. For higher quality retiming see OFlow Retiming.

To Retime All Frames in a Clip
1. Select Time > Retime to insert a Retime node into your script.
2. Enter a value in the speed parameter. Values higher than 1 increase playback speed; values less than 1 decrease playback speed.
3. Check the reverse box if you want to play the clip backwards - making the last frame the first, the first frame the last, and so on.
4. Increase the shutter parameter to enable frame-blending (For more information, see Interpolation).

To Retime a Range of Frames in a Clip
1. Select Time > Retime to insert a Retime node into your script.
2. Check the boxes for input range and enter the “in” and “out” frames.

For example, if your original clip is 50 frames, but you want to only retime the last half, you would input 25 for the in point and leave the out point at 50.
3. Check the box for output range and enter the “in” and “out” frames to retime to a specific clip length.
   or
   Enter a factor in the speed parameter and Nuke calculates the output range for you. Values higher than 1 increase playback speed; values less than 1 decrease playback speed.
4. Check the reverse box to invert the selected frame range.
5. Increase the shutter parameter to enable frame-blending.
Interpolation

Time distortions that slow down a clip require the creation of additional, or *interpolated*, frames. For example, suppose you want to slow down the collision, shown in the clip below, by a factor of two. This requires the creation of one interpolated frame for every existing frame in the clip.

![Interpolating frames.](image)

The simplest way for Nuke to interpolate is to duplicate existing frames and increase the length of the clip - this is the default method of interpolation. However, this method can create jittery playback, especially when the image depicts very fast motion and the clip is retimed to be considerably longer than its original length. For such cases, Nuke provides different nodes for smoothly interpolating the frames.

Frame-Blending

The FrameBlend node interpolates frames by generating an additive composite of the frames that precede and follow it, rather than creating mere copies between the existing frames.
Here is an example of the difference between frame-copy and frame-blend interpolation. In the first frame of the figure below, you see a copy of the preceding frame. In the second frame, you see a new image generated by blending the previous and subsequent frames.

Frame-copied versus frame-blended interpolation.

The latter method creates “ghosting” around all fast moving features (the window frame and the pages on the desk, for example). This may look odd when viewed as part of a still frame, but contributes to smoother motion during actual playback.

You can enable frame-blending by manipulating the shutter value of a retiming node. Higher shutter values generate more frame-blending. Or, you can insert a FrameBlend node before the temporal effect you want to influence. The below figure shows an example of frame-blending with the Retime node.

To Insert a FrameBlend Node

1. Select Time > FrameBlend from the toolbar.
   Remember to place it upstream from the temporal effect you want to influence.
2. Enter the number of frames to blend in the **Number of frames** field.
   
   or

   Check the **Custom** box and enter the starting and ending frames that you want to blend. To use the input range as your custom frame range, click **Input Range**.

3. If necessary, check **Foreground matte** and select the channel to limit the blending effect.

The **output image count to** option saves a floating point alpha image to a channel you specify; the result indicates the number of images that contributed to each pixel of the matte. To normalize the alpha, divide the number 1 by the number of frames averaged, and then multiply the alpha channel by this result. You can also use the inverse of this matte for additional degraining.
OFLOW Retiming

The OFLOW node generates high-quality retiming operations analyzing the movement of all pixels in the frames and then rendering new “in-between” images based on that analysis. This node can also add motion blur or enhance the existing motion blur in the image.

Tip: We recommend using NukeX’s Kronos node instead, as it includes an improved motion estimation algorithm. For more information, see Kronos.

To Retime with OFLOW

1. Select Time > OFLOW to insert an OFLOW node after the sequence you want to retime. The Input Range is set automatically by the source clip when you first create the node. After that, it is only updated if you click Reset.
2. If you intend to use Output Speed or Frame timing, attach a Viewer to the output of the OFLOW node, OR
   If you intend to use Input Speed timing, attach a Viewer before the OFLOW node.
3. By default, OFLOW is set to perform a linear half-speed slow down using Timing > Output Speed.
4. Enable Use GPU if available to render output on the Local GPU specified, if available, rather than the CPU. For more information on the minimum requirements, please see Windows, Mac OS X and macOS, or Linux or refer to the Nuke Release Notes available in Help > Release Notes.
5. Select the channels you want to apply the retime to using the channels dropdown. OFLOW retimes all channels by default.
6. To adjust the slow down or to speed the sequence up, enter a new value for the Output or Input Speed control. Values below 1 slow down the clip. Values above 1 speed up movement. The default value, 0.5, creates the half-speed slow down. Quarter-speed would be 0.25.
7. Alternatively, you can describe the retiming in terms of ‘at frame 100 in the output clip, I want to see frame 50 of the source clip’. To do so, set Timing to Source Frame. Go to a frame in the timeline, and set Frame to the input frame you want to appear at that output position. You’ll need to set at least two key frames for this to retime the clip. For example, to slow down a 50 frame clip by half, you can set Frame to 1 at frame 1, and to 50 at frame 100. To do a four times slow down, set Frame to 1 at frame 1, and to 25 at frame 100.
8. If you’d like to add motion blur to your retimed footage, proceed to Adding Motion Blur.
Tip: Some Nuke users may remember nodes called OpticalFlow and OptiFlow. OFlow replaces these nodes. It can be used for retiming and adding motion blur, but it does not have the capability to output motion vectors. To output motion vectors, you could use VectorGenerator (included in NukeX and Nuke).

For retiming and motionblur, we recommend using the Kronos node instead of OFlow. For more information, see Kronos.

Refining the Results

To refine the results:

1. To have the motion vectors displayed in the Viewer, expand the Advanced control and enable Overlay Vectors. Forward motion vectors are drawn in red, and backward motion vectors in blue.

   Overlay Vectors enabled.

   Note: Motion vectors displayed in the Viewer are added to your output if you don't turn off the overlay before rendering.

2. To set the spacing between motion vectors displayed on the Viewer, adjust Vector Spacing. The default value of 20 means every 20th vector is drawn. Note that Vector Spacing only affects the Viewer overlay rather than the retimed result.
3. Adjust **Vector Detail** to vary the density of the vector field. The larger vector detail is, the greater the processing time, but the more detailed the vectors should be. A value of 1 generates a vector at each pixel. A value of 0.5 generates a vector at every other pixel. For some sequences, a high vector detail near 1 generates too much unwanted local motion detail, and often a low value is more appropriate.

Areas of unwanted local motion detail. Lower **Vector Detail** is more appropriate in this case.

4. Vector fields usually have two important qualities: they should accurately match similar pixels in one image to another and they should be smooth rather than noisy. Often, it is necessary to trade one of these qualities off against the other. A high **Smoothness** misses lots of local detail, but is less likely to provide you with the odd spurious vector. A low **Smoothness** concentrates on detail matching, even if the resulting field is jagged. The default value of 1.5 should work well for most sequences.
Jagging as a result of a low Smoothness value.

Minimal jagging as a result of a high Smoothness value.

5. Set the type of resampling applied when retiming:
   - Bilinear - the default filter. Faster to process, but can produce poor results at higher zoom levels. You can use Bilinear to preview a retime before using one of the other resampling types to produce your output.
   - Lanczos4 and Lanczos6 - these filters are good for scaling down, and provide some image sharpening, but take longer to process.

6. If the overall brightness in your Source footage changes between frames, enable Flicker Compensation. This allows OFlow to take into account variations in luminance and overall flickering, which could otherwise cause problems with your output.

   Examples of variable luminance include highlights on metal surfaces, like vehicle bodies, or bodies of water that reflect light in unpredictable ways.

   Note that using Flicker Compensation increases rendering time.

7. In order to reduce processing time, much of the motion estimation is done on luminance only - that is, using monochrome images. In most cases this is perfectly acceptable, but the parameters in the Tolerances group allow you to concentrate on a particular feature in an image by adding bias to individual colours. You may, for example, wish to increase Weight Red to allow the algorithm to concentrate on getting the motion of a primarily red object correct, at the cost of the rest of the objects in a shot.
Warping Clips

Warping refers to slowing down, speeding up, or even reversing select frames in a clip without necessarily altering its overall length. Otherwise stated, warps add, subtract, or reverse the temporal detail in a range of frames within a clip. For example, the figure below depicts a snowmobile clip (down-sampled to just ten frames for easy representation) that we might want to warp.

One way - in fact, kind of the classic way - to warp this clip would be to play the frames just prior to the collision at their original rate, the frames involving the collision in slow motion, and the frames after the collision in fast motion.

You could achieve such a warp by sculpting the curve in Nuke’s TimeWarp curve, which is a part of the Retime node’s parameters, to look something like the one depicted below.
The basic “rules” for editing the warp curve are as follows:
• To slow down motion, decrease the slope of the curve.
• To speed up motion, increase the slope of the curve.
• To reverse motion, create a downward sloping portion on the curve (a dip, in other words).

To Warp a Clip
1. Click **Time > Retime** to insert a Retime node into your script.
2. Click the **TimeWarp** tab to reveal the TimeWarp curve.
3. Attach a Viewer to this node, so you can see the effect of your changes.
4. Sculpt the TimeWarp curve according to the rules above. (**Ctrl/Cmd+Alt** click to insert keyframe knots on the curve; **Ctrl/Cmd+drag** to reposition keyframe knots; **Ctrl/Cmd+drag** to rotate a keyframe knot control handles.)
5. If you want to enable frame blending on the output, either input a value larger than one in the Retime node’s **shutter** parameter, or insert a FrameBlend node prior to the Retime node.
Global Frame Range and Speed

Nuke automatically adjusts the timeline of every Viewer window you open to show the "in" and "out" frames for the clip you're viewing.

After you retime a clip in your compositing script, you may need to adjust the script's global frame range and playback speed (frames-per-second), to account for the retiming operations.

Adjusting the global frame range.

Select **Edit > Project Settings** (or press **S** over the Nuke window) and then enter the new **frame range** and **fps** in the Project settings properties panel.
Applying a TimeBlur Filter

When a fast-moving subject is recorded on film or video, its edges appear to smear as a result of the object's movement while the shutter is open. The longer the shutter remains open at each frame interval, the more obvious this effect. TimeBlur simulates this phenomenon by sampling its input at "divisions" times over "shutter" frames starting at the current frame plus "offset".

Time blur is commonly applied to garbage masks that are tracked to a fast moving feature. The time blur averages the incoming mask image over the shutter period, to better match the motion blur in the original image and creating a more convincing integration.

**Tip:** You can also use the Roto and RotoPaint *Global Blur* controls to achieve the same results. See *Adding Motion Blur* for more information on Roto and RotoPaint motion blur.

Applying Motion Blur to a Clip

To apply motion blur to a clip:

1. Click *Time > TimeBlur* to insert a TimeBlur node into your script. Place it downstream from the element to which you want to apply motion blur.
2. In the TimeBlur properties panel, set *divisions* to the number of times you want to sample the input over the shutter time. For images with fast-moving content higher values are necessary to eliminate "steppiness" or banding in the output.
3. Set *shutter* to equal the span of time (in frames) over which the input should be sampled. A shutter time of .5 is typical and would correspond with a camera shutter of 180 degrees.
4. Set the *shutteroffset* to control when the sampling of the input starts relative to the frame being rendered, analogous to when the camera shutter opened to capture corresponding film or video footage you might have at the same frame. You may need to adjust this by eye to align, for example, a garbage mask with an underlying feature.

**Tip:** You may find that using TimeBlur on all the upstream nodes in your composition can be unnecessary and very time consuming. In these cases, you can use NoTimeBlur node to limit the number of nodes to which you're applying TimeBlur. Just insert the NoTimeBlur node in your node tree above the TimeBlur and any nodes you want the TimeBlur node to process.
Editing Clips

As a node-based system, Nuke doesn’t have a timeline. Nevertheless, you can still perform editorial operations that you might associate with a timeline. You can slip clips (move them forward or backward in time), cut them, or splice them to other clips.

Offsetting and Slipping Clips

*Offsetting* a clip refers to moving the clip backwards or forwards in time. In Nuke, you can offset clips using the TimeOffset and TimeClip nodes.

Unlike TimeClip, the TimeOffset node can also be used with 3D nodes, for example, if you want to offset camera times.

**ToOffset a Clip Using the TimeOffset Node**

1. Click **Time > TimeOffset** to insert a TimeOffset node into your script. (Place it downstream from the element to which you want to offset.)

2. Attach a Viewer to this node, so you can see the effect of your changes.

3. In the TimeOffset properties panel, check **reverse input** if you want to invert the clip (make the last frame the first, and so on).

4. In the **time offset (frames)** field, type the number of frames by which you want to offset the clip. Enter a negative value to subtract frames from the head of the clip. Enter a positive value to add frames to the head of the clip.

5. Adjust the script length for the new output range. Select **Edit > Project Settings**, and enter **frame range** values that match the output range you specified.

**Note:** It’s not mandatory that you adjust the script’s frame range after slipping the clip. If you don’t, the Viewer fills the empty frames at the tail of the clip by holding on the last frame.

**Tip:** If you are using 3D nodes, such as cards, spheres, or cubes, together with a TimeOffset node, you can view the object in just a specified frame range by setting the **frame range** control for the object, and attaching a Viewer node to the TimeOffset node. Be sure to set the timeline range to **Input** to view the offset.
Tip: Using TimeOffset, you can also offset clips directly in the Dope Sheet. See Animating Parameters for more information.

Slipping a clip refers to changing the content of the clip that is seen, but not the position or duration. In Nuke, you can slip clips using the TimeClip node.

Just like TimeOffset, the TimeClip node also lets you move clips forwards or backwards in time and reverse the order of frames in a clip. In addition to this basic functionality, you can set the frame range for the clip, set what happens to frames outside of this frame range, fade the clip to or from black, and set expressions to adjust the node's behavior. Using TimeClip, you can also slip the clip directly in the DopeSheet, which can be handy if you prefer editing clips on a timeline.

To Slip a Clip Using the TimeClip Node
1. Click Time > TimeClip to insert a TimeClip node into your script. Place it downstream from the element that you want to adjust.
2. Attach a Viewer to the TimeClip node to see the effect of your changes.
3. In the Fade In and Fade Out fields of the TimeClip properties panel, type the number of frames, if any, you want to fade to or from black.
4. Set the new frame range for your clip by entering the first and last frame numbers.
5. In the before dropdown menu, set how Nuke should treat the frames before the first frame in the range (examples refer to a 20-frame sequence with a first frame value of 5):
   - hold - the first frame in the sequence is held until the first frame is reached. Example: 1, 1, 1, 1, 1, 2, 3, 4, etc.
   - loop - substitutes an equal number of frames, effectively creating a clip loop. Example: 17, 18, 19, 20, 1, 2, 3, 4, etc.
   - bounce - substitutes a reversed equal number of frames, creating a clip bounce. Example: 5, 4, 3, 2, 1, 2, 3, 4, etc.
   - black - frames are black until the first frame is reached.
6. In the after dropdown menu, set how Nuke should treat the frames after the last frame (examples refer to a 20 frame sequence with a last frame value of 5):
   - hold - the last frame in the sequence is held until the end of the sequence is reached. Example: 16, 17, 18, 19, 20, 20, 20, 20, etc.
   - loop - substitutes an equal number of frames, effectively creating a clip loop. Example: 16, 17, 18, 19, 20, 1, 2, 3, 4, etc.
   - bounce - substitutes a reversed equal number of frames, creating a clip bounce. Example: 16, 17, 18, 19, 20, 19, 18, 17, etc.
   - black - frames are black from last until the end of the sequence is reached.
7. Use the frame dropdown to set the frame mode:
   • **expression** - Lets you enter an expression in the field on the right. The expression changes the relation between the current frame and the frame read in.
     For example, if your clip begins from image.0500.rgb and you want to place this first frame at frame 1 rather than frame 500, you can use the expression **frame+499**. This way, 499 frames are added to the current frame to get the number of the frame that's read in. At frame 1, image.0500.rgb is read in; at frame 2, image.0501.rgb is read in; and so on.
     Another example of an expression is **frame*2**. This expression multiplies the current frame by two to get the number of the frame that's read in. This way, only every other frame in the clip is used. At frame 1, image.0002.rgb is read in; at frame 2, image.0004.rgb is read in; at frame 3, image.0006.rgb is read in; and so on.
   • **start at** - Lets you enter a start frame number in the field on the right. This specifies the frame where the first frame in the sequence is read in. In other words, all frames are offset so that the clip starts at the specified frame.
     For example, if your sequence begins from image.0500.rgb and you enter **1** in the field, image0500.rgb is read in at frame 1. Similarly, if you enter **100** in the field, image0500.rgb is read in at frame 100.
   • **offset** - Lets you enter a constant offset in the field on the right. This constant value is added to the current frame to get the number of the frame that's read in.
     For example, if your clip begins from image.0500.rgb and you want to place this first frame at frame 1 rather than frame 500, you can use **499** as the constant offset. This way, 499 is added to the current frame to get the frame that's read in. At frame 1, image.0500.rgb is read in; at frame 2, image.0501 is read in; and so on.
     You can also use negative values as the constant offset. For example, if you use the value **-10**, Nuke subtracts ten from the current frame to get the frame that's read in. At frame 20, image.0010.rgb is read in; at frame 21, image.0011.rgb is read in; and so on.

8. To reverse the clip within the specified frame range, check **reverse**.

9. While the original range fields don't actually affect the output of the node, they help make things clear in the Dope Sheet. Open the Dope Sheet and enter some values here and observe how they affect the TimeClip lines.

10. If necessary, adjust the script length for the new output range. Select **Edit > Project Settings**, and enter frame range values that match the output range you specified.

---

**Tip:** Using TimeClip, you can also offset, trim, and slip clips directly in the Dope Sheet. See Animating Parameters for more information.
Cutting Clips

The FrameRange node allows you to set a frame range for a clip. This controls which frames a downstream AppendClip node uses from the input, which frames are displayed in the Viewer when the timeline range dropdown menu is set to Input, and which frames are sent to the flipbook.

To make an edit, first use this node to cut portions out of input sequences and then append the results together using the AppendClip node.

To Cut a Clip
1. Click Time > FrameRange to insert a FrameRange node into your script.
2. Attach a Viewer to this node, so you can see the effect of your changes.
3. In the frame range fields, then enter the appropriate in and out point values.
   For example, if your original clip is 50 frames but you want to use only the last 25 frames in your composite, you would enter 25 as the In point and leave the Out point at 50.
4. Adjust the script length for the new output range. Select Edit > Project Settings, and enter frame range values that match the output range you specified.

   Note: It’s not mandatory that you adjust the script’s frame range after cutting the clip.

   Tip: Using FrameRange, you can also set the frame range for a clip directly in the Dope Sheet. See Animating Parameters for more information.

Splicing Clips

Splicing refers to joining clips head-to-tail, thus allowing action to flow from one shot to the next.
When you splice clips, you have options for:
• Fading to or from black.
• Dissolving from the first to second clip.
• Slipping the combined clip in time.

To Splice Slips
1. Click Time > AppendClip to insert an AppendClip node into your script.
2. Attach its 1 and 2 inputs to the clips you want to join. (The clip attached to input 1 precedes the one attached to input 2.)
3. Attach a Viewer to this node, so you can see the effect of your changes.
4. If necessary, expand the script length to accommodate the total length of the newly merged clip:
   • Click Edit > Project Settings. The project settings properties panel appears.
   • Enter frame range values that matches the total length.
5. In the Fade In and Fade Out fields of the AppendClip properties panel, type the number of frames, if any, you want to fade to or from black.
   For example, typing a 5 in the Fade In field would result in the following effect at the head of the merged clip.

   ![Fade In Example]

   (The inverse of this effect would occur at the tail of the merged clip were you type 5 in the Fade Out field.)
6. In the Cross Dissolve field, type the number of frames, if any, of overlap you want between the first and second clip.
   For example, leaving Cross Dissolve at the default of 0 creates a simple cut - the transition from the first to second clip is instantaneous. Typing in a 5 creates a time span of five frames in which the first clip's gain ramps downward to zero, while the second's ramps upward to 100%.

   ![Cross Dissolve Example]

   Dissolve.

   ![Cut Example]

   Cut.
7. In the First Frame field, type the number of frames, if any, by which you want to slip the clip.
   Enter a negative value to subtract frames from the head of the merged clip. Enter a positive value to add frames to the head of the clip.
8. Slipping the merged clips may create empty black frames at its head or tail. As appropriate, select First frame or Last frame if you want these empty frames to appear as copies of the first or last frame.
Freeze Framing Clips

Using the FrameHold node, you can set the output of a clip to a particular frame or frame interval, rather than the entire clip.

To Output a Freeze Frame
1. Click Time > FrameHold to insert a FrameHold node into your script.
2. Attach its input to the clip you want to affect.
3. Attach a Viewer to the FrameHold node, so you can see the effect of your changes.
4. Enter the first frame number to freeze as the output of the FrameHold node.

Note: The first frame control defaults to 0, setting the freeze frame as the first frame of the clip, regardless of the actual frame number.

To Output a Frame Interval
1. Click Time > FrameHold to insert a FrameHold node into your script.
2. Attach its input to the clip you want to affect.
3. Attach a Viewer to the FrameHold node, so you can see the effect of your changes.
4. Enter the increment frames to skip at output. For example, with a first frame of 1 and an increment value of 5, the frames passed down the node tree would be 1, 6, 11, 16, etc.
Working with Color

These pages explain how to use Nuke's color correction nodes and tools to adjust the appearance of the images in your composites. When starting out, this information provides a good overview of Nuke's scopes and color-correction nodes, however not all options are covered here.
Using Scopes

Nuke provides scopes to help you evaluate your footage. Scopes are accessed from the contents menu, Windows > New Scope sub-menu.

There are a number of global controls (Preferences > Panels > Scopes) that affect how the Scopes display information:

• black point - sets the black out of range warning level.
• white point - sets the white out of range warning level.
• luma/chroma encoding - sets the video standard to use when converting RGB to luma or chroma values in the scope displays, either REC601 or REC709.
• Include viewer color transforms - when enabled, scope data includes the applied Viewer color transforms (gain, gamma, and LUT). When disabled, scope data does not include the applied Viewer color transforms. This may slow down rendering, as it may require image calculation.
• Force full frame - When enabled, scopes display data for the full frame, regardless of what portion of that frame is displayed in the Viewer. When disabled, scopes only display data for the current area requested by the Viewer rather than the full frame.

Histogram

The content menu Windows > New Scope > Histogram provides three color channel and luma channel information that describes the distribution of red, green, blue, and luma pixels throughout the current frame.

The Histogram graphs the number of pixels at each brightness level, and from left to right, the areas of the Histogram represent shadow, mid tones, and highlights.
There are a number of view controls on the **Histogram** tab:

- **Viewer selection** - if you have multiple Viewers open, use the dropdown menu to associate Histogram output to the required Viewer.

  The default value, **ActiveViewer**, automatically displays details on the last Viewer you selected.

- **Channel selection** - select the channels to output. The default setting displays RGB, but you can also view channels separately.

- **Mode selection** - select the mode to output. The default setting displays ganged RGB, but you can also view the channels separately by checking **Parade**.

- **Current View** - describes the view currently displayed in the scope, whether it's the A or B buffer and the view. The view defaults to **main**, unless **main** has been replaced in multi-view scripts or projects.

Depending on which Viewer tools and views you have active, you can have up to four scopes displayed at once.

For example, with two stereo Read nodes, one in each input buffer, and **wipe** and **Anaglyph** active, the scopes display something like this:

The scopes feature global customizable guides to help you view data. Navigate to **Preferences > Panels > Scopes** and enter values between 0 and 1 for **Blackpoint** and **Whitepoint**.

The guides at the edges of the Histogram turn red to warn you when the distribution is out of range:
Using the Histogram Node

The properties panel for the Histogram node includes a window that graphs the number of pixels at each brightness level. This is a useful gauge to see whether an image has a good distribution of shadows, midtones, and highlights.

To Define Tonal Range with the Histogram Node

1. Click Color > Histogram to insert a Histogram node at the appropriate place in your script.
2. Connect a Viewer to the output of the Histogram node so you can see the effect of your changes.
3. Drag the leftmost input range slider till it roughly lines up with the initial boundary of the histogram.
4. Drag the rightmost input range slider till it roughly lines up with the final boundary of the histogram.
5. Drag the middle input range slider to define the midtone, or neutral, value.
Waveform

The content menu Windows > New Scope > Waveform scope provides information on luminance, or brightness, which you can use to decide whether the footage is over or under exposed. The information represents luminance values from 0 - 100% (black through the spectrum to white). The higher the waveform, the brighter the image in the Viewer.

Tip: You can pan the view area by holding Alt, or the middle mouse button, and dragging in the panel.

The upper white marker is used to measure when over exposure could be a problem. If your waveform has a lot of traces over the white marker, you could consider reducing the brightness of the image. The opposite is true of the lower black marker.

There are also Viewer and Channel selection controls on the Waveform tab:
• Viewer selection - if you have multiple Viewers open, use the dropdown menu to associate Waveform output to the required Viewer.

The default value, Active Viewer, automatically displays details on the last Viewer you selected.
• Channel selection - select the channels to output. The default setting displays RGB, but you can also view channels separately.
• Mode selection - select the mode to output. The default setting displays ganged RGB, but you can also view the channels separately by checking Parade.
• Current View - describes the view currently displayed in the scope, whether it's the A or B buffer and the view. The view defaults to main, unless main has been replaced in multi-view scripts or projects.
Depending on which Viewer tools and views you have active, you can have up to four scopes displayed at once.

For example, with two stereo Read nodes, one in each input buffer, and `wipe` and `Anaglyph` active, the scopes display something like this:

![Scope Display Example](image)

The scopes feature global customizable guides to help you view your data. Navigate to `Preferences > Panels > Scopes` and enter values between 0 and 1 for `Blackpoint` and `Whitewpoint`.

The guides at the top and bottom of the Waveform turn red to warn you when the distribution is out of range:

![Scope Display Example](image)

**Vectorscope**

The content menu `Windows > New Scope > Vector` display color, saturation, and hue information for the current frame. Similar to color wheels, vector scopes display information radially, from the center outward. The farther from the center the data spans, the more saturation is represented.

In the image on the left, you can see that the frame represented contains mostly yellows and reds, but the values are not oversaturated. The image on the right represents a badly saturated frame.
Notice the spill of red traces distributed toward the edge of the scope pass the target (the highlighted square).

![Normal saturation.](image1) ![High Saturation.](image2)

**Tip:** You can pan the view area by holding Alt, or the middle mouse button, and dragging in the panel.

There is also a **Viewer** selection control on the **Vectorscope** tab:

- **Viewerset** - if you have multiple Viewers open, use the dropdown menu to associate vector scope output to the required Viewer.

  The default value, **ActiveViewer**, automatically displays details on the last Viewer you selected.

- **Current View** - describes the view currently displayed in the scope, whether it’s the A or B buffer and the view. The view defaults to **main**, unless **main** has been replaced in multi-view scripts or projects.

  Depending on which Viewer tools and views you have active, you can have up to four scopes displayed at once.

  For example, with two stereo Read nodes, one in each input buffer, and **wipe** and **Side by Side** active, the scopes display something like this:
Using the Pixel Analyzer

Nuke's Pixel Analyzer enables you to analyze single and multiple pixels, or the entire image, and compare color values between Viewers. The analyzer stores current, minimum and maximum, average, and median values which can then be copied by value to controls on other nodes. For example, you might use the minimum and maximum values from an analysis to set the black and white points on another image. The analyzer is accessed from the contents menu **Windows** sub-menu.

The Pixel Analyzer has two modes:

- **pixel selection** - the default mode, allows you to make single or multiple pixel selections in the Viewer for analysis. You can also analyze an area of the Viewer using a Region of Interest tool.

- **full frame** - analyzes the contents of the current frame, regardless of any selections you've made in the Viewer.

By default, **full frame** samples the visible region of the Viewer. As a result, actions that change the visible area, such as zooming in and out, alter the available color values.

**Note:** The **Preferences > Panels > Scopes > Include Viewer color transforms** control does not affect the Pixel Analyzer.
Analyzing Pixel Selections

The analyzer’s **pixel selection** mode allows you to select pixels in the Viewer, individually or as a group, and display the color information in swatches. To make selections in the Viewer:

1. Connect a Viewer to the output you intend to analyze. You can connect multiple Viewers to a single output, and vice versa, for comparison.
2. Click the contents menu and select **Windows > Pixel Analyzer**. The **Pixel Analyzer** panel displays.
3. Use the **sample** dropdowns to control which Viewer and layer provides the swatch information:
   - **viewer** - the default, **current viewer**, always samples the active Viewer if there is more than one. Selecting pixels in a Viewer or clicking on a Viewer’s tab causes the Viewer to become the active or current Viewer. Alternatively, you can select a named Viewer from the dropdown when more than one is open.
   - **layer** - the default, **current layer**, displays the layer(s) specified in the Viewer **channels** dropdown.

   **Note:** If **rgba** channels are not present, the first four available channel values are written into the Pixel Analyzer’s **rgba** controls.

   **Note:** The color swatches may not update immediately if you select a layer from the **sample** dropdown that is not currently visible in the Viewer, because the new layer must be rendered in the background before the swatch is calculated.

Alternatively, you can select a named layer from the dropdown. When switching between Viewers, the Pixel Analyzer attempts to match the selected layer in the new Viewer. If that layer doesn’t exist, the default **current layer** is displayed.

   **Tip:** You can use the **layer** dropdown to sample a layer that is not displayed in the Viewer, allowing you to compare position and color values between layers.

4. Ensure that the mode dropdown is set to **pixel selection**.
5. If you want to make multiple selections, without losing your previous selections, enable **accumulate**.
6. There are three methods of selecting pixels for analysis:
   - **single** - hold **Ctrl/Cmd** and click in the Viewer to select a pixel to add it to the swatches.
• **multiple** - hold **Ctrl/Cmd** and click-and-drag in the Viewer to select multiple pixels as you move the pointer.

• **region** - hold **Ctrl/Cmd+Shift** and click-and-drag in the Viewer to create a Region of Interest. Adjust or move the ROI using the Viewer handles.

**Tip:** To deselect all pixels, hold **Ctrl/Cmd** and right-click in the Viewer.

With multiple selections in the Viewer, the color swatches provide the following information:

• **current** - the color value of the last pixel selected.

• **min/max** - the darkest and brightest color values from the selection.

• **average** - the mean average color value from the selection.

• **median** - the mid-point color between the darkest and brightest colors in the selection.

You can enable **live median update** to calculate the **median** swatch dynamically, rather than at pen-up.

**Note:** In **single** mode, all swatches show identical information and in **region** mode, the **current** swatch is disabled.

7. Click on the swatches to display **rgba** and **hsvl** color values for the selected swatch.

The Pixel Analyzer also detects **inf** (infinite) and **nan** (not a number) color values. The ! point in the swatch indicates inf/nan values.

8. You can reset the swatches at any time by clicking **clear selection**, except when **full frame** mode is enabled, in which case the button is grayed out.

9. Use the **range** dropdown to select the color bit-depth you intend to display. For example, selecting **8-bit** limits your color values to 255.

10. Set the **min/max** channel you want to use to calculate the min/max color swatches.
This control defaults to luminance (l), but if you wanted to display the minimum values in only the red channel, for example, you could set min/max to red (r).

Analyzing Full Frames

The analyzer’s full frame mode samples the visible region of the Viewer by default. As a result, actions that change the visible area, such as zooming in and out, alter the available color values.

1. Connect a Viewer to the output you intend to analyze. You can connect multiple Viewers to a single output, and vice-versa, for comparison.
2. Click the contents menu and select Pixel Analyzer.
   The Pixel Analyzer panel displays.
3. Use the sample dropdowns to control which Viewer and layer provides the swatch information:
   • viewer - the default, current viewer, always samples the active Viewer if there is more than one. Selecting pixels in a Viewer or clicking on a Viewer’s tab causes the Viewer to become the active or current Viewer. Alternatively, you can select a named Viewer from the dropdown when more than one is open.
   • layer - the default, current layer, displays the layer(s) specified in the Viewer channels dropdown.

   **Note:** Enabling full frame processing forces the Pixel Analyzer to analyze the entire frame, regardless of the portion of the Viewer currently visible.

   **Note:** If rgba channels are not present, the first four available channel values are written into the Pixel Analyzer’s rgba controls.

   **Note:** The color swatches may not update immediately if you select a layer from the sample dropdown that is not currently visible in the Viewer, because the new layer must be rendered in the background before the swatch is calculated.

   Alternatively, you can select a named layer from the dropdown. When switching between Viewers, the Pixel Analyzer attempts to match the selected layer in the new Viewer. If that layer doesn’t exist, the default current layer is displayed.
Tip: You can use the layer dropdown to sample a layer that is not displayed in the Viewer, allowing you to compare position and color values between layers.

4. Ensure that the mode dropdown is set to full frame.
5. Click on the swatches to display rgba and hsvl color values for the selected swatch.
   The Pixel Analyzer also detects inf (infinite) and nan (not a number) color values. The ! point in the swatch indicates inf/nan values.

6. Use the range dropdown to select the color bit-depth you intend to display. For example, selecting 8-bit limits your color values to 255.
7. Set the min/max channel you want to use to calculate the min/max color swatches.
   This control defaults to luminance (l), but if you wanted to display the minimum values in only the red channel, for example, you could set min/max to red (r).

Applying Analysis Data

You can use color information stored in the Pixel Analyzer swatches in any other control that includes a swatch. For example, min/max color swatches analyzed in another shot can be applied to the current shot by dragging the swatch to the Grade nodes's blackpoint and whitepoint controls.
Making Tonal Adjustments

Defining tonal range (the black point, white point, and neutral value) is typically the first step in color correcting a clip. Tonal range adjustments often improve contrast, but more importantly, they set the stage for subsequent color corrections by properly dividing the colorspace into shadow, midtone, and highlight regions.

Before tonal adjustment. After tonal adjustment.

Several of Nuke’s color correction effects offer tonal adjustment tools. Of these, Grade and Histogram are probably the most intuitive to operate.

Sampling White and Black Points

The Grade node lets you define white and black points by sampling pixels from a Viewer frame.

To Define Tonal Range with the Grade Node

1. Click Color > Grade to insert a Grade node at the appropriate place in your script.
2. Connect a Viewer to the output of the Grade node so you can see the effect of your changes.
3. In the Grade properties panel, use the channels dropdown menu to select the channels you wish to process.
4. Click the blackpoint parameter’s color swatch.
   The eye dropper icon appears.
5. In the Viewer, press Ctrl/Cmd+Shift while clicking on the pixel you want to define as the black point (typically the darkest pixel).
6. Click the whitepoint parameter’s color swatch. The eye dropper icon appears.
7. In the Viewer, press Ctrl/Cmd+Shift while clicking on the pixel you want to define as the white point (typically the lightest pixel).
Making Basic Corrections

Adjustments to contrast, gamma, gain, and offset often comprise the bulk of the work in color correction. Some artists prefer to make these adjustments via sliders; others prefer curves (which represent the range of color values in an image.) Nuke's ColorCorrect and ColorLookup nodes offer tools to suit either preference.

See Using ColorCorrect Sliders or Using ColorLookup Curves
Using ColorCorrect Sliders

The ColorCorrect node is particularly convenient for making quick adjustments to contrast, gamma, gain, and offset. A single window houses sliders for all these basic corrections and allows you to apply these to a clip's master (entire tonal range), shadows, midtones, or highlights.

To Adjust Contrast, Gain, Gamma or Offset with the ColorCorrect Node

1. Click **Color > ColorCorrect** (or press **C**) to insert a ColorCorrect node at the appropriate place in your script.
2. Connect a Viewer to the output of the ColorCorrect node so you can see the effect of your changes.
3. In the ColorCorrect properties panel, use the **channels** dropdown menu to select the channels you wish to process.
4. Drag the slider appropriate to the region you want to affect an operation you want to apply. For example, to brighten the images highlights, you would drag on the **highlights gain** slider.

Remember too that you can use the color sliders to apply any of the corrections on a per channel basis.

5. On the **Ranges** tab, enable **test** to show what is considered to be in the shadows, midtones, or highlights.

This overlays the output with black (for shadows), gray (for midtones), or white (for highlights). Green and magenta indicate a mixture of ranges.
6. Still on the **Ranges** tab, you can use the **shadow** and **highlight** lookup curves to edit the range of the image that is considered to be in the shadows or highlights. You can also look up color information for the current pixel in the Viewer.

To return a curve to its default values, select it and click **reset**.

⚠️ **Warning:** Do not adjust the **midtone** curve. Midtones are always equal to $1$ minus the other two curves.

7. To control how much of the original luminance is preserved after the color correction, enable and adjust **mix luminance**. A value of 0 means the altered luminance is used in the output image. A value of 1 produces a luminance value close to that of the original input image.

**Note:** When **mix luminance** is set to 1, the resulting luminance value is close to the original luminance, but not exactly the same. The difference may vary depending on the color corrections applied to the source image.
Adjusting Black Levels with the Toe Node

Toe lifts the black level, in a similar way to gain controls, but with a roll-off so that whites are mostly not affected.

1. Click Color > Toe to create a Toe node. Connect it to the image whose black levels need adjusting.
2. Adjust the lift slider to change the black values to the specified gray value, without affecting any original white values of the image.
3. If necessary, you can limit the effect to a particular channel with the channels controls.
4. If you need to, you can pick a channel in the (un)premultby dropdown to divide the image first with that channel and then multiply it again afterward. Doing this can sometimes improve your color correction results on anti-aliased edges.
5. You can also use the mix control to dissolve between the original input (value 0) and the full effect of the Toe node (value 1). If you only want to use one channel for mixing, you can specify that using the mask control.

Using ColorLookup Curves

If you prefer to work with color curves, you can use the ColorLookup node to make contrast, gamma, gain, and offset adjustments (and, in fact, many others). Color curves refer to line graphs of a given color channel's brightness. The horizontal axis represents the channel's original, or input, values, and the vertical axis represents the channel's new, or output values.
As Figure shows, you can edit the ColorLookup node’s color curves to make all of the types of corrections that are possible through the ColorCorrect node - and you can generally make these corrections with more flexibility and precision than is possible with sliders.

**To Make Basic Corrections with the ColorLookup Node**

1. Click **Color > ColorLookup** to insert a ColorLookup node at the appropriate place in your script.
2. Connect a Viewer to the output of the ColorLookup node so you can see the effect of your changes.
3. In the ColorLookup properties panel, click **red, green, blue**, or **alpha** if you want to limit the subsequent operations to a particular channel.
   
   You can select multiple curves in order to edit one curve with reference to another. Otherwise, select the **master** curve (which represents all channels).
4. To speed up the color calculation, the ColorLookup node uses a precomputed lookup table between 0 and the value specified in the range field. You can adjust the range value, or uncheck the use precomputed table box if necessary to tell ColorLookup to not use a precomputed table.

5. In the Viewer, drag the cursor over the pixels you want to sample for the correction. In the ColorLookup properties panel, press Ctrl+Alt (Cmd+Alt on a Mac) while clicking on the curve to set points at the places where the red, green, and blue lines intersect with the color curve.

6. Edit the position of the points and adjust the tangent handles to adjust the curve shape for the color correction.

As an alternative to steps 4 and 5, you can use the source control to pick a source color for adding points. Then, use target to pick a destination color. Finally, do one of the following:

• Click Set RGB to add points on the red, green, and blue curves, mapping source to target.
• Click Set RGBA to add points on the red, green, blue, and alpha curves, mapping source to target.
• Click Set A to add points on the alpha curve, mapping source to target.

You can use these controls to match shadow, midtone, and highlights on two plates, for example. Set source to shadow rgb in one, target to shadow rgb in the other, then press Set RGB. Same for midtone and highlight areas.
Making Hue, Saturation, and Value Adjustments

For certain color correction tasks like spill suppression, you ideally want to influence only a very narrow range of color values. For such tasks, it’s often helpful to use effects that employ the Hue, Saturation, and Value (HSV) color model. As its name indicates, the HSV color model breaks color into three components:

- **Hue**, which refers to the color’s location on the traditional color wheel.
- **Saturation**, which refers to the extent to which the color has “soaked up” its hue.
- **Value**, which refers to the brightness of the color.

Nuke offers effects that allow you to correct the hue, saturation, and value components individually or collectively.
Correcting HSV

Nuke's HSVTool node lets you simultaneously adjust hue, saturation, and value components from a single properties panel. It also features a color replacement tool. The main strength of this node is the precision it offers in limiting corrections to a narrow swath of colors.

For example, suppose you wanted to add a bit more punch to the waterfront scene by diversifying the rooftop hues. To do so, you could limit the correction to the rooftop's ochre-colored hues by sampling a few pixels, then shift their values. Because you limited the color range, the surrounding image would be generally unaffected by the shift.

To make HSV corrections with the HSVTool node

1. Click Color > HSVTool to insert an HSVTool node at the appropriate place in your script.
2. Connect a Viewer to the output of the HSVTool node so you can see the effect of your changes.
3. Limit, as appropriate, the range of colors you want subsequent corrections to influence:
   
   - In the HSVTool properties panel, click the srccolor color swatch. Ctrl/Cmd+click on the Viewer to sample a single color displayed, or Ctrl/Cmd+Shift+drag to sample a range of colors. To sample a single color from the node's input while viewing its output, Ctrl/Cmd+Alt+click on the Viewer. To sample a region from the input, Ctrl/Cmd+Alt+Shift+drag on the Viewer.

   Tip: You can discard sampled pixels by Ctrl/Cmd+right-clicking in the Viewer.

   - The Range sliders on Hue, Saturation, and Brightness clamp to the sampled range.
   - For any color component, drag on the Range sliders to expand the color range as necessary.
   - For any color component, drag on the Range Rolloff slider to fine tune the color range. Doing so, adjusts the amount of falloff allowed past the limits defined by the Range sliders.
4. Make the necessary HSV corrections:
• For hue corrections, drag on the **Rotation** slider to input color wheel value between 0 and 360.
• For saturation corrections, drag on the **Saturation Adjustment** slider to input values between -1 (completely desaturated to some shade of gray) and 1 (completely saturated).
• For value corrections, drag on the **Brightness Adjustment** slider to input values between -1 (black) and 1 (white).

You can also make color replacements using the srccolor and dstcolor parameters: First sample the color you wish to replace with the srccolor color swath, then sample the color which you wish to use as the replacement with the dstcolor color swath. The color in dstcolor replaces the color in srccolor throughout the image.

Also, keep in mind that the HSVTool node makes an excellent keyer. You can use its Hue, Saturation, and Brightness range sliders to precisely select a range of colors, then use the channel output dropdown at the bottom of the dialog to output this selection as a matte channel. This dropdown lets you specify which color components (hue, saturation, value, etc.) are added to the matte.

**Correcting Hue Only**

Nuke’s HueCorrect node lets you make precision adjustments to the levels of saturation in a range of hues. You do so via edits to a series of suppression curves.

![Editing the suppression curve.](image)

By selecting which curve you edit and how much of that curve you alter, you can precisely limit the influence of the effect.
Suppressing spill

For the compositor, HueCorrect is obviously of greatest use in diminishing green, blue, or redscreen spill.

To suppress spill with the HueCorrect node:

1. Click Color > HueCorrect to insert a node at the appropriate place in your script.
2. Connect a Viewer to the output of the HueCorrect node so you can see the effect of your changes.
3. In the HueCorrect properties panel, select the channels you want to influence:
   - Click sat to influence all channels (red, green, blue, and alpha) equally.
   - Click lum to influence all channels, but with luminance weighting in effect (meaning that the red channel receives approximately 30% of the effect; the green, 60%; and the blue, 10%).
   - Click red to apply the curve as a lookup on the red channel only, looking up the pixel's hue on the curve and then multiplying the red value in the pixel by the lookup result.
   - Click green to apply the curve as a lookup on the green channel only, looking up the pixel's hue on the curve and then multiplying the green value in the pixel by the lookup result.
   - Click blue to apply the curve as a lookup on the blue channel only, looking up the pixel's hue on the curve and then multiplying the blue value in the pixel by the lookup result.
   - Click r_sup to apply a suppression function to reduce the level of the red channel. While the red curve is used to directly multiply the red channel by the curve value, the r_sup curve is used to control the amount that the red channel is suppressed.
   - Click g_sup to apply a suppression function to reduce the level of the green channel. While the green curve is used to directly multiply the green channel by the curve value, the g_sup curve is used to control the amount that the green channel is suppressed.
   - Click b_sup apply a suppression function to reduce the level of the blue channel. While the blue curve is used to directly multiply the blue channel by the curve value, the b_sup curve is used to control the amount that the blue channel is suppressed.

   Note that you can select multiple curves in order to edit one curve with reference to another.

4. If necessary, drag the cursor over the Viewer to sample the image pixels that are representative of the part of the image you want to correct. Then, in the HueCorrect properties panel, press Ctrl+Alt (Cmd+Alt on a Mac) while clicking on the curve to plot a particular pixel's value on the curve. This lets you see what portion of the curve you want to edit.
5. Edit the curve as necessary - typically this means dragging down on control points in the hue region that you wish to suppress.
6. To control how much of the original luminance is preserved after the color correction, enable and adjust **mix luminance**. A value of 0 means the altered luminance is used in the output image. A value of 1 produces a luminance value close to that of the original input image.

![Original image.](image)

Mix luminance set to 0.  Mix luminance set to 1.

**Note:** When **mix luminance** is set to 1, the resulting luminance value is close to the original luminance, but not exactly the same. The difference may vary depending on the color corrections applied to the source image.

---

### Correcting Saturation Only

For the times when you just want to correct the saturation component and don’t require limiting the correction to any particular channel, you can use Nuke’s Saturation node. Its controls are bare bones - basically, just a **saturation** slider.

**To make saturation corrections with the Saturation node**

1. Click **Color > Saturation** to insert a Saturation node at the appropriate place in your script.
2. Connect a Viewer to the output of the Saturation node so you can see the effect of your changes.
3. Drag the **saturation** slider to make the necessary corrections.
Masking Color Corrections

Virtually all the color-correction effects in Nuke include **mask** parameters that let you limit the correction to the non-black pixel values of a matte image. For example, suppose you want to add a blue cast to the following scene without affecting the buildings.

![Original image.][1]

![Color-corrected image.][2]

![Mask image.][3]

![Color-corrected image.][4]

You could create a garbage mask that covers the river, then boost the red channel's gamma in the area of the frame that underlies the mask.

Typically, mask controls are located toward the bottom of the properties panel. However, in the case of multi-purpose effects like HSVTool, there may be multiple mask controls, so that you can limit each type of correction with a different mask.

![Selecting a mask channel.][5]

**To Mask a Color Correction**

1. Open the node's properties panel and locate the **mask** controls.
2. Select the channel you wish to use as the mask from the dropdown menu.

3. If you check inject in the mask controls, the mask from the mask input is copied into the predefined mask.a channel. This way, you can use the last mask input again downstream. You can also set a stream of nodes to use mask.a as the mask, and then change the masking of all of them by simply connecting a new mask into the mask input connector of the first node.

4. If necessary, check the invert box to reverse the mask.

5. To blur the edges of the mask, check fringe.

6. If the overall effect of the node is too harsh, you can blend back in some of the input image by dragging on the mix slider.

7. If you want to output only the portion of the frame underlying the mask, check the (un)premult by box.
Applying Grain

*Grain matching* - ensuring that all of the elements in a composite, including those which were digitally generated, look like they were shot on the same film stock - is often one of the final steps in achieving a convincing integration of all of a composite's elements. Nuke offers effects for synthetically creating grain and for reading in practically-created grain (grain derived from actual film stock).

An example of applying grain to an image: Grainless image. An example of applying grain to an image: Grained image.

Using Synthetic Grain

Nuke offers several nodes for creating synthetic grain: Dither, Grain, ScannedGrain, and if you have a NukeX license, F_ReGrain. Of these, Dither is the crudest - it basically lets you specify the amount of noise per channel.

Grain includes presets for matching film stock and a means for controlling the mix between the generated grain and the backplate. ScannedGrain offers film stock presets, plus synthetic grain controls for applying practical grain.

F_ReGrain, for NukeX, is designed to sample an area of grain from one image and then to generate unlimited amounts of this grain with exactly the same statistics as the original. This new grain can then be applied to another image. See Using F_ReGrain for more information.

To add synthetic grain with the Grain node

1. Click **Draw > Grain** to insert a Grain node at the appropriate place in your script.
2. Connect a Viewer to the output of the Grain node so you can see the effect of your changes.
3. From the **presets** dropdown menu, select one of the film stock you want to match.
4. Adjust the **Size** sliders for the red, green, and blue channels to shrink or enlarge the granules.
5. Adjust the **Irregularity** sliders to increase or decrease the random quality of the grain, according to the different channels.
6. Adjust the **Intensity** sliders to increase or decrease the contrast of the grain against the original image.

### Using Practical Grain

Although Nuke’s ScannedGrain node offers controls for creating synthetic grain (ones comparable to those just discussed), it’s main use is for reading in and applying scanned grain - that is, grain derived from actual film stock. If your facility has such sequences available, you can read them in and apply them using the ScannedGrain node. Creating grain files is described below, as well as using the resulting grain files with the ScannedGrain node.

### To Create Film Stock Sequences

1. Film a gray card. Only about 50 frames are needed.
2. Scan the film in.
3. Select **Image > Read** to load the scanned image into Nuke.
4. Add a Blur node (**Filter > Blur**) after the image to blur the image until you cannot see any grain. Then, blur the image a bit more.
5. Select **Merge > Merge** to insert a Merge node in your script. Connect the A input of the Merge node into the original image, and the B input into the Blur node. Then, open the Merge controls and select **minus** from the **operation** dropdown menu. The blurred image is subtracted from the original image.

   The purpose of this and the previous step is to subtract any overall gray level from the grain so that only the grain is left.
6. Select Color > Math > Add to insert an Add node after the minus node. In the Add node controls, enter 0.5 in the value field. This adds a value of 0.5 to all channels.

This step is necessary, because the ScannedGrain node subtracts 0.5 from the channels when it reads the grain file (the subtraction is needed to store negative numbers in most file formats).

7. Select Image > Write to insert a Write node after the Add node. Render the output. Any file format will do (for example, we have used the .rgb extension in the grain files on our website).

---

To Add Scanned Grain with the ScannedGrain Node

1. Click Draw > ScannedGrain to insert a ScannedGrain node at the appropriate place in your script.
2. Connect a Viewer to the output of the ScannedGrain node so you can see the effect of your changes.
3. Click the folder icon of the grain field and navigate to the appropriate film stock sequence. Select Open.
4. If necessary, check the resize box to scale the grain sequence up or down to match your working resolution.
5. In the min. width field, define a minimum width (in pixels) that images have to have in order to receive grain.
6. Enter values into the red, green, and blue amount fields to increase or decrease on a per-channel basis the density of granules. (This is accomplished, crudely speaking, by boosting or reducing the gain of the grain sequence.)

Now you’re ready to fine-tune the blending between the grain and backplate.
To Mix the Grain and Backplate

1. Drag on the **saturation** slider to increase or decrease the intensity of the grain’s hue across all channels.

2. If necessary, you can also use the supplied curve editor to edit the grain sequence's color curves. In this manner, you can alter gain, gamma, contrast, etc. on a per channel basis. (These curves function in the same manner as those described in Using ColorLookup Curves).

3. To set a low threshold, based on the input image, below which the grain is not subtracted, adjust the **low_clip** slider.
Applying Mathematical Operations

Nuke’s Color icon in the Toolbar houses a number of nodes which are designed to apply common mathematical operations to channels. These operations include clamps, offsets, inversions, multiplications, and expressions.

Clamping Channel Values

To clamp a channel’s values is to ensure that its blackest blacks and whitest whites are visible on an intended display device. Nuke’s Clamp node lets you assign “legal” values to colors that are either too light or dark for the intended display device.

For this effect, you use Nuke’s Clamp node.
1. Click Color > Clamp to insert a Clamp node at the appropriate point in your script.
2. Connect a Viewer to the output of the Clamp node so you can see the effect of your changes.
3. In the Clamp properties panel, use the channels field to select the channel you wish to clamp.
4. Drag the minimum slider to the legal value. (This has the effect of causing black values to go gray.)
5. Drag the maximum slider to the legal value. (This has the effect of causing white values to go gray.)
Offsetting Channel Values

To *offset* a channel's values is to add a fixed value to them, which, in effect lightens the whole channel. You can also add a negative value to a channel, in which case the channel gets darker.

For this effect, you use Nuke's Add node.

1. Click **Color > Math > Add** to insert a Add node at the appropriate point in your script.
2. Connect a Viewer to the output of the Add node so you can see the effect of your changes.
3. In the Add properties panel, use the **channels** field to select the channel you wish to offset.
4. Use the **value** slider to input the value you wish to add to the channel's values.
5. If you are using premultiplied input images, you may want to check **(un)premult by** and select **rgba.alpha** from the dropdown menu. This simulates doing the addition before the premultiplication was done.

Inverting Channel Values

To *invert* a channel is to subtract its values from one, which causes its blacks to become white and its whites to become black. In the course of building a script, you'll have frequent need to invert mattes in particular.

To invert channels you use Nuke's Invert node.

1. Click **Color > Invert** to insert an Invert node at the appropriate point in your script.
2. Connect a Viewer to the output of the Invert node so you can see the effect of your changes.
3. In the Invert properties panel, use the **channels** field to select the channel you wish to invert.

**Multiplying Channel Values**

To *multiply* a channel's values is to times them by a given factor, which has the effect of lightening the channel while preserving the black point. (This operation is also known as gain.)

For this effect, you use Nuke's Multiply node.

1. Click **Color > Math > Multiply** to insert a Multiply node at the appropriate point in your script.
2. Connect a Viewer to the output of the Multiply node so you can see the effect of your changes.
3. In the Multiply properties panel, use the **channels** field to select the channel whose values you wish to multiply.
4. Use the **value** slider to input the factor by which you want to times the channel's values.

**Applying Expressions to Channel Values**

Up till now, the discussion has focused on how to apply simple mathematical formulae - additions, subtractions, multiplications, etc. - to a channel's values. Nuke's Expression node, however allows you to apply complex formulae to a channel's values. The actual syntax for expressions is rather complex, and thus must be deferred to **Expressions**. For now, you can read about the basics of how to operate the Expression node.

1. Click **Color > Math > Expression** to insert an Expression node at the appropriate point in your script.
2. Connect a Viewer to the output of the Expression node so you can see the effect of your changes.
3. In the Expression properties panel, use the channel dropdown menus and buttons to select the channel to which you wish to apply an expression.
4. Type the actual expression in the **=** field next to the channel.
   
   For example, to assign noise to the red channel, then boost the gain of that result by 20 you would type **(random*r)*20**.
5. If necessary, you can apply different expressions to different sets of channels by repeating the above steps for the other channel dropdown menus and buttons.

6. If you need to use a long expression in several fields, you can use the fields on top of the properties panel for assigning the expression temporarily to a variable. Enter your variable on the left side of the = sign, and the expression on the right. You can then use the variable to represent the expression in the = fields next to the channels.

![A checkerboard modified using an Expression node.](image)
Transforming the Color Space

Whenever you read a clip into a script, it is automatically converted to Nuke’s native color space, which is 32-bit per channel RGB, a linear format. This conversion takes place even if the clip you read in is in the Kodak Cineon format, which is a logarithmic format.

The reverse of this conversion, called a lin-to-log conversion, also automatically takes place when you write the processed element back out of the script - that is, Nuke automatically converts it back into a Cineon element.
Overriding the Default Cineon Conversion

Nuke uses the Kodak-recommended settings when making Cineon conversions in either direction. It’s rare that you would want to override these settings, but if it becomes necessary you can use Nuke’s Log2Lin or PLogLin nodes.

To Override the Default Cineon Conversions Using Log2Lin

1. Double-click on the Read node of the Cineon element whose conversion you wish to override.
2. In the Read properties panel, set the colorspace dropdown menu to linear. This halts the automatic log-to-lin conversion.
3. Click Color > Log2Lin to insert a log2lin node directly after the Read node.
4. In the Log2Lin properties panel, set the operation dropdown menu to log2lin.
5. Set black, white, and gamma to the appropriate values.
6. Copy the Log2Lin node and insert it just before the element’s Write node.
7. Open up the properties panel of the second Log2Lin node and set the operation dropdown menu to lin2log. This gives you the reverse of the conversion you created above.
8. Double click on the element’s Write node.
9. In the Write properties panel, set the colorspace dropdown menu to linear. This halts the automatic lin-to-log conversion and lets the one you create above have priority.

To Convert Between Logarithmic and Linear Color Spaces

You can also convert between logarithmic and linear color spaces with the PLogLin node. This alternative method is better known as the Josh Pines log conversion, and it’s based on using a single gray value, rather than a black and a white one, like in the Log2Lin node. To use the PLogLin node:

1. Much like with the Log2Lin node, when you’re using the PLogLin node you need to first make sure your Read or Write nodes aren’t automatically converting your color space. To do this, click on your Read or Write node, and select linear in the colorspace dropdown.
2. Create a PLogLin node by clicking Color > PLogLin. Connect it to either your Read node’s output or the Write node’s input, depending on which one you’re converting the color space for.
3. In the operation dropdown, select the operation you want PLogLin to perform. Select log to lin to convert from logarithmic to linear, and lin to log to do the reverse.

4. Adjust the linear reference value slider to the linear value that corresponds with the logarithmic reference value and the log reference value slider to the value that corresponds with the linear reference value.

5. You can also adjust the film response gamma value in the negative gamma field, and use the density per code value field to tell PLogLin what type of change occurs in the negative gamma for each log space code value when converting.

6. If you need to, you can pick a channel in the (un)premult by dropdown to divide the image first with that channel and then multiply it again afterward. Doing this can sometimes improve your color correction results on anti-aliased edges.

7. You can also use the mix control to dissolve between the original input (value 0) and the full effect of the PLogLin node (value 1). If you only want to use one channel for mixing, you can specify that using the mask control.

To Create a 3D LUT in Log Color Space (for example Cineon)

For good visual fidelity when using 3D LUTs, it is recommended that a log color space is used for 3D LUT generation. To create a 3D LUT in a log colorspace (Cineon in this example):

1. Create a CMSTestPattern node at a required density.

2. Connect this to a Colorspace node to convert in: Cineon to out: Linear. This is a reverse log conversion, and gives a higher density of samples in the shadows of your grading curve, with reduced density in the highlights.

3. Connect the Colorspace node to your grading nodes.

4. Connect the grading nodes to another Colorspace node to convert in: Linear to out: Cineon. This is the log conversion, and converts the grade samples back to the normalized range (0-1) for 3D LUTs.

   It is important to distribute the samples appropriately due to the low resolution of 3D LUTs.

5. Connect a GenerateLUT node to the last Colorspace node.

Tip: To reproduce the grade, set a Vectorfield node to Cineon for both the colorspace in and colorspace out controls.
Making Other Colorspace Conversions

You can also convert elements from Nuke’s native color space to other color spaces more appropriate to a given process or intended display device. For conversions such as this, use Nuke’s Colorspace node, which supports RGB, HSV, YUV, CIE, and CMS formats (and various sub-formats).

To Convert an Element in Nuke’s Native Color Space into Another Color Space

1. Click **Color > Colorspace** to insert a Colorspace node into the appropriate place in your script.
2. In the Colorspace properties panel, set the rightmost dropdown menu in the **out** controls to the appropriate standard.
3. Set the dropdown menu in the middle of the **out** controls to the appropriate standard.
4. Set the leftmost dropdown menu in the **out** controls to the color space of your choice.
5. If you wish to reverse this conversion later in the script:
   * Copy the Colorspace node and insert it at the appropriate point in your script.
   * Set the **out** controls to **sRGB**, **D55**, and **RGB**.
   * Set the **in** controls to match the values you entered in steps 2, 3, and 4 above.
6. If you wish write out the element in the new color space:
   * Double-click on the element’s Write node.
   * In the Write properties panel, set the **colorspace** dropdown menu to **linear**. This halts the automatic conversion and lets the one you create above have priority.

Converting Color Spaces with the OCIOColorSpace Node

Much like the ColorSpace node, you can use the OCIOColorSpace node for converting an image sequence from one colorspace to another. Conversions with the OCIOColorSpace node are based on the OpenColorIO library (for more information, see [http://opencolorio.org](http://opencolorio.org)). Using OCIOColorSpace is pretty simple:

1. Select **Color > OCIO > OCIO ColorSpace** and connect it to your image sequence.
2. Select the channel or layer you want the conversion to affect using the **channels** controls.
3. In the **in** dropdown, select the colorspace of your input image.
4. In the **out** dropdown, select the colorspace you want to convert the image to.
For more information on the OCIOColorSpace node, and other nodes based on the OpenColorIO library, see Color Nodes in the Nuke Online Help for more information.

**Note:** Nuke is bundled with a pre-compiled version of the OpenColorIO library and a suite of OCIO nodes. If you already have a pre-built version of the OCIO library on your system and your environment is not set up correctly for Nuke, you may encounter problems. For OCIO to work correctly, Nuke requires the compiled versions of PyOpenColorIO and libOpenColorIO to match. To override Nuke's pre-packaged version with your own custom version, follow the steps below:

- **Linux:**
  Manually replace the Nuke packaged versions of `libOpenColorIO.so` and `PyOpenColorIO.so` with your custom versions of these files. These can be found in the `<NukeInstall>` and `<NukeInstall>/plugins` directories respectively.

- **Mac:**
  Set your `NUKE_PATH` to the location of `PyOpenColorIO.so` and your `DYLD_LIBRARY_PATH` to the location of `OpenColorIO.so`. For example:
  ```
  export NUKE_PATH="/myOCIOLibraryLocation/
  export DYLD_LIBRARY_PATH="/myOCIOLibraryLocation/
  ```
  NOTE: Due to an Apple security update in Mac OS X 10.7 (Lion), DYLD variables can't be loaded from the environment.plist. See [http://support.apple.com/kb/TS4267](http://support.apple.com/kb/TS4267) for more information.

- **Windows:**
  Both `OpenColorIO.dll` and `PyOpenColorIO.pyd` must be in the same directory. You then need to set your `NUKE_PATH` to this directory. For example:
  ```
  set NUKE_PATH="\myOCIOLibraryLocation\"
  ```

- **All Platforms:**
  In addition to the steps above, you need to set the OCIO environment variable to point to your OCIO configuration file. This overrides the configuration file specified in Nuke's preferences (see Setting Preferences). For example:
  ```
  export OCIO="/myOCIOModuleLocation/config.ocio"
  ```
  The OCIO nodes in Nuke are compiled against a specific version of the OCIO libraries (for the current version, see [Third-Party Libraries and Fonts](#)). If you're using your own custom libraries, recompile the OCIO nodes against your versions of the libraries. Failure to follow these steps may result in errors when adding OCIO nodes or modifying OCIO options in the preferences.
Changing the Viewer Color Space

By default, a script's Viewers display images in Nuke's native color space. You can, however, set a script's Viewers to display images in non-native color spaces. Changing the display color space in no way affects your rendered output. You are applying a display-only lookup table.

Select the desired color space from the viewer process dropdown menu.

Adding Context Viewer Processes

You can add variables to register certain viewer processes in OCIODisplay nodes by creating a custom config.ocio file and then specifying the variable in the to_reference file transform.

In this example, the variable is called SHOT. If you intend to use the same name, make sure that SHOT is not assigned as an environment variable.

To edit the oio.config file:

1. Open your oio.config file in a text editor and review the colorspace entries. The example shows the sRGB entry.

   ```xml
   <ColorSpace
      name: sRGB
      family: "/"
      equalitygroup: ""
      bitdepth: 32f
      description: |
      Standard RGB Display Space
      isdata: false
      allocation: uniform
      allocationvars: [-0.125, 1.125]
   ```

```xml
```
2. Edit the `to_reference` line to include the variable you intend to create.
   `to_reference: <!FileTransform> {src: "$SHOT.spild", interpolation: linear}`
3. Save the file.

To register the viewer process:
1. Launch Nuke.
2. Press `S` over the Node Graph to open the `Project Settings`, or navigate to `Edit > Project Settings`.
3. Switch to the `Color` tab and use the `color management` dropdown to select `OCIO`.
4. Set `OCIO config` to `custom` and enter the file path to your `ocio.config` file.
5. Click the viewer process dropdown and select the process you want to register, in this case `sRGB (default)`.
   An error message displays in the Viewer because Nuke can't find the specified reference.
6. Click the viewer process dropdown and select `show panel` to open the OCIODisplay node properties.
7. Click the `Context` tab and enter the `key1` and `value1` pair as SHOT/sRGB.

The LUT specified by the variable is applied to the Viewer.

**Note:** Viewer processes are display-only LUTs and do not affect rendered output.
Filtering and Spatial Effects

This chapter explains how to create custom filter effects using the Convolve node (see Applying Convolves) and how to simulate depth-of-field (DOF) blurring using the ZDefocus node (see Simulating Depth-of-Field Blurring).

Applying Convolves

The Convolve node lets you create custom filter effects by supplying your own filter image. This image is used as the convolution matrix. In other words, the new value of a pixel is calculated by centering the filter image on the pixel, examining its neighbors, multiplying each pixel value by the corresponding pixel values in the filter image, and then adding the results together. This allows you to defocus a clip and create lens blur effects (bokeh) in the shape of the filter image (also see Simulating Depth-of-Field Blurring).

Using the Convolve Node

To use the Convolve node:
1. Click **Filter > Convolve** to insert a Convolve node after the image you want to receive the convolution filter effect (the **image** input).

2. Attach a Viewer to the output of the Convolve node.
   
   An error appears in the Viewer because we haven't connected the **filter** input yet.

3. Connect a filter image to the **filter** input. This image represents the shape and size of the camera aperture used to shoot the **image** input. It can be any shape you like, for example, a pentagon or a hexagon.
   
   If you don't have a filter image, you can create one using the Roto node (**Draw > Roto**) or the Flare node (**Draw > Flare**).

   ![A simple Convolve setup.](image)

   Note that you don't necessarily need to crop the filter image to a smaller size, as fast Fourier transforms are used to speed up convolutions with large filter images.

4. In the Convolve node controls, set **channels** to the channels of the **Source** image that you want to affect. By default, the convolve effect is applied to all channels.

5. To select what channel to use from the **filter** input, do one of the following:
   
   - Set the **filter channel** dropdown menu to the channel you want to use. By default, this menu is set to **rgba.alpha**.
   
   - To use the same channels from the **filter** input as the **image** input (that is, whatever **channels** is set to), check **use input channels**.

6. In most cases, you can leave **normalize** checked. This means the **filter** input is divided by the sum of all the pixels in it before using it, and ensures that the convolution doesn't change the overall brightness of the image.

7. Use the **filter bounds** dropdown menu in the Convolve controls to select whether you want to limit the filter image to:
   
   - **shape** - the **filter** input's bounding box. In this case, Convolve only uses the bounding box area, and the center of the filter is the center of the bounding box. This is the default value. You may want to use it, for example, if your **filter** input is a roto shape with a small bounding box that doesn't fill the entire format area.
• **format** - the `filter` input’s format. In this case, Convolve uses the entire format area, allowing you to offset the filter image within the format.

8. If you want to mask the convolve effect, check `mask` and select a mask channel using the controls on the right. For example, you can select a depth channel as a mask to simulate depth-of-field blurring.

![A simple Ramp node used as the mask.](image)

![The result.](image)

Note that you can also use the ZDefocus node for more accurate depth-of-field blurring. For more information, see Simulating Depth-of-Field Blurring.

9. To dissolve between the original `image` input and the full convolve effect, adjust the `mix` slider.

**Tip:** You can check **Use GPU if available** to have the node run on the graphics processing unit (GPU) rather than the central processing unit (CPU).

For more information on the minimum requirements, please see Windows, Mac OS X and macOS, or Linux or refer to the Nuke Release Notes available in Help > Release Notes.

You can select the GPU to use in the Preferences. Press `Shift+S` to open the Preferences dialog, make sure you're viewing the Preferences > Performance > Hardware tab, and set **default blink device** to the device you want to use. You must restart Nuke for the change to take effect.

If you are using a render license of Nuke, you need to add **--gpu** on the command line.
Simulating Depth-of-Field Blurring

The ZDefocus node blurs the image according to a depth map channel. This allows you to simulate depth-of-field (DOF) blurring.

The original image.

Using ZDefocus to simulate a more narrow depth of field. In this case, areas under the green sign remain in focus, whereas areas in front of and behind it are blurred. Any defocused highlights in the image bloom in the shape of the filter image, creating a bokeh effect.
In order to defocus the image, ZDefocus splits the image up into layers, each of which is assigned the same depth value everywhere and processed with a single blur size. After ZDefocus has processed all the layers, it blends them together from the back to the front of the image, with each new layer going over the top of the previous ones. This allows it to preserve the ordering of objects in the image.

Quick Start

Here's a quick overview of the workflow:
1. Create a ZDefocus node and connect it to your script.
   See Connecting ZDefocus.
2. Adjust the blur settings.
   See Adjusting Blur Settings.
3. Adjust the shape of any out-of-focus highlights.
   See Adjusting the Shape of Out-of-Focus Highlights.
4. If necessary, enhance the highlights to make lens shape effects more visible.
   See Enhancing Out-of-Focus Highlights.
5. If necessary, mask the blur effect.
   See Masking Blur Effects.

Tip: You can check Use GPU if available to have the node run on the graphics processing unit (GPU) rather than the central processing unit (CPU).

For more information on the minimum requirements, please see Windows, Mac OS X and macOS, or Linux or refer to the Nuke Release Notes available in Help > Release Notes.

You can select the GPU to use in the Preferences. Press Shift+S to open the Preferences dialog, make sure you're viewing the Preferences > Performance > Hardware tab, and set default blink device to the device you want to use. You must restart Nuke for the change to take effect.

If you are using a render license of Nuke, you need to add --gpu on the command line.

Connecting ZDefocus

To connect ZDefocus:
1. Create a ZDefocus node by clicking Filter > ZDefocus.
2. Connect the ZDefocus node's `image` input to the image you want to blur. Note that this image also needs to contain a depth map channel. If your depth channel and rgba channels exist in two different files, you can use a ShuffleCopy node to combine them.

3. Use the `channels` dropdown menu to select the channels you want to blur.

4. Set `depth channel` to the channel in the `image` input that contains the depth map. By default, the depth information is taken from `depth.Z`. Note that the depth map should not be anti-aliased. If it is, pixels along an edge between two objects can be assigned a depth that is in-between the depth of the front object and back objects. This looks wrong, as it suggests that those edge pixels are floating somewhere between the objects.

5. If you want to use your own filter image (rather than the predefined `disc` or `bladed` images), connect that to the ZDefocus node's `filter` input and set `filter type` to `image` in the node's properties.

   The filter image represents the shape and size of the camera aperture used to shoot the input footage. As the clip in the `image` input is blurred, any highlights in the clip bloom in the shape of the `filter` image, creating a bokeh effect.

   **Note:** Loading scripts from pre-Nuke 8.0v7 enables the `legacy resize mode` checkbox automatically, for backward compatibility, and uses the `filter bounds` dropdown to determine how images used in filtering are resized.

   Adding new ZDefocus nodes hides the `legacy resize mode` checkbox and allows you to use the `image filter` dropdown to give you more flexibility when calculating blur.

   See Using a Custom Filter Image for more information.

You can create a filter image using the Roto node (Draw > Roto) or the Flare node (Draw > Flare), for example.

Note that you don't necessarily need to crop the filter image to a smaller size, as fast Fourier transforms are used to speed up convolutions with large filter images.

6. Attach a Viewer to the ZDefocus node.
A ZDefocus script.

7. Proceed to Adjusting the Shape of Out-of-Focus Highlights below.

Adjusting Blur Settings

To adjust the Blur settings:

1. Use `math` to specify how you want to use the `depth channel` to calculate the distance between the camera and an object. For example, some programs use higher values to denote further away, while in others they mean closer to the camera:
   - `direct` - The Z value in the `depth channel` directly controls blur. For example, if Z is 0.5, then the blur size is 0.5 times the value of the size control (unless this is bigger than `maximum`, in which case it is clamped to `maximum`).
   - `depth` - The Z value in the `depth channel` is the distance between the camera and whatever is in the image at that pixel.
   - `far = 0` - The Z value in the `depth channel` is equal to 1/distance. The values are expected to decrease from large positive values close to the camera to zero at infinity. This is compatible with depth maps generated by Nuke and RenderMan.
   - `far = 1` - Near plane = 0, far plane = 1. This is compatible with depth maps generated by OpenGL.
   - `-direct` - As with the `direct` mode, the Z value in the `depth channel` directly controls blur. In other words, each layer is blurred by the same amount as in the `direct` mode. However, in this mode, the layers are interpreted as being in the opposite order, so a higher depth value places a layer in front of another rather than behind it.
   - `-depth` - The Z value in the `depth channel` is -distance in front of the camera. This is the same as `depth`, but the distances are negative to start with.
   - `far = -0` - The Z value in the `depth channel` is equal to -1/distance. This is compatible with depth maps generated by Maya.
   - `far = -1` - Near plane = 0, far plane = -1.

2. In the Viewer, drag the `focal point` widget on top of the area that you want to be entirely in focus. This automatically updates the `focal point` coordinates in the ZDefocus properties and sets the `focus plane` control to the Z depth value at those coordinates. Any layers with this Z depth value are left in focus. You can also set a depth slice around the focus plane that is in focus (as described in step 4), but any other areas of the image are blurred according to the depth map.
Focal point widget under the green sign.

Focal point widget near the cars in the distance.

**Tip:** The fill foreground control, enabled by default, attempts to compensate for missing information by filling regions in the foreground which are revealed when the foreground goes out of focus. However, because the true image information isn't available in these regions, fill foreground can sometimes introduce undesirable artefacts by adding things which aren't there.

If you see blurry artefacts in the foreground, rather than sharp edge artefacts, try disabling this control.

To better see the effect of your changes, you can also set output to focal plane setup. ZDefocus now displays depth-of-field (DOF) info in the rgb channels:

- **red** - Less than DOF (in front of the area that's in focus).
• **green** - Inside DOF (in focus). Note that if **depth of field** is set to 0, nothing is displayed in green.
• **blue** - Greater than DOF (behind the area that’s in focus).

When **show image** is enabled, this information is overlaid upon the input image.

![Focal point widget under the green sign.](image1)
![Focal point widget near the cars in the distance.](image2)

3. To widen the area that is entirely in focus, increase **depth of field**. This sets a depth slice around the focus plane that is entirely in focus (and shown in green whenever **output** is set to **focal plane setup**). True theoretical depth of field would set this to zero.

![Depth of field set to 0.](image3)
![Depth of field set to 1.](image4)

4. To apply a small amount of blur to the in-focus region, make sure **blur inside** is enabled. This gives a smoother transition between the in-focus region and the out-of-focus regions around it.

5. Set **output** back to **result** and adjust the amount of blur at infinite depth by setting the **size** value. Note that the amount of blur nearer the camera than the focus plane may be larger.

   If you have set **math** to **direct**, the size is multiplied by the depth to give the blur size at that depth. Setting **size** to 1 allows you to use the values in the depth map as the blur size directly.

6. If you increased the **size** value in step 5, it’s a good idea to also increase the **maximum** value. No blurring greater than this value is generated no matter where the object is in relation to the camera.

   For maximum processing speed, you may want to keep this value as low as possible.

7. By default, **automatic layer spacing** is enabled, which means ZDefocus automatically works out how many depth layers to use, based on the maximum blur size (**maximum**). In this mode, the layers are closer together near to the focal plane, where a small change in the blur amount is more obvious, and increasingly more widely-spaced further away (this is equivalent to setting **layer curve** to a value of 1 when controlling the layers manually; see step 8).
To visualize the layers, you can set output to layer setup. This is like focal plane setup, but displays depth-of-field (DOF) information after the depth has been divided into layers.

Output set to layer setup.

The maximum number of blur sizes that are used between 0 and maximum is 256. This means you can have up to 256 layers behind the focal plane, and up to 256 in front of it as well.

8. If you uncheck automatic layer spacing, you can control the number of layers manually using depth layers. Note that the more layers you use, the longer it takes to process the blur.

Depth layers set to 10. Depth layers set to 50.

If necessary, you can also increase layer curve to concentrate the layers around the focal plane. The default value of 0 produces evenly spaced layers.

Layer curve set to 0. This produces evenly spaced layers. Increasing the layer curve value concentrates the layers around the focal plane.

9. Proceed to Adjusting the Shape of Out-of-Focus Highlights.
Adjusting the Shape of Out-of-Focus Highlights

As the clip in the image input is blurred, any out-of-focus highlights (bokeh) in the clip assume the shape of the filter image (of the filter type selected).

The original image.  Out-of-focus highlights (bokeh) in the shape of the filter image.

How you create the filter image is up to you. You can:

• Use a predefined disc shape as the filter image. See Using a Predefined Disc Image.
• Use a predefined bladed image (polygon) as the filter image. See Using a Predefined Bladed Image.
• Use your own custom image in the filter input as the filter image. See Using a Custom Filter Image.

Using a Predefined Disc Image

1. To see the effect of your changes, set output in the ZDefocus controls to filter shape setup.
2. Set filter type to disc.
3. Use the filter shape control to dissolve the filter shape between Gaussian at 0 and disc at 1.

Filter shape set to 0 (Gaussian). Filter shape set to 1 (disc). This is the default value.

4. Use the aspect ratio control to set the filter aspect ratio, which is 1:1 by default. Values less than 1 squeeze the filter on the x axis, and values larger than 1 squeeze it on the y axis.

This allows you to simulate the cat’s eye effect, caused by vignetting inherent within some lens designs.
5. Proceed to Enhancing Out-of-Focus Highlights.

Using a Predefined Bladed Image

1. To see the effect of your changes, set output in the ZDefocus controls to filter shape setup.
2. Set filter type to bladed.
3. Use the blades control to set the number of iris blades that make up the camera's diaphragm. A value of 3 produces a triangle, 4 a square, 5 a pentagon, 6 a hexagon, and so on.
   This field only accepts integers larger than 1.

4. Adjust roundness to control the rounding of the filter polygon's sides. A value of 0 equals no rounding.

5. Use rotation to rotate the filter image in degrees. Positive values produce counter-clockwise rotation, and vice-versa.
6. Use the **aspect ratio** control to set the filter aspect ratio, which is 1:1 by default. Values less than 1 squeeze the filter on the x axis, and values larger than 1 squeeze it on the y axis.

This allows you to simulate the cat’s eye effect, caused by vignetting inherent within some lens designs.

![Aspect ratio comparison](image)

**Aspect ratio** set to 0. **Aspect ratio** set to 2.

7. To adjust the distribution of light inside the out-of-focus highlights:
   - Adjust **inner size** to control the size of the inner polygon, as a percentage of the outer polygon.

![Inner size comparison](image)

**Inner size** set to 0.2. **Inner size** set to 0.8.

- Use **inner feather** to add outward or inward feathering around the inner polygon. With values larger than 0.5, your feather effect is outward and, respectively, if your values are smaller than 0.5, the feather effect is inward. A value of 0.5 produces no feathering.

![Inner feather comparison](image)

**Inner feather** set to 0.2. **Inner feather** set to 0.8.

- Adjust inner brightness to control the brightness of the inner polygon, where 0 is equal to black and 1 to white.
Simulating Depth-of-Field Blurring | Adjusting the Shape of Out-of-Focus Highlights

Inner brightness

- Inner brightness set to 0.3.
- Inner brightness set to 0.7.

- If you want to simulate catadioptric lenses, check **catadioptric**. When using catadioptric lenses, the defocused areas of the image are annular, producing donut-shaped bokeh. You can use **catadioptric size** to adjust the size of the catadioptric hole in the filter.

Catadioptric size

- Catadioptric size set to 0.3.
- Catadioptric size set to 0.7.

8. Proceed to **Enhancing Out-of-Focus Highlights**.

Using a Custom Filter Image

1. Set **filter type** in the ZDefocus controls to **image**.
   This tells ZDefocus to use the **filter** input rather than a predefined shape as the filter image.
   Note that the **filter** image can be a color image. This can be useful, for example, if you want to add color fringing to your out-of-focus highlights to simulate chromatic aberration.

   The **filter** image.

2. If you want to display the **filter** image in the Viewer, set **output** to **filter shape setup**.
3. To select what channel to use from the filter input, do one of the following:
   • Set the filter channel dropdown menu to the channel you want to use. By default, this menu is set to rgba.alpha.
   • To use the same channels from the filter input as the image input (that is, whatever channels is set to), check use input channels.
4. In newer Nuke scripts, use the image filter dropdown to select the required filter. See Choosing a Filtering Algorithm for more information on the available filters.

Note: When using filters that employ sharpening, such as Rifman and Lanczos, you may see a haloing effect. If necessary, check clamp image filter to correct this problem.

In older Nuke scripts with legacy resize mode enabled, use the filter bounds dropdown to select whether you want to limit the filter image to:
   • shape - the filter input's bounding box. In this case, ZDefocus only uses the bounding box area, and the center of the filter is the center of the bounding box. This is the default value. You may want to use it, for example, if your filter input is a roto shape with a small bounding box that doesn't fill the entire format area.
   • format - the filter input's format. In this case, ZDefocus uses the entire format area, allowing you to offset the filter image within the format.
5. Proceed to Enhancing Out-of-Focus Highlights.

Enhancing Out-of-Focus Highlights

To enhance the out-of-focus highlights:
1. To make bokeh lens shape effects warmer and more visible, check gamma correction.
   This means a gamma lookup curve of 2.2 is applied before blurring and then reversed for the final output.
2. You can also make the lens shape effects more visible by enabling **bloom**. Any highlights above the **bloom threshold** value are multiplied by the **bloom gain** value. Highlights below the **bloom threshold** are not affected. This allows you more control over the highlights than **gamma correction**; however, **gamma correction** may bring out some of the highlights better.

3. Proceed to **Masking Blur Effects**.

**Masking Blur Effects**

To mask the blur effect:

1. Do **one** of the following:
   - Make sure there is a mask channel in the ZDefocus node's **image** input and nothing is connected to the **mask** input.
   - Connect a mask to the **mask** input of the ZDefocus node. If you cannot see the **mask** input, open the node's controls and make sure **mask** is set to **none**.
     - If you want the mask from the **mask** input copied into the predefined **mask.a** channel, also check **inject**. This way, you can use the **mask** input again downstream.

2. If you don't want to use the alpha channel as the matte, select the channel you want to use from the **mask** dropdown menu.
   - By default, the blur is limited to the non-black areas of this channel.

3. If necessary, check **invert** to reverse the mask, so that the blur is limited to the non-white areas of the mask.

4. To blur the edges of the mask, check **fringe**.
5. To dissolve between the original image (at 0) and the full ZDefocus effect (at 1), adjust the **mix** slider. A small light-gray square appears on the node in the Node Graph to indicate that the full effect is not used.
Creating Effects

Several nodes in Nuke let you create various effects on your input images. In this chapter, we describe three of these nodes: LightWrap, Glint, and Text.

Quick Start

Here's a quick overview of the workflow:

1. To adjust the soft edges and light spills that occur in the border between your foreground and background, you can use the LightWrap node (Draw > LightWrap). For more information, see Background Reflections on Foreground Elements.

2. With the Glint node you can add star-shaped glints on your image. See Creating Star Filter Effects on Image Highlights.

3. Using the Text node, you can add text elements, such as credits, to your footage. For more information, see Creating Text Overlays.
Background Reflections on Foreground Elements

You can use the LightWrap node to create background reflections on foreground elements. The node creates a reflection of light around the edges of your foreground element by blending in whatever is in the background.

The composite without the LightWrap effect.  
The composite with the LightWrap effect.

You may have noticed that objects filmed in front of very bright backgrounds have the appearance of softened edges as the light spills round from the background. When adding foreground layers onto a background plate, using the LightWrap node can dramatically improve the quality of your composite.

The composite without the LightWrap effect.  
Note the hard edges around the trailing hand and the bottom of the skirt.

The composite with the LightWrap effect.  
Note how the foreground fits better into the background.

If you want to use LightWrap, you should apply it on your foreground element before you composite the foreground over the background with the Merge node.
Using the LightWrap Node

To use the LightWrap node:

1. Select **Draw > LightWrap** to add a LightWrap node after your foreground and background images.
2. Connect your foreground element to input **A** of the LightWrap node, and the background image to input **B**.
3. Connect a Viewer to the output of the LightWrap node so you can see the effect of your changes.
4. Adjust the **Diffuse** and **Intensity** sliders to control both the spread and brightness of the reflections on the foreground element. These sliders need to be balanced out together. You may want to start by bringing Diffuse all the way down to better see what you are blending in from the background. Then, adjust Intensity before going back to the Diffuse slider and, if necessary, Intensity again until you are happy with the result.

![Low Diffuse](image1)

![High Diffuse](image2)

![Low Intensity](image3)

![High Intensity](image4)

5. If you want to create a uniform effect around the edges of the foreground rather than have the effect adjust itself according to the background, check **Disable luminance based wrap** on the **LightWrap** tab.
6. In case you don’t want to merge the LightWrap effect with the foreground element in order to keep the LightWrap effect as a separate element, check **Generate wrap only** on the **LightWrap** tab.
7. By default, the LightWrap effect is only applied inside the foreground element’s alpha. If you want to extend the effect outside it, making the element seem to glow, check Enable Glow.

8. On the Tweaks tab, you can also adjust the following controls:
   • FGBlur to determine how much the foreground matte is blurred. The more blur, the more of the background is added to the foreground.
   • BGBlur to control how much the background is blurred before it is merged with the foreground element.
   • Saturation to adjust the saturation of the effect.
   • Luma Tolerance to increase or decrease the luminance values of the effect.
   • Highlight Merge to control how the foreground element is merged with the background. The default merge operation, called plus, adds the elements together, producing a glow effect.
   • Check Use constant highlight to use a constant color of your choice rather than the background in the LightWrap effect. Select the color using the controls next to Constant Highlights Color.

9. On the CCorrect tab, you can color correct the LightWrap effect produced.
Creating Star Filter Effects on Image Highlights

With the Glint node, you can create star-shaped rays around all the bright points in an image.

The original image with bright points.

The image after using the Glint node.

Using the Glint Node

To use the Glint node:

1. Select Draw > Glint to add a Glint node after the image you want to add star-shaped rays to.
2. From the channels dropdown menu and checkboxes, select the channels to which you want to apply the effect.
3. In the no. of rays field, enter the number of rays you want coming out of the bright points in your image. For example, if you want to create five-pointed stars, enter 5.
4. To change the threshold for how bright the highlights in the input image need to be to cause the glint effect, adjust the tolerance slider. Only the pixels above the threshold bloom with the effect.

Low tolerance value. High tolerance value.

5. To determine the length of the rays, adjust the length slider. To give every other ray a different length and determine that length, adjust the odd ray length slider.
6. To determine how many steps the rays are formed of, enter a value in the **steps** field. The more steps you use and the shorter the rays are, the smoother the rays become.

   ![Image showing steps value set to 3 and 5.]

   The **steps** value set to 3.    The **steps** value set to 5.

7. To rotate the star-shapes, adjust the **rotation** slider. Increasing the value rotates the rays clockwise, whereas decreasing the value rotates them counter-clockwise.

8. To change the color in the beginning of the rays near the center point of the stars, adjust the **from color** slider. To change the color in the end of the rays, adjust **to color**. By default the from color is set to white, and the to color to black.

   ![Image showing from color set to white and to color set to black.]

   **From color** set to white and **to color** set to black.    ![Image showing from color set to pink and to color set to yellow.]

   **From color** set to pink and **to color** set to yellow.

9. If needed, you can also make the following adjustments:
   • If you want to change the aspect ratio of the stars, adjust the **aspect ratio** slider.
   • By default, the brightest image on the rays is used as the center point for the star. However, if you prefer the images forming the rays to be added up in forming the center point, uncheck **max**.
   • To only output the Glint effect without merging it into the original input image used to create it, check **effect only**.
Creating Star Filter Effects on Image Highlights | Using the Glint Node

Effect only disabled.  Effect only enabled.

- To mask the shape that is used to create the rays, check w and select the mask channel from the dropdown menu.
- To perform a gamma correction on the highlights that cause glint before the glint effect is applied, adjust the gamma slider.
- To mask the glint effect, check mask and select a mask channel using the controls on the right.
- To dissolve between the original input image and the full glint effect, adjust the mix slider.
Creating Text Overlays

Using Nuke's Text node, you can add text overlays on your images. You can simply type in the text you want to have displayed, or use Tcl expressions (such as [metadata values]) or Tcl variables to create a text overlay.

See Entering Text for a list of Tcl expressions, Tcl variables, HTML named entities, hex entities, and decimal entities you can use in the Text node.

Text overlays can also be animated using animation layers so that their properties, such as position, size, and color, change over time. These features make the Text node useful, for example, for creating slates or scrolling credits.

Preparing a Text Overlay

Creating text overlays involves some preparation, particularly which channels hold the text and the positioning of the initial text box.

1. Select Draw > Text to create a Text node and connect it to a Viewer.
2. In the Text node properties, select the channels you want the text to appear in from the output controls.
3. If you want to multiply any channels by the drawn text so that they are set to black outside the text shape, select those channels using the premult controls.
4. From the clip to dropdown menu, select how you want to restrict the output image.
   - no clip - do not restrict the output image.
   - bbox - restrict the output image to the incoming bounding box.
   - format - restrict the output image to the incoming format area.
   - union bbox+format - restrict the output image to the combination of the incoming bounding box and format area.
   - intersect bbox+format - restrict the output image to the intersection of the incoming bounding box and format area.
5. If you want to clear the affected channels to black before drawing on them, check replace. By default, replace is not enabled and the text is drawn on top of the input image.
6. If you're happy with the text initially appearing in the top left corner of the format area, you can start Entering Text using the message field.
7. If you'd like to have more control over the text's initial position or draw a text box to constrain your text, ensure that edittext is enabled above the Viewer and then either:
   - Click in the Viewer to place the cursor where you want the text to appear,
OR

- Draw a box in the Viewer to contain your text.

Tip: The cursor appears as in the Viewer when you initially add the Text node.

Text entered in a custom box is limited by the box, r, and t boundaries on the Transform tab, but the y boundary can be overstepped.

Tip: You can adjust the box at any time using the boxyrt controls or by dragging the Viewer handles.

8. If you want to mask out part of the image, see Masking Regions of the Viewer
9. Once you’re happy with the cursor position, proceed to Entering Text.
Entering Text

The message field in the properties panel is used to enter the text, Tcl expressions, Tcl variables, or a combination of these that you want to display in your output. For some examples of these formats, see the table below.

Text entry behaves much the same as regular text editors, but there are a few rules to observe:

- The edit text control above the Viewer must be enabled if you want to type directly in the Viewer, though you can enter text in the message field at any time.
- Press Return to begin a new line in both the Viewer and the message field.
- Navigate around the Viewer or message field using the arrow keys.
- Tcl expressions must be placed in square brackets, such as [date].
- To display special Unicode characters, such as foreign language characters and copyright signs, you can:
  - Use HTML named entities, such as &lt; to display <, or &copy; to display ©.
  - Use hex entities, such as &#x3c; to display <, or &#xa9; to display ©.
  - Use decimal entities, such as &#60; to display <, or &#169; to display ©.
  - Type Unicode characters, such as < or ©, on your keyboard or cut and paste them from other applications. UTF-8 character encoding is used to store them in the control's value and in the saved Nuke script.

These special characters only work if the font you are using supports the required character.

Note: We recommend using the above entities rather than typing <, for example. This is because future versions of the Text node may interpret HTML mark-up. In HTML, some characters, such as the greater than and less than signs, are reserved. If you used these signs within your text now, future versions could mistake them for HTML mark-up.

Example Variables and Entities

The following table gives examples of Tcl expressions, Tcl variables, HTML named entities, hex entities, and decimal entities you can use in the message field of the Text node.
<table>
<thead>
<tr>
<th>Message</th>
<th>Prints</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tcl expressions</td>
<td></td>
</tr>
<tr>
<td>[date]</td>
<td>Week day, day, month, hh:mm:ss, and time zone. For example, <em>Thu Jan 15 14:22:20 GMT.</em></td>
</tr>
<tr>
<td>[date %a]</td>
<td>Abbreviated week day name. For example, <em>Thu.</em></td>
</tr>
<tr>
<td>[date %A]</td>
<td>Full week day name. For example, <em>Thursday.</em></td>
</tr>
<tr>
<td>[date %b]</td>
<td>Abbreviated month name. For example, <em>Jan.</em></td>
</tr>
<tr>
<td>[date %B]</td>
<td>Full month name. For example, <em>January.</em></td>
</tr>
<tr>
<td>[date %d]</td>
<td>Day (01-31).</td>
</tr>
<tr>
<td>[date %D]</td>
<td>Date (dd/mm/yy). For example, <em>15/01/10.</em></td>
</tr>
<tr>
<td>[date %H]</td>
<td>Hour (00-23).</td>
</tr>
<tr>
<td>[date %I]</td>
<td>Hour (01-12).</td>
</tr>
<tr>
<td>[date %m]</td>
<td>Month (01-12).</td>
</tr>
<tr>
<td>[date %M]</td>
<td>Minutes (00-59).</td>
</tr>
<tr>
<td>[date %p]</td>
<td>AM or PM.</td>
</tr>
<tr>
<td>[date %r]</td>
<td>Time (12-hour clock). For example, <em>11:04:07 AM.</em></td>
</tr>
<tr>
<td>[date %S]</td>
<td>Seconds (00-59).</td>
</tr>
<tr>
<td>[date %T]</td>
<td>Time (24-hour clock). For example, <em>14:06:54.</em></td>
</tr>
<tr>
<td>[date %y]</td>
<td>Abbreviated year (00-99). For example, <em>10.</em></td>
</tr>
<tr>
<td>[date %Y]</td>
<td>Full year. For example, <em>2010.</em></td>
</tr>
<tr>
<td>[date %z]</td>
<td>Numeric time zone. For example, <em>-0800.</em></td>
</tr>
<tr>
<td>[date %Z]</td>
<td>Time zone. For example, <em>GMT.</em></td>
</tr>
<tr>
<td>[frame]</td>
<td>Frame number. For example, <em>23.</em></td>
</tr>
<tr>
<td>[metadata]</td>
<td>List of all the keys in the incoming metadata.</td>
</tr>
<tr>
<td>Message</td>
<td>Prints</td>
</tr>
<tr>
<td>---------</td>
<td>--------</td>
</tr>
<tr>
<td>[metadata values]</td>
<td>List all of the keys and values in the incoming metadata.</td>
</tr>
<tr>
<td>[metadata key]</td>
<td>Value of the key in the incoming metadata. Replace key with the name of the key whose value you want to display. For example, you may be able to use [metadata input/filename] to display the name and location of the image file, or [metadata input/ctime] to display the timestamp for an input file.</td>
</tr>
<tr>
<td>[value root.name]</td>
<td>Script directory path and script name. For example, Users/john/Nuke_scripts/myscript.nk.</td>
</tr>
</tbody>
</table>

**Tcl variables**

| $env(ENVIRONMENT_VARIABLE) | The value of the environment variable specified. Replace ENVIRONMENT_VARIABLE with an environment variable you have set. For example, you can use $env(USER) to display the user name (for example, john) on Mac and Linux, or $env(USERNAME) to display it on Windows and Linux. For a list of environment variables specific to Nuke, see Environment Variables. |
| $version_long | The full version number of Nuke. For example, 11.3v6. |
| $threads | Number of render threads used to calculate images. This is in addition to a main thread used to update the graphical user interface (GUI). |

**HTML named entities**

<p>| &amp; | &amp; |
| ' | ’ |
| Å | Å |
| Á | Á |
| Â | Â |
| Æ | Æ |
| À | Å |
| Ç | Ç |</p>
<table>
<thead>
<tr>
<th>Message</th>
<th>Prints</th>
</tr>
</thead>
<tbody>
<tr>
<td>©</td>
<td>©</td>
</tr>
<tr>
<td>é</td>
<td>é</td>
</tr>
<tr>
<td>ê</td>
<td>è</td>
</tr>
<tr>
<td>è</td>
<td>è</td>
</tr>
<tr>
<td>ë</td>
<td>è</td>
</tr>
<tr>
<td>€</td>
<td>€</td>
</tr>
<tr>
<td>&gt;</td>
<td>&gt;</td>
</tr>
<tr>
<td>&lt;</td>
<td>&lt;</td>
</tr>
<tr>
<td>Ñ</td>
<td>Ñ</td>
</tr>
<tr>
<td>ø</td>
<td>ø</td>
</tr>
<tr>
<td>õ</td>
<td>ö</td>
</tr>
<tr>
<td>Ö</td>
<td>Ö</td>
</tr>
<tr>
<td>ö</td>
<td>ö</td>
</tr>
<tr>
<td>&quot;</td>
<td>&quot;</td>
</tr>
<tr>
<td>®</td>
<td>®</td>
</tr>
<tr>
<td>ß</td>
<td>ß</td>
</tr>
<tr>
<td>Ü</td>
<td>Ü</td>
</tr>
<tr>
<td>ü</td>
<td>ü</td>
</tr>
</tbody>
</table>

**hex entities**

<table>
<thead>
<tr>
<th>Hex entity</th>
<th>ASCII</th>
</tr>
</thead>
<tbody>
<tr>
<td>#</td>
<td>#</td>
</tr>
<tr>
<td>%</td>
<td>%</td>
</tr>
<tr>
<td>&amp;</td>
<td>&amp;</td>
</tr>
<tr>
<td>*</td>
<td>*</td>
</tr>
<tr>
<td>Message</td>
<td>Prints</td>
</tr>
<tr>
<td>-----------</td>
<td>--------</td>
</tr>
<tr>
<td>@</td>
<td>@</td>
</tr>
<tr>
<td>�</td>
<td>™</td>
</tr>
<tr>
<td>œ</td>
<td>ò</td>
</tr>
<tr>
<td>š</td>
<td>š</td>
</tr>
<tr>
<td>&lt;</td>
<td>&lt;</td>
</tr>
<tr>
<td>&gt;</td>
<td>&gt;</td>
</tr>
<tr>
<td>©</td>
<td>©</td>
</tr>
<tr>
<td>é</td>
<td>é</td>
</tr>
</tbody>
</table>

**decimal entities**

<table>
<thead>
<tr>
<th>Message</th>
<th>Prints</th>
</tr>
</thead>
<tbody>
<tr>
<td>£</td>
<td>£</td>
</tr>
<tr>
<td>©</td>
<td>©</td>
</tr>
<tr>
<td>®</td>
<td>®</td>
</tr>
<tr>
<td>¿</td>
<td>¿</td>
</tr>
<tr>
<td>ê</td>
<td>è</td>
</tr>
<tr>
<td>ß</td>
<td>ß</td>
</tr>
<tr>
<td>à</td>
<td>à</td>
</tr>
</tbody>
</table>

**Tip:** To get a list of all the Tcl expressions you can use with `date`, type `x` on the Node Graph, set the script command dialog that opens to `Tcl`, enter `date -h`, and click `OK`. 
Fonts and Font Properties

The FreeType library used by the Text node supports a large number of fonts, including TrueType, OpenType, and PostScript fonts. In order to display the available fonts faster in the font dropdown menu, these fonts are cached as an XML file when you open a Text node's properties panel. The font cache XML file is called `fontmapping.fcache` and its location is specified by the environment variable NUKE_TEMP_DIR. See Updating the Font Cache for more information.

**Note:** See Environment Variables for more information on setting environment variables.

**Note:** The font cache file is not used when rendering and is not required on a render farm.

Nuke retrieves the fonts available in the Text node from various locations in a set order, before caching them:
1. The location specified in the Project Settings > Font > project font path control.
2. The `.nuke/fonts` directory and all other plug-in folders.
3. The location specified by the NUKE_FONT_PATH environment variable.

**Note:** Locations 1 and 3 allow you to specify multiple paths using the OS standard syntax. On Windows, for example, you could specify "C:\windows\font1;c:\windows\font2;..."

4. The local fonts folder.
5. The system fonts folder, assuming that Project Settings > Font > include system fonts is enabled. For example, on Windows `C:\Windows\Fonts`.

**Note:** If several locations contain the same font, Nuke uses the font from the first directory it finds it in.

In general, there is no guarantee that a font family and style, such as `{Arial : Regular}`, will render identically cross-platform. To avoid this, Nuke ships with several fonts common to all platforms. If you disable Project Settings > Font > include system fonts, only the fonts that ship with Nuke are available in the font controls.

See Third-Party Libraries and Fonts for a list of fonts that ship with Nuke.
**Note:** Only fonts that are physically stored in a file are available in the font controls. As a result, the list of system fonts available in Nuke may be different to that available in other software applications on the same machine.

## Selecting a Font

1. Highlight the text you want to affect in the message field or Viewer.
2. In the Text properties panel, click the font family dropdown to select the required family.
3. Select the required font style from the dropdown.

### Note: The styles available depend on the family selected.

4. Click the information icon to display the location of the selected font. For example:

![Font Selection Panel](image)

The font family and style used are saved in the Nuke script, along with some extra information, in order to make sure the same family and style are used if the script is opened on a different machine or operating system. If the font family and style are not the same, Nuke displays a warning message.

## Adjusting Font Size and Spacing

1. Highlight the text you want to affect in the message field or Viewer.
2. To adjust the overall size of the font, use the font size slider. When leading is set to 0, this parameter also controls the spacing between each line of text.

   When rendering the font, font size controls the font hinting used. Font hinting adjusts which pixels are interpolated to more clearly render a font. At small sizes and on low resolution output devices, it has a big impact on the legibility of the font. For best results, you should use the font size parameter (rather than the scale control on the Group tab) to control the size of the font and keep scale set to 1.
Tip: Use the **global font scale** control to adjust the scale multiplier for all characters in the current Text node, regardless of the **font size** specified for individual characters.

3. You can adjust fonts asymmetrically using the **font width** and **font height** controls. You can also use the controls above the Viewer to adjust font size.

4. To increase or decrease the spacing between individual characters, adjust the **kerning** slider. By using negative values, you can make the letters overlap.

Note: If you select more than one character or the last character in a group, the **kerning** control is disabled.

You can also use the control above the Viewer to adjust kerning.

5. To increase or decrease the space between each character and the previous character adjust the **tracking** control. Negative values move characters towards the preceding character. You can also use the control above the Viewer to adjust tracking.

6. To adjust the selected text's height above the baseline, in screen space, use the **baseline shift** control. The baseline is the imaginary line upon which most letters rest. This control allows you to reliably line up the baseline of different fonts. You can also use the control above the Viewer to adjust baseline.

7. If your text overlay contains several lines of text, you can adjust the spacing between each line by using the **leading** slider. By using negative values, you can make the letters overlap.
Note: Unlike the baseline shift control, leading affects all text within the bounding box regardless of selection.

Justifying Fonts

The justify controls affect the positioning of the text within the bounding box on the x and y axes.

1. Use the xjustify dropdown to control how you want to align the text horizontally:
   - left - align the text along the left edge of the on-screen text box. This leaves the right edge of the text ragged.

```
```

- center - align the text from the center of the on-screen text box. This leaves both edges of the text ragged.

```
```

- right - align the text along the right edge of the on-screen text box. This leaves the left edge of the text ragged.
• **justify** - align the text both along the left and the right edge of the on-screen text box. This leaves no ragged edges. The justification is done by expanding the spaces between letters. If there are no spaces or the spaces get more than about three times wider than they were, letters are expanded.

2. Use the **justify** dropdown to control how you want to align the text vertically:
   - **top** - align the text against the top edge of the on-screen text box.
   - **center** - align the text from the center of the on-screen text box.
• **bottom** - align the text against the bottom edge of the on-screen text box.
Updating the Font Cache

Once the font cache has been generated, Nuke accepts the content of the cache and no longer generates it from scratch every time it is requested by the font controls. If you add or remove fonts, the cache is not automatically updated and you need to update it manually. There are several ways to do so:

• Navigate to Project Settings > Font and click Rescan font paths.
• Manually remove the fontmapping.fcache.xml font cache file from NUKE_TEMP_DIR and restart Nuke.
• Open Nuke's Script Editor and execute nuke.rescanFontFolders()

Go to Help > Documentation > Python Developer's Guide for more information on Python commands relating the font cache.
Transforming Text

The Text node allows you to transform selections in the Viewer or create groups of selections in the animation layers table and transform them together. Both methods support animation using keyframes.

Transforming Selections

Selections in the Viewer are transformed by hand, that is, by manipulating handles drawn on your selections.

1. Enable **transform** in the controls above the Viewer.
2. Select the required character(s) in the Viewer to enable the transform handles.
3. If you hover the cursor over a selection, the cursor changes to show the handle's function depending on position.

4. Translate, rotate, and scale your selections by dragging the required handles.
   If you want to animate your changes, see *Animating Transforms* for more information.

**Tip:** By default, scaling is done uniformly around the center of your selections. To scale relative to the opposite corner or side instead, hold down **Ctrl/Cmd** while dragging the transform handles.
Transforming Groups

The **Group** tab saves group selections in the **animation layer** table allowing you to affect layers separately using the transform controls.

1. Open the properties and click the **Group** tab to display the transform controls and **animation layers** table.

2. Select the required character(s) in the Viewer, or **message** field, and then click the + button to create an animation layer, or click **create group** above the Viewer.

   You can create as many animation layers as required, each one is named according to the selections you made in the Viewer.

3. Select an animation layer and use the transform controls on the **Group** tab to transform your selection.

   ![Transforming Text](image)

   If you want to animate your changes, see **Animating Transforms** for more information.

4. Click the - button to remove an animation layer.
Animating Transforms

The Text node supports animation in a similar way to most other Nuke nodes, in that you can animate individual controls in the properties panel, but the **Group tab animation layer** table enables you to visualize your animations more easily.

**Animating Selections**

1. Set the starting position for the selection(s) using the Viewer handles as described in Transforming Text.
2. Click the button above the Viewer to add a starting keyframe.
3. Move the playhead to a new frame and transform the selection using the Viewer handles.
   A keyframe is added automatically on the new frame.
4. Repeat the process to produce the required animation.
5. Click to remove a keyframe at the current frame.

**Animating Groups**

1. Set the starting position for the groups using the transform controls as described in Transforming Text.
2. Select a group in the animation layers table and then click the button above the Viewer to add a starting keyframe.
3. Move the playhead to a new frame and transform the group using the transform controls or the handles in the Viewer.
   A keyframe is added automatically on the new frame.
4. Repeat the process to produce the required animation.
5. Click to remove a keyframe at the current frame.
Changing the Text Color

To change the text color:

1. In the Text node properties, click the **Color** tab.

2. Adjust the **color** parameter or click on the color picker button 🌈 to select a color for the text.

3. If you want to create a color gradient across the text, select anything other than **none** from the ramp:
   - **linear** - the ramp changes linearly from one color into another.
   - **smooth0** - the ramp color gradually eases into the point 0 end. This means colors in the point 0 end are spread wider than colors in the point 1 end.
   - **smooth1** - the ramp color eases into the point 1 end. This means colors in the point 1 end are spread wider than colors in the point 0 end.
   - **smooth** - the ramp color gradually eases into both ends. This means colors in the point 0 and point 1 ends are spread wider than colors in the center of the ramp.

4. Use the **color** control to select a color for the ramp at the point 1 end (by default, the top end). Then, use **color 0** to select the color for the ramp at the point 0 end (by default, the bottom end).

5. To adjust the spread and angle of the ramp, drag the **p0** and **p1** Viewer handles to a new location. You may need to press **O** twice on the Viewer to make these controls appear. Alternatively, you can enter new **x** and **y** values for the **point 1** and **point 0** controls in the Text node properties.
Masking Regions of the Viewer

If you want to mask the effect of the Text operation, the easiest way is to use a Roto node connected to the **mask** input:

1. After connecting the Roto node, draw the mask in the Viewer. You can use any channel, but Roto defaults to the **alpha** channel.
2. Select the mask channel from the Text node's **mask** dropdown menu, in this case **rgba.alpha**.
3. To invert the mask as shown, check **invert**.

![Standard masking.](image1.png) ![Inverted masking.](image2.png)
Adding Shadows to Text

You can add shadows to your text using the Shadows tab in the Text node's properties.

To add a shadow to your text, simply select the **Shadows** tab and check **enable drop shadows**.

If you want the drop shadow to be the same color as the input image, enable **inherit input color**. Otherwise, use the **color** controls to select a color for the drop shadow.

Use **opacity** to adjust the opacity of the drop shadow relative to the input's alpha channel.

To adjust the direction of the shadow, use the **angle** slider. A value of 0 or 360 equals left direction.
**Add Shadows to Text**

**Angle** set to 0 (or 360).

**Angle** set to 225.

Use **distance** to set the distance of the shadow from the input content.

**Distance** set to 15.

**Distance** set to 45.

If you want to blur the shadow, increase the **softness** value.

**Softness** set to 0.

**Softness** set to 20.

If necessary, set the **shrink/expand** slider to a negative value to erode the shadow or a positive value to dilate it.
Adding Shadows to Text | Animating Groups

**Shrink/expand** set to -5.  
**Shrink/expand** set to 5.
Adding a Drop Shadow

The DropShadow gizmo allows you to create a drop shadow for any input image that has an alpha channel with values greater than 0. To add a drop shadow an image:

1. Select Filter > DropShadow to insert a DropShadow gizmo after your image Read node.
2. Connect your background image to the DropShadow gizmo's bg input. Your image and drop shadow are merged over the background image.
3. In the DropShadow gizmo properties, check enable drop shadow to enable the drop shadow effect.

4. If you want the drop shadow to be the same color as the input image, enable inherit input color. Otherwise, use the color controls to select a color for the drop shadow.

5. Use opacity to adjust the opacity of the drop shadow relative to the input's alpha channel.

---

 vendaux user guide

 808
6. To adjust the direction of the shadow, use the angle slider. A value of 0 or 360 equals left direction.

![Image of angle slider set to 0 and 225 degrees]

**Angle** set to 0 (or 360) **Angle** set to 225

7. Use distance to set the distance of the shadow from the input content.

![Image of distance slider set to 5 and 15 units]

**Distance** set to 5 **Distance** set to 15

8. If you want to blur the shadow, increase the softness value.

![Image of softness slider set to 0 and 10 units]

**Softness** set to 0 **Softness** set to 10

9. If necessary, set the shrink/expand slider to a negative value to erode the shadow or a positive value to dilate it.

![Image of shrink/expand slider set to 0 and 3.5 units]

**Shrink/expand** set to 0 **Shrink/expand** set to 3.5
10. Use the input [operation] bg dropdown menu to select how you want to combine the input and bg inputs.
Analyzing and Matching Clips

This chapter concentrates on the CurveTool node. The node analyzes an aspect of a frame sequence and creates an animation curve based on the analysis. You can then use the curve data to drive effects elsewhere. For instance, you can add matching flicker to a CG render.

Introduction

You can use the CurveTool node to analyze four different aspects of your frame sequence, depending on which curve type you select in the node controls:

- **AutoCrop** finds black regions (or any color you pick) around the edges of the frame sequence and tracks their size and position over time. This is useful for running a Crop node to remove unnecessary outer pixels and speed up the calculation. For more information, see Cropping Black Edges.

- **Avg Intensities** is useful for obtaining the average pixel values in a frame sequence and then matching that intensity elsewhere. It takes the first value in the frame range and the next value selected, adds them together and divides by two, returning the average between the two. You might want to use it to match the background plate's fire flicker in the smoke in the foreground plate, for example. For more information, see Analyzing the Intensity of a Frame Sequence and Removing Flicker.

- **Exposure Difference** analyzes the exposure changes in the frame sequence. It takes the first value in the frame range and the next value selected, and returns the difference between the two. You can use the results to match the same exposure elsewhere. For more information, see Analyzing Exposure Differences.

- **Max Luma Pixel** tracks the brightest and dimmest pixels in the frame sequence. This can be useful in the following case, for example. Let's say you have a night-time sequence depicting a person moving inside a dark house holding a flashlight, and want to add lens flare on the moving flashlight. Knowing where the brightest pixel is located over time allows you to match-move the lens flare and position it correctly without having to manually animate it. For more information, see Tracking the Brightest and Darkest Pixels.
Tip: If you are familiar with Shake, you may have used the PixelAnalyzer node. The CurveTool node is the Nuke equivalent of PixelAnalyzer.
Cropping Black Edges

You can crop black edges (or any color you choose) from your footage to eliminate unnecessary computation:

1. Select `Image > CurveTool` to insert a CurveTool node after the image sequence you want to analyze.
2. Make sure a Viewer is connected to the CurveTool node.
3. In the CurveTool controls, select `AutoCrop` from the `Curve Type` dropdown menu.
4. Using the `color` parameters, select the color you want to track.
5. To control how far the color can deviate from the selected color and still be cropped off, use the `Intensity Range` slider.
6. From the `channels` dropdown menu and checkboxes, select the channels you want to analyze.
7. If you want to analyze an area in the frames rather than entire frames, define a region of interest either by dragging the edges of the frames to a new position in the Viewer, or by defining the area using parameters labeled `ROI`.
8. Click `Go!` to analyze the frames. This opens the `Frames to Execute` dialog.
9. In the dialog, define the frames to analyze. Enter the first frame, followed by a comma and the last frame. Click `OK`. Nuke starts analyzing the frame sequence.
10. You’ll find the results of the analysis on the `AutoCropData` tab where the parameter values have turned blue to indicate they are animated over time. To see the animation curve, right-click on a parameter field and select `Curve editor`.

Once Nuke has created the animation curve, you can copy the animation or any of its values into a Crop node, for example, to match the analyzed crop area there. `Ctrl/Cmd`+click on the animation button and drag and drop it to another parameter to create an expression linking the two.
Analyzing the Intensity of a Frame Sequence

You can analyze your footage to find out the average intensity values in it:

1. Select **Image > CurveTool** to add a CurveTool node in an appropriate place after the image sequence you want to analyze and match.
2. Connect a Viewer to the CurveTool.
3. In the node’s controls, select **Avg Intensities** from the **Curve Type** dropdown menu.
4. Select the channels you want to analyze from the **channels** dropdown menu and checkboxes.
5. By default, the region of interest that is to be analyzed covers the entire frame. If you want to analyze a smaller area, resize and reposition the region of interest in the Viewer by dragging its edges to a new position. You can also resize the region of interest using the **ROI** parameters in the properties panel.
6. In the **# frames for base average field**, enter the range of frames that each frame being analyzed is compared against. The frames are compared onwards from each frame analyzed. Thus, a value of 1 would compare each frame to the frame following it, whereas a value of 5 would compare each frame to the following 5 frames.
   The higher frame range you use, the more accurate and time-consuming the calculation becomes. However, a high frame range is not always needed. For analyzing and matching fire flicker, you’d probably want to go frame by frame, whereas removing flicker would require a wider frame range to ensure a good average is obtained as the result.
7. To analyze the sequence, click **Go!**. This opens the **Frames to execute** dialog.
8. In the dialog, specify the frame range you want to analyze and match. Enter the first frame, followed by a comma and the last frame. Click **OK**. Nuke now analyzes the frame sequence.
9. Move to the **IntensityData** tab where you’ll find the results of the analysis. You’ll notice that the parameter input fields have turned blue. This indicates that they are animated. To see the animation curve, right-click on the values and select **Curve editor**.

Once Nuke has created the animation curve, you can copy the animation or any of its values into a color correction node, for example, to match the analyzed intensity there. **Ctrl/Cmd**+click on the animation button and drag and drop it to another parameter to create an expression linking the two.
Removing Flicker

You can also use the CurveTool to stabilize flickering in your footage. To do this:

1. Connect a CurveTool node to your footage.
2. In the Curve Type dropdown, select Avg Intensities.
3. If necessary, select the channels you want analyzed in the channels controls, and adjust the region of interest (ROI) box in the Viewer to cover your analysis area.
4. When you're ready, click Go! and specify a frame range in the Frames to Execute dialog that opens. CurveTool analyzes your footage, and the resulting values appear on the IntensityData tab.
5. Create a Grade node (Color > Grade) and connect it to your footage.
6. From the CurveTool's IntensityData tab, Ctrl/Cmd+drag the analysis result to the multiply field in the Grade node controls.
7. Right-click on the multiply field, and select Edit expression, or press the equals (=) button with the field selected.
8. In the dialog that appears, add 1/ in front of the Expression field entry. This inverts the brightness values detected by the CurveTool node, and enables the Grade tool to stabilize brightness changes causing flickering in your footage.
Analyzing Exposure Differences

You can analyze the differences in exposure in your frame sequence:

1. Select **Image > CurveTool** to add a CurveTool node after the image sequence you want to analyze.
2. Add a Viewer after the CurveTool.
3. Under **Curve Type**, select **Exposure Difference**.
4. From the **channels** dropdown menu and checkboxes, select the channels you want to analyze.
5. If you want to analyze an area in the frame rather than the entire frame, define a region of interest either by dragging the edges of the frame box to a new position in the Viewer, or by defining the area using parameters labeled **ROI**.
6. To analyze the sequence, click **Go!**. The **Frames to Execute** dialog opens.
7. Specify the frame range you want to analyze. Enter the first frame, followed by a comma and the last frame. Click **OK**. Nuke now performs the analysis.
8. You can find the results of the analysis on the **IntensityData** tab where the parameter input fields have turned blue to indicate they are animated. To see the animation curve, right-click on one of the input fields and select **Curve editor**....

Once Nuke has created the animation curve, you can copy the animation or any of its values into a color correction node, for example, to match the analyzed exposure there. **Ctrl/Cmd**+click on the animation button and drag and drop it to another parameter to create an expression linking the two.
Tracking the Brightest and Darkest Pixels

You can track the brightest and the darkest pixels in your frame sequence:

1. Select **Image > CurveTool** to add a CurveTool node after the image sequence you want to analyze.
2. Connect a Viewer to the CurveTool.
3. From the **Curve Type** dropdown menu, select **Max Luma Pixel**.
4. Click **Go!** to analyze the frame sequence. This opens the **Frames to Execute** dialog.
5. Define the frame range you want to analyze. Enter the first frame, followed by a comma and the last frame. Then, click **OK**. Nuke analyzes the frame sequence, tracking both the position and the values of the brightest and darkest pixels.
6. You can find the results of the analysis on the **MaxLumaData** tab. You'll notice that the input fields have turned blue to indicate that they are animated over time. To see the animation curve, right-click on an input field and select **Curve editor**.

Once Nuke has created the animation curve, you can copy the animation or any of its values into another node to match that node's effect with the brightest or darkest pixels in the frame sequence. **Ctrl/Cmd**+click on the animation button and drag and drop it to another parameter to create an expression linking the two.
3D Compositing

Nuke’s 3D workspace allows you to set up a 3D composite for camera moves, set replacement, and other applications where you need to simulate a "real" dimensional environment.

Overview

This chapter explains how to set up a 3D scene in Nuke, and how to add objects and cameras in the 3D workspace. You’ll also see how to texture objects, transform objects and cameras, and render out scenes for use in other areas of your script.

Although the 3D workspace has many potential uses, you’re most likely to use it - at least initially - to create pan-and-tile scenes. These are scenes with 2D image planes arranged into a curved shape, and then rendered out through an animated camera to give the illusion of a seamless environment.

The 3D objects in Nuke appear as round shapes to differentiate them from objects that perform 2D operations. As shown above, you can mix 2D and 3D objects together in the node tree. For example, you can texture a 3D object with a 2D clip, or take the rendered output from a 3D scene and use it as a 2D background.
Script with 2D and 3D operators
Setting Up a Scene

Each 3D scene includes the following objects: a Scene node, a Camera node, one or more geometry nodes (i.e., card, sphere, obj), and a ScanlineRender node. Examples of 3D scenes are shown in Overview and the figure below. In the example, the Scene node receives the output from the geometry nodes and sends the composite of those objects to the ScanlineRender node, where the output is converted back to 2D.

Core nodes for a 3D composite.

Your script may contain multiple Scene nodes, cameras, and 3D render nodes. All 3D objects loaded in the Properties Bin appear in the 3D Viewer, regardless of whether they are connected to the same Scene node.

The Scene Node

Regardless of its location in your script, the Scene node is the highest-level node in the scene hierarchy because it references all the elements in a 3D workspace - all the geometric objects, cameras, and materials.

To Add a Scene Node

Select 3D > Scene from the Toolbar.
The ScanlineRender Node

Every Scene node in a script should be connected to a ScanlineRender node, which tells Nuke to render the results of the scene. The ScanlineRender node also allows you to toggle between a 2D and 3D view of the scene.

To Add a ScanlineRender Node

1. Select the Scene node.
2. Select 3D > ScanlineRender from the Toolbar.
3. Connect the obj/scn input to a Scene or geometry node.
4. Connect the cam input to the main camera.
5. Connect the optional bg input to composite a background image into the scene.
6. Press Ctrl+I (Cmd+I on a Mac) to open a new Viewer to display the output of the ScanlineRender node.

When an image is connected to the bg input, its resolution becomes the output resolution for the ScanlineRender node.

The Camera Node

Cameras may be connected to either the Scene node or the ScanlineRender node. The camera connected to the ScanlineRender node is the camera used for rendering.

1. Select 3D > Camera to insert a camera node.
2. Drag an output connector from the Camera node to a Scene node or connect the Camera node to a ScanlineRender node’s cam input.

When connecting cameras for the 3D scene, the camera you want to use for rendering should be connected to the ScanlineRender node, like this:
Any additional cameras should be connected to the Scene node. When you have multiple cameras associated with a 3D scene, you can switch between them by selecting the viewing camera from the dropdown menu at the top of the Viewer. See the next section, Using the 3D Viewer, for more information.
Using the 3D Viewer

When you have a 3D setup in your script, any Viewer window can toggle between the 2D and 3D display modes. The 2D mode shows the result of a rendered scene, the 3D mode shows the perspective from cameras in the scene.

![Image of 3D Viewer]

The 3D Viewer.

When you do not have a Camera node in your script, the 3D Viewer uses default views (see the figure below for the list of options). These views are similar to different cameras that you can look though, but they don't appear as objects that you can manipulate in the scene.

Switching to the 3D Viewer

Open a Viewer and press Tab or V to toggle between the 2D and 3D modes - or select the view you want from the dropdown menu at the top right corner of the Viewer window.
Switching to the 3D Viewer.

The “built-in” views give you different perspectives on your 3D scene. You can quickly switch between the views by pressing the keyboard shortcuts for right view (X), left view (Shift+X), top view (C), bottom (Shift+C), front (Z), back (Shift+Z), and three-quarter perspective (V).

To Navigate in the 3D Viewer

- Dolly: Press Alt and middle-mouse-button drag.
- Pan: Press Alt and left-mouse-button drag.
- Tilt: Press Ctrl/Cmd and left-mouse-button drag.
- Spin: Press Ctrl/Cmd and left-mouse-button drag.
- Roll: Press Ctrl/Cmd+Shift and left-mouse-button drag.
- Look through camera: Select a camera object, press H.
- Fit the scene: Press F to fit the entire 3D scene within the Viewer.

To Change the 3D Viewer Display Properties

1. Open the Preferences dialog (Shift+S), and select Panels > Viewer Handles.
2. Make the desired changes to the 3D bg and fg colors.
3. From the 3D control type dropdown menu, select the navigation control scheme you want to use (Nuke, Maya, Houdini, Lightwave, or Modo).
4. Click OK.

Note: The 3D control type also affects the mouse button assignments for panning and zooming in the node graph and 2D Viewers.

To Look Through a Camera
1. Press V to make sure you are looking through the 3D perspective view, and not one of the orthographic views.
2. From the 3D Viewer window, select the camera from the dropdown menu in the top right corner.
To Lock the 3D Camera View

You can choose to lock the 3D view to the selected camera or light. You can toggle between the unlocked and locked modes by clicking the 3D view lock button, or by pressing Ctrl/Cmd+L.

- **unlocked**: to freely move in the 3D view without restrictions. The 3D view lock button is gray.
- **locked**: to lock your movement to the camera or light you’ve selected in the dropdown menu on the right side of the 3D view lock button. The 3D view lock button is red.

To Use the Interactive 3D Camera View Mode

With the interactive 3D camera view mode you can change the camera or light values according to your movement in the Viewer. You can activate the interactive mode by Ctrl/Cmd+clicking the 3D view lock button. When the interactive mode is on, the 3D view lock button turns green. In order to activate the interactive mode, you need to have a Camera or a Light node selected in the dropdown on the right side of the 3D view lock button.

When the interactive mode is on, you can use the plus (+) and the minus (-) keys to change the translate values of the camera or light you’ve selected. When the interactive mode is off, these keys zoom your view in and out.
3D Scene Geometry

There are several options for inserting 3D geometry into your scenes. You can:

• create primitive shapes: cards, cubes, cylinders, and spheres. See Using Built-in Primitive Geometry.
• import models and point clouds created in other 3D applications and exported into OBJ (Wavefront), FBX, or Alembic files. See Importing Geometry and Point Clouds from Other Applications.
• create your own geometry and point clouds from scratch. See Creating Point Clouds and Geometry from Scratch on page 841

Once you've inserted a 3D geometry object into your scene, you can:

• adjust how the object is displayed in the Viewer. See Object Display Properties on page 848.
• select entire objects, as well as vertices or faces on objects. See 3D Selection Tools on page 850.
• merge objects together. See Merging Objects on page 855.
• modify object shapes. See Modifying Object Shapes on page 856.
Using Built-in Primitive Geometry

This section describes the primitive shapes built in with Nuke: cards, cubes, cylinders, and spheres. You can use them as the building blocks for other, more complex shapes, or place them in the background of your 3D scene where extra detail isn't visible. If you modify them with transforms, they can also be useful as foreground objects.

Working with Cards

A card is the simplest type of object you can add to a scene (and probably the type you will use most often). It's merely a plane onto which you can map a texture - typically a clip you are using as part of a pan-and-tile setup.

A card object.

A card object may be deformed as a bilinear or bicubic object with controls contained in the card’s parameters. You can also insert other 3D nodes, such as ProceduralNoise or RadialDistort, to change the card geometry.

Card nodes have extended bicubics (bicubics with more control points). They allow you to subdivide a card, giving you finer control for warping an area. You can subdivide the card into an evenly spaced grid or pick a location to add a row, column, or both.

To Add a Card Object:

1. Click **3D > Geometry > Card** to insert a Card node.
2. Drag the Card node’s **img** pipe to the Read node that has the image you want to apply to the card.
3. Connect the Card node to the appropriate Scene node to add it to the 3D scene.
4. Use the card object’s transform controls to manipulate the position, scale, and rotation of the card in 3D space. For more information, see Transforming from the Node Properties Panel.

**Deforming Card Objects**

The Deform tab in the Card Properties panel lets you convert the card into a mesh surface that may be pulled and reshaped.

A *bicubic deformation* offers the greatest degree of surface elasticity. You can add any number of control points on the card and translate these points and their tangents in any direction. The control point tangents exert a magnetic-like influence over the objects surface.

![Image of card with control points]

The Card node can have any number of control points you can translate.

To deform a Card object:
1. Double-click the Card node to open its controls.
2. Go to the Deform tab, and select the mesh type for the deformation: **bilinear** or **bicubic**.

![Image of Deform tab]

3. By default, the card has three control points on the x axis, and three on the y axis. To add more control points, do any of the following:
   - Enter new values in the x/y points fields and click the **new shape** button. For example, to create a shape with 4 points on the x axis and 6 on the y axis, change the x points value to 4 and the y points value to 6, and click **new shape**.
• To evenly subdivide the current shape in the x or y directions, click the x subdivide or y subdivide buttons. This adds one control point between every existing control point in the selected direction. The x/y points fields are also updated to reflect the current number of control points.
• To add one row or column of control points, adjust the u or v slider. The u slider specifies the position of new columns, and the v slider the position of rows. In the Viewer, a yellow cross marker indicates the position of the new row or column. You can also move the cross marker by dragging it to a new position in the Viewer. The u and v sliders’ values are updated as you move the marker. When you are happy with the position, click the uv subdivide button. A row or column is added in the position you specified. Clicking the button again has no effect, because there is already a subdivision at the specified position.

4. If you selected bicubic under type, you can adjust the way control point tangents behave when you are making your changes to the card. Do any of the following:
   • To have the original tangents adjusted to create a more uniform subdivision when you are using x subdivide, y subdivide, or uv subdivide, check uniform subdivision. If you do not check this, Nuke maintains the original tangents.
   • You can move the tangents in the Viewer by clicking and dragging. If you want to move a tangent together with the opposite tangent so that the two tangents form a continuous line, check smooth tangent. To break the tangent from the opposite tangent and move the tangent alone, uncheck smooth tangent.
   • To change the length of the opposite tangent to always match the length of the tangent you are moving, check mirror tangent. If you do not check this, the opposite tangent length is not changed.

5. Drag the points displayed in the mesh to deform the card.

To translate the control points and tangents:
1. If necessary, double-click on the Card node to display its controls, and go to the Deform tab.
2. Only the controls for the selected point are displayed in the bottom of the Card properties panel. To translate another point, you can select a new point in the Viewer or use the arrow buttons in the bottom of the Card controls to move to the next control point.

3. To translate the control points and their tangents:
• Increment or decrement the numbered x, y, and z fields. For each control point, the controls for translating the point itself are shown on top of the controls for translating the tangents.
• Or drag on any control point or tangent to translate it relative to the current angle of view.

Working with Cubes

A cube is the familiar six-sided polyhedron. You can transform any of its sides (and, of course, texture it with clips).

To Add a Cube
1. Click 3D > Geometry > Cube to insert a Cube node.
2. Drag the Cube node's img pipe to the Read node containing the clip you want to use as a texture.
3. Drag one of the Scene node's numbered pipes to the Cube node to place the cube in the scene.
4. Use the cube object's transform controls to manipulate the position, scale, and rotation of the cube in 3D space. For more information, see Transforming from the Node Properties Panel.
5. Translate any of the cube's sides to alter its shape.

To Translate a Cube's Sides
1. If necessary, double click on the Cube node to display its parameters (and thereby select the object in the scene).
2. Increment or decrement the cube fields. (Assuming a positive z view of the object, x refers to the left side; y, the bottom side; n, the back side; r, the right side; t, the top side; and f, the front side.) Or drag on any side order to translate it relative to the current angle of view.
Working with Cylinders

A cylinder is an object with two identical flat ends that are circular or elliptical and one curved side. You can control its geometric resolution, or face count (and, of course, texture it with images).

To Add a Cylinder

1. Click 3D > Geometry > Cylinder to insert a Cylinder node.
2. Drag the Cylinder node's img pipe to Read node containing the clip you want to use as a texture.
3. Drag one of the Scene node's numbered pipes to the Cylinder node to place the cylinder in the scene.
4. Use the cylinder object's transform controls to manipulate the position, scale, and rotation of the cylinder in 3D space. For more information, see Transforming from the Node Properties Panel.

Adjusting Geometric Resolution

By default, a cylinder has 30 rows and 30 columns. You can, however, increase or decrease either number as appropriate. For example, the figure below shows a cylinder whose geometric resolution has been decreased to 10 rows and 3 columns.
A low-resolution cylinder.

To adjust a Cylinder’s geometric resolution:
1. If necessary, double click on the Cylinder node to display its parameters (and thereby select the object in the scene).
2. Increment or decrement rows field to adjust the number of latitudinal divisions on the cylinder.
3. Increment or decrement columns field to adjust the number of longitudinal divisions on the cylinder.

Working with Spheres

A sphere is the familiar globe-shaped polyhedron. You can control its geometric resolution, or face count (and, of course, texture it with images).

To Add a Sphere
1. Click 3D > Geometry > Sphere to insert a Sphere node.
2. Drag the Sphere node’s img pipe to Read node containing the clip you want to use as a texture.
3. Drag one of the Scene node’s numbered pipes to the Sphere node to place the sphere in the scene.

4. Use the sphere object’s transform controls to manipulate the position, scale, and rotation of the sphere in 3D space. For more information, see Transforming from the Node Properties Panel.

**Adjusting Geometric Resolution**

By default, a sphere has 30 rows and 30 columns. You can, however, increase or decrease either number as appropriate. For example, the figure below shows a sphere whose geometric resolution has been decreased to 2 rows and 4 columns (making it, in effect, an octahedron).

![An octahedron generated with a low-resolution sphere.](image)

To adjust a Sphere's geometric resolution:

1. If necessary, double click on the Sphere node to display its parameters (and thereby select the object in the scene).

2. Increment or decrement **rows** field to adjust the number of latitudinal divisions on the sphere.

3. Increment or decrement **columns** field to adjust the number of longitudinal divisions on the sphere.
Importing Geometry and Point Clouds from Other Applications

Sometimes, you may need to import files or objects created in 3D applications, such as Maya or Boujou. Depending on what you want to import and from where, there are different ways of doing the import:

• To import geometry from OBJ (.obj) files, see Importing Geometry from OBJ Files.
• To import geometry or point clouds from FBX (.fbx) files, see Importing Geometry and Point Clouds from FBX Files. FBX is a standard file format many applications can export to, and .fbx files contain 3D scenes from which you can import cameras, lights, transforms, meshes, and point clouds into Nuke.
• To import geometry or point clouds from Alembic (.abc) files, see Importing Geometry and Point Clouds from Alembic Files. You can export to Alembic from most popular 3D applications.

Importing Geometry from OBJ Files

You can import into a Nuke scene 3D objects from other software programs that have been saved out in the .obj (Wavefront) format. You cannot manipulate .obj objects at the vertex level from inside Nuke, but you can texture and transform them.

An imported OBJ object.

**To import an OBJ object**

1. Click 3D > Geometry > ReadGeo to insert a ReadGeo node.
2. In the ReadGeo parameters, click the file field's folder icon. The file navigation dialog appears.

4. Drag the ReadGeo node's img pipe to the Read node containing the clip you want to use as a texture.

5. Drag one of the Scene node's numbered pipes to the ReadGeo node to place the OBJ object in the scene.

Importing Geometry and Point Clouds from FBX Files

FBX is a standard 3D file format that gives you access to 3D scenes created in other applications supporting the same format. What you generally have in an .fbx file is an entire 3D scene containing cameras, lights, meshes, non-uniform rational B-spline (NURBS) curves, transformation, materials, and so on. From this scene, you can extract cameras, lights, transforms, and meshes into Nuke. This way, you can, for example, create a mesh in Maya, export it in a .fbx file, and use the same mesh again in Nuke.

**Note:** For the FBX SDK version used in Nuke, see Third-Party Library Versions.

**Tip:** If you have trouble with .fbx files, it may be because they were written with an older version of FBX. If they load very slowly, it is also possible that they are ASCII rather than binary. To get around these problems, you can use the FBX converter on the Autodesk website (http://usa.autodesk.com/fbx/download/). It converts between various different formats, including older FBX versions, ASCII, and binary, and is available on Windows, Mac, and Linux.

Importing Meshes from FBX Files

The ReadGeo node lets you import meshes (or NURBS curves/patch surfaces converted to meshes) from FBX files. Using one ReadGeo node, you can read in a single mesh or all the meshes in a .fbx file.

The mesh's vertices, normals, UVs, and vertex colors are read on a per frame basis or at frame 0. If there are any shape or cluster deformers, they are applied to the vertices. Materials or textures are not read in.

To import a mesh from an .fbx file:

1. Select 3D > Geometry > ReadGeo to insert a ReadGeo node into your script.
2. In the ReadGeo controls, click the folder icon next to the file field and navigate to the .fbx file that contains the mesh you want to import. Click Open.

3. From the animation stack dropdown menu, select the take you want to use. FBX files support multiple takes. Usually, one of them is a default take that contains no animation.

4. From the node name dropdown menu, select the mesh you want to import from the .fbx file.

5. To adjust the frame rate used to sample the animation curves, enter a new value (frames per second) in the frame rate field. The frame rate you enter is only used if you check use frame rate. Otherwise, the frame rate from the .fbx file is used.

6. If you want to import all the meshes in the .fbx file rather than just one, check all objects. This overrides whatever you have selected under node name. If the objects are animated, check read on each frame. This bakes each object's transform into the mesh points and preserves the animation.

7. If you want to modify the transform properties imported from the .fbx file, uncheck read transform from file and make the necessary modifications. As long as read transform from file is unchecked, your changes are kept.

8. To reload the transform properties from the .fbx file, click the Reload button.

**Importing Point Clouds from FBX Files**

The ReadGeo node also lets you import point clouds from FBX files.

To import a point cloud from an .fbx file:

1. Select 3D > Geometry > ReadGeo to insert a ReadGeo node into your script.

2. In the ReadGeo controls, click the folder icon next to the file field and navigate to the .fbx file that contains the point cloud you want to import. Click Open.

3. From the animation stack dropdown menu, select the take you want to use. FBX files support multiple takes. Usually, one of them is a default take that contains no animation.

4. In the objecttype dropdown, select PointCloud.

5. To adjust the frame rate used to sample the animation curves, enter a new value (frames per second) in the frame rate field. The frame rate you enter is only used if you check use frame rate. Otherwise, the frame rate from the .fbx file is used.

6. If you want to modify the transform properties imported from the .fbx file, uncheck read transform from file and make the necessary modifications. As long as read transform from file is unchecked, your changes are kept.

7. To reload the transform properties from the .fbx file, click the Reload button.
Importing Geometry and Point Clouds from Alembic Files

You can import meshes (or NURBS curves/patch surfaces converted to meshes) and point clouds from Alembic files (.abc file format) into a Nuke scene. During the import, Nuke allows you to control which nodes in the Alembic scene get loaded by using an import dialog. If there is only one item in the Alembic file, it loads automatically. Below is an example of an imported octopus Alembic file, courtesy of Sony Pictures Imageworks.

For more information on Alembic, see http://code.google.com/p/alembic/

Tip: In addition to meshes and point clouds, you can also import cameras and transforms from Alembic files.

To learn how to export files in the Alembic (.abc) format, refer to Exporting Geometry, Cameras, Lights, Axes, or Point Clouds for more information.

To Import Meshes and Point Clouds from an Alembic File

1. Click Image > Read or press R on the Node Graph. The Read File(s) dialog displays.
2. Select the Alembic file you want to import from the file location, then click Open. The Alembic import dialog displays. By default, all items are selected in the Scene Graph when the import dialog is opened, as shown below:
Selected parent items are indicated by a yellow circle, and selected child items by a yellow bar (these turn orange when you select them in the list). Unselected items do not have an indicator next to them.

3. To import specific items, you must first deselect the root item by clicking on the yellow circle. This de-selects the root and any child items. Then, select specific items in the scenegraph by clicking on the blank space where the circles had been, as shown below:

![Screenshot of scenegraph]

Alternatively, you can right-click on an item and select:

- **Select as parent** - to select this item and make it a parent to other items. This allows you to create a separate Nuke node for this item (and any child items underneath it) in the next step.
- **Select as child** - to select this item and make it a child to the nearest parent item up the tree.
- **Deselect** - to deselect this item (that is, not import it from the scene).

You can also select multiple items by pressing Ctrl/Cmd or Shift while clicking them.

4. Do one of the following:

- Click **Create all-in-one node** to create one Nuke node for everything that's selected, regardless of whether the items are selected as parent or child.
- Click **Create parents as separate nodes** to create one Nuke node for each parent item (yellow circle) in the tree. This node contains all the child items (yellow bars) under the parent.

**Tip:** If you always want .abc files to import all-in-one without displaying the import scenegraph, enable **Preferences > Behaviors > File Handling > always load abc files as all-in-one.**

Nuke creates ReadGeo, Camera, and Axis nodes as necessary, depending on what you selected to import from the scene.

5. On ReadGeo nodes, you can adjust the following:

- If the objects you imported are animated, make sure **read on each frame** is checked. This preserves the animation. When **read on each frame** is disabled, use **lock frame** to select the frame the object is loaded at.
- If you don't want to render motion blur, disable **sub frame** for faster UI interactions.
- In the **frame rate** field, define a frame rate (frames per second) to sample the animation curves.
- Use **render points as** to determine how point primitives are rendered: either as Nuke point clouds or Nuke particles.
• Enable **use geometry colors** to apply geometry color attributes read from *.abc* files and apply them to the Nuke geometry.

**Note:** When disabled, this control can cause differences in rendered output when compared to previous versions of Nuke. If this occurs, enable **use geometry colors** in the ReadGeo properties panel.

• Any mesh items imported are listed on the **Scenegraph** tab of the ReadGeo node. To display all items that exist in the Alembic file in the list, check **view entire scenegraph**. This allows you to add items to the list or remove them from it by clicking on the yellow and blank indicators on the right.

**Tip:** To load specific items from an Alembic file, you can also create a ReadGeo, Camera, or Axis node, check **read from file**, and click the folder icon on the **File** tab to browse to an *.abc* file.
Creating Point Clouds and Geometry from Scratch

In addition to using the built-in primitives and imported geometry, Nuke also allows you to create dense point clouds and geometry from scratch.

This section focuses on creating point clouds using the DepthToPosition, PositionToPoints, and DepthToPoints nodes available in Nuke. However, if you have a NukeX license, you can also use the CameraTracker and PointCloudGenerator nodes to create point clouds or the ModelBuilder node to build 3D models for 2D shots. See Creating Dense Point Clouds and Using ModelBuilder for more details.
Creating a Position Pass Using the DepthToPosition Node

DepthToPosition takes depth data contained in an image file and the camera data to create a 2D position (xyz) pass. This pass is created by projecting the depth through the camera and recording the xyz positions of each projected point. The DepthToPosition node can be used together with the PositionToPoints node to create a point cloud, similar to the effect achieved with the DepthToPoints node.

To Create a Position Pass using DepthToPosition

1. From the 3D > Geometry menu, select DepthToPosition to add the node to your script.
2. Read in an image with a depth pass and connect it to the image input of the node.
3. From the 3D menu, select a Camera and connect it to the camera input of the DepthToPosition node.
4. In the DepthToPosition tab of the DepthToPosition node, select the depth channel from the depth dropdown menu. You should now see a position pass in the Viewer.
5. If you want to change the output channel, select the channel you would like, or create your own, from the **output** dropdown menu.

6. Set the **far** control value to specify at what distance from the camera the depth values are ignored. This prevents large depth values from creating unwanted banding caused by the precision of the floating math calculations.

The DepthToPosition node allows you to create a position pass from an image's depth pass, and feed that into PositionToPoints to ultimately create a point cloud. In fact the three nodes, DepthToPosition, PositionToPoints, and DepthToPoints can be used together, as DepthToPoints is a gizmo that contains the DepthToPosition and PositionToPoints nodes.

To learn more about generating depth maps, please refer to *Generating Depth Maps*. 

---

**To Create a Position Pass using**

---

**Creating a Position Pass Using the DepthToPosition Node**

---

**NUKE USER GUIDE**
Creating a Dense Point Cloud Using the PositionToPoints Node

PositionToPoints takes position data contained in an image file (rendered from a 3D application) and recreates the image as a dense 3D point cloud in Nuke. The x, y, and z vertices in the position channel are used to define point positions in 3D space, the size and number of which can be adjusted using the point size and point detail controls.

To Create a Point Cloud Using PositionToPoints

1. From the 3D > Geometry menu, select PositionToPoints to add the node to your script.
2. Read in a position pass and connect it to the node.
   - If you have multiple images containing separate position and normal information, connect them to the pos and norm inputs, respectively.
   - Note that the pos and norm inputs only appear once you've connected the unnamed input.

   ![Diagram](image)

   The normals are contained in the source image, while a second image contains the position pass.

3. Select the position channel from the surface point dropdown menu. If your image contains a normal channel, select this from the surface normal dropdown menu. You should now see the point cloud in the 3D Viewer.
   - If you supplied the node with separate position and normal information via the pos and norm inputs, the surface point and surface normal dropdowns are disabled.
4. To change the number of points, adjust the point detail slider. A value of 0 means no points are displayed. A value of 1 displays all available points.
5. To change the size of the points, adjust the point size slider.
Point cloud created from the PositionToPoints node.
Creating a Point Cloud Using the DepthToPoints Node

DepthToPoints is a gizmo containing the DepthToPosition and PositionToPoints nodes. It can be used to generate a 3D point cloud from a depth pass and 3D camera. DepthToPoints takes the depth data and color information contained in an image file and recreates the image as a 3D point cloud.

To Create a Point Cloud Using DepthToPoints

1. From the 3D > Geometry menu, select DepthToPoints to add the gizmo to your script.
2. Read in an image with a depth pass and connect it to the image input of the gizmo.
3. From the 3D menu, select a Camera and connect it to the camera input of DepthToPoints.
4. In the DepthToPoints User tab, select the depth channel from the depth dropdown menu. If your channel contains a normal channel, select this from the surface normal dropdown menu, or connect an image containing normal data to the norm input of the node. You should now see the point cloud in the 3D Viewer.
5. To change the number of points, adjust the point detail slider. A value of 0 means no points are displayed. A value of 1 displays all available points.
6. To change the size of the points, adjust the point size slider.
Tip: If you are having trouble viewing the image and point cloud, ensure that your image's alpha is not set to black.
Object Display Properties

You can adjust the display characteristics of all geometric objects in a scene. These settings don’t affect the render output of the scene; these are for display purposes only in the 3D Viewer.

To Edit an Object’s Display Attributes

1. Double click on the object’s node to display its parameters.
2. From the display dropdown menu, select the display type that you want for the object.

These are how each of the display options appear:

- **wireframe** displays only the outlines of the object’s geometry.

- **solid** displays all geometry with a solid color.

- **solid+wireframe** displays the geometry as solid color with the object’s geometry outlines.
• **textured** displays only the surface texture.

• **textured+wireframe** displays the wireframe plus the surface texture.
3D Selection Tools

Using the 3D selection tools in the top right corner of the Viewer, you can select nodes, vertices and faces on a 3D object. Additionally, you can select individual 3D objects in the 3D view, which is handy when you have more than one object in a node.

Selection Modes

Nuke features three selection modes, Rectangle, Ellipse, and Lasso. Selection modes work the same in all Viewer contexts, whether you're selecting vertices, faces, or objects. The selection mode dropdown is located above the Viewer at the top-right.

Each mode uses the same selection modifiers:

- **Shift** - additive selection. Holding **Shift** and selecting vertices, faces, or objects adds the selection to any existing selection.
• ***Alt + Shift*** - subtractive selection. Holding **Alt + Shift** and selecting vertices, faces, or objects removes the selection from any existing selection.

### Selecting Nodes

You can select nodes with the Node selection tool 🔄. This is the default selection tool, and corresponds to the familiar way of selecting 3D object nodes in the Viewer.

### Selecting Vertices on a 3D Object

You can select vertices on a 3D object using the Vertex selection tool. To save selections into your script, you need to use the GeoSelect node. You can find the 3D selection tools in the top right corner of the Viewer.
Note: The selection mode is set by the Viewer - changing geometry nodes translates your selections to the new object.

You can also use the Vertex selection tool to edit geometry using the EditGeo node. See Modifying Objects Using the EditGeo Node.

To select vertices in the Viewer

1. Attach a 3D object to the Viewer and activate the Viewer’s 3D mode.
2. Click the **Vertex selection** tool to activate it. It is disabled by default.
3. Select vertices on the object by dragging a marquee over the vertices you want to select.
4. By default, a new selection replaces an existing one, but you can also add and remove from an existing selection:
   - To add to existing selection, hold down **Shift** while selecting.
   - To remove from existing selection, hold down **Shift+Alt** while selecting.
5. You can also include hidden items by selecting the **occlusion test** button. Deselect it if you don’t want to include occluded vertices in your selection. Select if you want to select both visible and occluded vertices.

To Save or Restore Selected Vertices Using the GeoSelect Node

With the default Viewer 3D selection tools you can only make a temporary selection. To save or restore a selection, you can use the same 3D selection tools in the Viewer with the GeoSelect node:

1. Create a GeoSelect node (**3D > Modify > GeoSelect**) and attach a 3D object to its input. The GeoSelect properties panel needs to be open in order for you to use the node for selecting vertices.
2. Click the **Vertex selection** tool from the 3D selection panel.
3. Select vertices on the object by dragging a marquee over the vertices you want to select.
4. By default, a new selection replaces an existing one, but you can also add and remove from existing selection:
   - To add to existing selection, hold down **Shift** while selecting.
   - To remove from existing selection, hold down **Shift+Alt** while selecting.
5. In the GeoSelect properties panel, you can also uncheck the **selectable** box to freeze your selection so it can't be changed. Checking the box again enables selection again and you can continue selecting vertices as normal.

6. With the **display** and **render** controls you can select the display option you want for your object. For more information on these controls, see **Object Display Properties**.

7. When you're happy with your selection, click **save selection** to store the vertices in the GeoSelect node.

8. The vertices are saved by the GeoSelect node and can be restored by clicking **restore selection**.

### Selecting Faces on a 3D Object

You can select faces on a 3D object using the **Face selection** tool 🎨. Each 3D object is divided into polygonal faces by wireframe lines, and with the Face selection tool you can select one or many of these.

You can also use the Face selection tool to edit geometry using the EditGeo node. See **Modifying Objects Using the EditGeo Node**.

1. Click the **Face selection** tool to activate it. It is disabled by default.
2. Select faces on the object by dragging a marquee over the faces you want to select.
3. By default, a new selection replaces an existing one, but you can also add and remove from existing selection:
   - To add to existing selection, hold down Shift while selecting.
   - To remove from existing selection, hold down **Shift+Alt** while selecting.

![Marquee-selecting faces on a cube.](image)
Selecting 3D Objects

When you have more than one 3D object in one node (such as in a case when you’re reading in a .fbx file with multiple objects), you can use the Object selection tool to select them. To select objects, activate the Object selection tool and click on the object you want to select in the 3D view.

Matching Position, Orientation, and Size to 3D Selection

You can match the position, orientation, and size of a 3D object, such as a Card node, to your selected vertices in another object. Do the following:

1. Click the snap dropdown menu on the properties panel of the 3D object you want to move and select Match selection position. The object is now positioned to match the selected vertices.
2. If you select snap > Match selection position, orientation, your 3D object is positioned and aligned according to the 3D vertex selection.
3. If you select snap > Match selection position, orientation, size, your 3D object is positioned, aligned, and scaled according to the selected vertices.
Merging Objects

With the MergeGeo node, you can merge your 3D objects together to process all of them at the same time. For example, after merging your objects, you can use a Transform node to move the objects together, or add an ApplyMaterial node to apply a global material to them (note that this overrides any individual materials applied to the geometry before it was merged).

To Merge Your 3D Objects

1. Select 3D > Modify > MergeGeo to insert a MergeGeo after the 3D objects in your script.
2. Connect the objects you want to merge to the MergeGeo node's inputs.
   
   You can now process all the objects you connected to the MergeGeo node together.
Modifying Object Shapes

Many nodes under the Modify menu let you modify the shape of an object as a whole. Modifying only selected portions of an object is also supported using the EditGeo node.

You can modify 3D objects by selecting vertices or faces in the Viewer using the EditGeo node or by using lookup curves, power functions, images, a Perlin noise function, a distortion function, or trilinear interpolation.

Modifying Objects Using the EditGeo Node

EditGeo allows you to directly modify an object’s vertices or faces, depending on the Viewer selection mode currently active. You can also modify multiple objects simultaneously if the EditGeo is downstream of a MergeGeo node with multiple geometry inputs.

To Modify Objects Using EditGeo

1. Select the 3D object or MergeGeo you want to modify in the Node Graph.
2. Navigate to 3D > Modify > EditGeo to insert an EditGeo node.
3. In the node’s controls, use the display dropdown menu to select how you want to view your object in the Viewer while making changes to it.

   **Tip:** If you don’t have an image or texture associated with your geometry, you may find that enabling the headlamp control in the Viewer properties improves visibility.

   Press S in the Viewer to display the Viewer properties, then enable the 3D > headlamp checkbox.

4. Set the Viewer selection mode using the 3D selection tools in the top right corner of the Viewer:
   - **Vertex selection** - allows you to select individual vertices on geometry, giving you fine control while editing. You can also marquee select multiple vertices or use additive selection by holding down Shift.
   - **Face selection** - allows you to select individual faces on geometry. You can also marquee select multiple faces or use additive selection by holding down Shift.
Note: You can turn off occlusion testing by clicking the disable icon in the Viewer to allow you to select vertices and faces that are hidden from the view point.

For example, selecting faces on the “front” of a cube with occlusion testing disabled also selects faces on the opposite, hidden side of the cube.

See 3D Selection Tools for more information.

5. Set how the axis handles are aligned with your selection using the axisalignment dropdown:
   • object - the position of the xyz axis is determined by the average position of all vertices in the selection. The orientation of the axis is the same as the object's orientation.
   • average normal - the position of the xyz axis is determined by the average position of all vertices in the selection. The orientation of the axis is aligned to the average of the current selection's normals.

In this mode, the z axis always points directly away from the current selection.

Tip: Holding Ctrl/Cmd+Alt allows you to drag the center of the axis across the surface of the geometry, with the orientation set to the nearest face's normal.
6. Make vertex or face selections in the Viewer as required. Hold **Shift** to add to a selection or
**Alt+Shift** to remove selections.

7. When you're satisfied with your selections, drag the axis associated with the selection to a new
position to edit the geometry.

![Image of 3D model with selections and handles]

### To Animate Edits

1. Set the Viewer selection mode using the 3D selection tools in the top right corner of the Viewer:
   - **Vertex selection** - allows you to select individual vertices on geometry, giving you fine control
     while editing. You can also marquee select multiple vertices or use additive selection by holding down
     **Shift**.
   - **Face selection** - allows you to select individual faces on geometry. You can also marquee select
     multiple faces or use additive selection by holding down **Shift**.

2. Make selections in the Viewer and then click ☰ in the properties panel.

3. Scrub the playhead to the required frame and transform the geometry using the handles in the
   Viewer.
   - A keyframe is added automatically on the new frame.

4. Repeat the process to produce the required animation.

   **Tip:** Click ⏩ to remove a keyframe at the current frame.

5. Click **reset geometry** to return the vertices to their original positions.
Modifying Objects Using Lookup Curves

The CrosstalkGeo and LookupGeo nodes offer you direct global control over each of the vertex x, y, and z values respectively. You can, for example, only modify all the y values without touching the x and z values.

You change the different vertex values (x, y, or z) by modifying their associated 2D curves in lookup tables (LUTs). The x axis in the LUT represents the current vertex value, and the y axis the new vertex value.

By default, the curve is a diagonal line where all the points in the curve have the same value on the y axis (the new value) as they do on the x axis (the current value). Because both x and y values are the same, there is no change in the object's shape.

By modifying, for example, the CrosstalkGeo node's y LUT the following way, you can set some of the vertex y values of a sphere to 0 to squash its bottom half:

![Image of CrosstalkGeo node's LUT modification]

Modifying the CrosstalkGeo node's LUT.

With the CrosstalkGeo node, you can also use one of the vertex x, y, and z values to evaluate the lookup curve and then add the result to another vertex value. For example, you could modify the x->y curve, using the vertex x value to find the new value on the curve, and then add that to the vertex y value. This way, you can modulate the y values by another channel.

By default, these curves are horizontal lines at y=0. They produce no change, because the value added to the vertex (the new value on the y axis) is 0.
To Modify Objects Using Lookup Curves

1. Select 3D > Modify > CrosstalkGeo or LookupGeo to insert a CrosstalkGeo or LookupGeo node anywhere after the 3D object you want to modify.
2. Attach a Viewer to the node to see your changes.
3. In the node’s controls, use the display dropdown menu to select how you want to view your object in the Viewer while making changes to it.
4. From the list on the left, select the curve you want to modify. For example, you’d select z to only modify the vertex z values.
   In the case of the CrosstalkGeo node, you can also select y->x, for example, to use the vertex y value to evaluate the curve and add the result to the vertex x value.
5. Adjust the curve as necessary. To insert points on the curve, Ctrl/Cmd+Alt+click on the curve.
Modifying Objects Using a Power Function

The LogGeo node lets you modify the shape of your 3D objects using a power function. Using this node, you can raise each of the vertex x, y, and z values to a power ($X^a$, $Y^b$, $Z^c$). This can have a different effect depending on whether you are dealing with negative or positive values.

To Modify Objects Using a Power Function

1. Select 3D > Modify > LogGeo to insert a LogGeo node anywhere after the 3D object you want to modify.
2. Attach a Viewer to the node to see your changes.
3. In the node's controls, use the display dropdown menu to select how you want to view your object in the Viewer while making changes to it. See Object Display Properties.
4. Check swap. This swaps the values and the powers they are raised to around (for example, changes $5^7$ into $7^5$).
5. In the log x, y, and z fields, enter the power you want to raise the respective vertex values to. For example, if you want to raise the vertex z values to the power of 20, enter 20 in the z field.
   Alternatively, you can adjust your 3D object in the Viewer by dragging the white control point to a new location. You can find the control point just outside the object.
6. To clamp the negative x, y, and z values to 0.0, check clamp black. This option is only valid if you have checked swap.

Tip: If you set the log x, y, and z values to 1 and check swap, the LogGeo node produces no change in the incoming geometry. If you want to try out how the node works, this is a good place to start as you can then gradually adjust the values from there.

The following images illustrate the effect of the LogGeo node on the default Nuke cylinder and sphere when swap is checked in the LogGeo controls. Notice that if these objects were not positioned in the default location (centered around 0,0,0), the results would be different.
Modifying Objects Using a Power Function

The LogGeo node applied to the default cylinder and sphere: Log $x$, $y$, and $z$ set to 0.5.

The LogGeo node applied to the default cylinder and sphere: Log $x$, $y$, and $z$ set to 1 (no change).

The LogGeo node applied to the default cylinder and sphere: Log $x$, $y$, and $z$ set to 2.

The LogGeo node applied to the default cylinder and sphere: Log $x$, $y$, and $z$ set to 2.
Modifying Objects Using an Image - Method 1

With the DisplaceGeo node, you can modify geometry based on an image. When using the node, each vertex is displaced along its normal with a value corresponding to the image pixel the vertex’s uv attribute points to. The higher the pixel value, the greater the displacement.

The following image illustrates the principle behind the DisplaceGeo node. A Card node is modified to resemble the pattern of a Checkerboard image.

![DisplaceGeo node diagram](image)

Using the DisplaceGeo node to modify geometry.

To Modify Objects Using an Image

1. Select 3D > Modify > DisplaceGeo to insert a DisplaceGeo node anywhere after the 3D object you want to modify.
2. Attach a Viewer to the node to see your changes.
3. In the node's controls, use the **display** dropdown menu to select how you want to view your object in the Viewer while making changes to it.

4. Read in your image map and connect it to the DisplaceGeo node's **displace** input.

5. Adjust the following controls:
   - From the **channels** dropdown menu and check boxes, select the channels to use for the displacement value.
   - From the **source** dropdown menu, select the source for the displace value. For example, if you selected **rgb** or **rgba** from the **channels** dropdown menu, you can use the red, green, blue, or alpha channel or the pixel luminance as the source. You can also select **rgb relative** to move the vertices on the x, y, and z axes by the amounts in rgb, or **rgb absolute** to move the vertices to the values in rgb.
   - To define the scale of the displacement, adjust the **scale** slider. The higher the value, the bigger the displacement.
   - To give x, y, and z different weightings, enter new weights the **weight** fields. By default, each weighting is set to 1. If you don't want to make changes to a value, set its weight to 0.
   - To offset x, y, and z values, enter the value by which you want to offset them in the **offset** fields. For example, if you enter 0.5 in the y offset field, 0.5 is added to the y value.
   - To change the size of the filtering applied to the image before the displacement, adjust the **filter size** slider.
   - To select the filtering algorithm applied to the image before the displacement, select an algorithm from the **filter** dropdown menu. For more information, see [Choosing a Filtering Algorithm](#).
   - To change the name of the attribute that's used as the vertex's UV coordinates to find the image pixel, enter a name in the **attrib name** field.
   - Usually, the normals aren't correct after the vertices have been moved. To recalculate them after the displacement, check **recalculate normals**.
Modifying Objects Using an Image - Method 2

Like the DisplaceGeo node, the Displacement shader node also performs displacement mapping and at first glance the nodes seem very similar. However, the approach they have on performing displacement mapping is different.

Displacement mapping is a technique for adding geometric detail to object surfaces as you render them. Unlike the DisplaceGeo node, The Displacement node does this on the fly, only displacing those parts of the geometry that are visible at any given moment. Displacement considers the point of view of camera to determine which parts of the displacement need rendering, thus saving render time. It's also possible to optimize the level of tessellation to be the level that you need for an object at a certain distance.

Connecting the Displacement Node

1. Create the Displacement node by clicking 3D > Shader > Displacement.
2. Connect your geometry to the Displacement node's output. If you want, you can connect a texture to the Displacement node's input.
3. Connect the image you want to create the displacement from in the displacement input.
4. Optionally, you can use a separate map for calculating the normals. Connect this to the normals input.
5. Proceed to Adjusting the Displacement Controls below.

Adjusting the Displacement Controls

1. Use the displacement channel dropdown menu to select the channel from your displacement input that you want to use as the displacement map.
2. If you're using a normals input, uncheck build normals and set normal expansion to:
   - none to use the normals as they are,
   - XY to multiply them on x and y dimensions, and
   - XYZ to multiply them in x, y and z dimensions.
3. Use scale to set the overall scale of the displacement.
4. Set filter size to the size of the filter you want to use when sampling the input image.
5. Use the filter dropdown menu to select a filtering algorithm. For more information, see Choosing a Filtering Algorithm.

6. If you want to automatically calculate normals after the displacement, check build normals. Uncheck this, if you want the normals calculated from the normals input.

7. Proceed to Adjusting Displacement Controls for Rendering below.

Adjusting Displacement Controls for Rendering

Before rendering a scene where you’ve used Displacement, adjust the controls on the Tessellation tab to gain speed and quality for the render process:

1. Use max subdivision to set the maximum number of iterations of polygon subdivision that occurs in tessellation.
   If you’re used to working with DisplaceGeo or displacement in other applications, it’s worth noting that you can get a similar amount of detail with lower geometry subdivisions using the Displacement node.
   For example, a 30x30 Card is constructed from the tessellation of 1800 triangles (30x30x2), but you can achieve a similar tessellation levels using the Displacement node with fewer card subdivisions:

<table>
<thead>
<tr>
<th>Card subdivisions</th>
<th>max subdivisions</th>
<th>tessellation triangles</th>
</tr>
</thead>
<tbody>
<tr>
<td>1x1 (2 triangles)</td>
<td>5</td>
<td>2048</td>
</tr>
<tr>
<td>2x2 (8 triangles)</td>
<td>4</td>
<td>2048</td>
</tr>
<tr>
<td>4x4 (32 triangles)</td>
<td>3</td>
<td>2048</td>
</tr>
</tbody>
</table>

**Note:** Using a high number of geometry and Displacement subdivisions is likely to slow down rendering so far it becomes unusable - A 10x10 card with max subdivisions set to 4 generates 51200 triangles!

2. Set the mode dropdown menu to the mode used in polygon subdivision:
   • uniform - uniform polygon tessellation. This is a good option mainly for testing your result, and only in rare occasions the best option to actually use for a displacement render.
   • screen - tessellation is determined by the screen size. This is the default and often the best mode option. The tessellation is determined by the size of the tessellate polygons on the screen. This mode ensures that no new polygons are created once a particular polygon edge screen length is reached.
• **adaptive** - tessellation is determined by the complexity of the displacement. This option attempts to estimate flat areas in the image where tessellation is unnecessary. The calculation is based on the threshold controls which are only active if the **adaptive** mode is selected.

3. If you set **mode** to **screen** or **adaptive**, set **pixel edge length** to the maximum size of polygons used in tessellation.

4. If you set **mode** to **adaptive**, you can also adjust the following:
   - **edge threshold** - edges larger than this threshold get divided, whereas edges smaller than this are subdivided according to the normal and displace thresholds.
   - **normal threshold** - detects normal orientations to determine if the surface is flat or not. If the angle between adjacent normals is larger than this threshold, tessellation occurs.
   - **displace threshold** - compares the degrees to which two points on the surface are displaced, and if the results do not match, tessellation occurs.

**Note:** Keep in mind that applying the Displacement shader to a very complex high resolution geometry can be very slow.
Modifying Objects Using a Perlin Noise Function

The ProcGeo, or ProceduralNoise, node lets you modify your 3D objects using a Perlin noise function that creates seemingly random noise. For example, you could use the ProcGeo node to generate animated noise for rippling waves or clouds, or to create a terrain from a flat card, like in the following image:

Using the ProcGeo node to create a terrain from a card object.

You can select the type of noise and control its look in the ProcGeo node’s parameters.

To Modify Objects Using a Perlin Noise Function

1. Select 3D > Modify > ProceduralNoise to insert a ProcGeo node anywhere after the 3D object you want to modify.
2. Attach a Viewer to the node to see your changes.
3. In the node's controls, use the **display** dropdown menu to select how you want to view your object in the Viewer while making changes to it.
4. From the **ProceduralNoise Method** dropdown menu, select the type of noise you want to use: **Turbulence** or **fBm** (Fractal Brownian Motion).
5. To select whether to modify the x, y, or z values or all of them, use the **Orientation** dropdown menu.
6. To change the look of the noise, adjust the rest of the parameters. For example, to control the amount of detail of the noise, adjust **Octaves**.
Modifying Objects Using a Distortion Function

The RadialDistort node is a non-linear transformation of the vertices along directions from the object center, giving either a barrel or pin-cushion distortion. In the following image, two cylinders have been distorted using the RadialDistort node.

Barrel and pin-cushion distortions.

To Modify Objects Using a Distortion Function

1. Select 3D > Modify > Radial Distort to insert a RadialDistort node anywhere after the 3D object you want to modify.
2. Attach a Viewer to the node to see your changes.
3. In the node's controls, use the display dropdown menu to select how you want to view your object in the Viewer while making changes to it.
4. To select whether the distortion is a barrel or pin-cushion, adjust the distortion slider. Values below 0 produce a barrel distortion, whereas values above 0 produce a pin-cushion distortion. If you set the value to 0, the 3D object is not distorted.

5. To control the magnitude of the distortion, adjust the power bias slider. The higher the value, the more distorted the object becomes.

6. To move the center point of the distortion, enter new coordinates in the rotation center fields.

7. To control the amount of distortion in each of the x, y, or z directions, adjust the values in the scale fields.

8. To keep the object’s center in its original place in the 3D space, check preserve center.
Modifying Objects Using a Trilinear Interpolation

With the Trilinear node, you can warp the object as a whole by using a trilinear interpolation to warp the object’s bounding box. For example, you can use this node to create animated object deformations, such as the squish/squash of a bouncing ball.

To Modify Objects Using a Trilinear Interpolation

1. Select 3D > Modify > Trilinear to insert a Trilinear node anywhere after the 3D object you want to modify.
2. Attach a Viewer to the node to see your changes.
3. In the node’s controls, use the display dropdown menu to select how you want to view your object in the Viewer while making changes to it.
4. To move each corner of the bounding box, enter new coordinates in the p0, p1, p2...p7 fields. To cancel your changes and reset the box, select reset shape to input.
5. To not use the object’s bounding box but define a box yourself, go to the Source box tab and uncheck use incoming bounding box. Adjust the src0 and scr1 coordinates define the box. To change the color of the box, click the box button.
Materials and Textures

This section teaches you how to:

• use the nodes in the 3D > Shader menu to control what material your objects seem to be made of. See Object Material Properties.
• merge two Shader nodes together. See Merging Two Shader Nodes.
• merge a material with the objects rendered behind it in the 3D scene. See Merging a Material with the Objects Behind.
• replace selected material channels with a constant color to make one object hold out the others. See Replacing Material Channels with a Constant Color.
• project texture images onto your 3D objects. See Projecting Textures onto Objects.
• import a set of texture patches following the UDIM scheme and apply them to the surface of a 3D object. See Importing UDIM Patches.
Object Material Properties

The nodes under the **Shader** menu let you define the material attributes of geometric objects in a scene, including the quality of light reflected back to the camera from an object's surface. Using these nodes, you can control what material your objects seem to be made of.

You can also add several Shader nodes one after the other to produce more complex effects. For this, you should use the unlabeled inputs on the Shader nodes.

The material property settings you apply affect the render output of the scene.

You can insert 3D shader nodes in the following places in your scripts:
• between the 2D image you're using for the surface texture and the 3D object node that creates the surface, or
• after the 3D object nodes using the ApplyMaterial node. This is a good way to apply a global material to all objects. See *Applying a Material Using the ApplyMaterial Node*.

You can use the **map** connectors to input a mask image to limit the effect of the material change.

![Diffuse and Specular nodes.](image)

**Note:** You can only see the effect of your changes to an object's material properties in the 2D view.
Applying a Material Using the ApplyMaterial Node

The ApplyMaterial node applies a material from the `mat` input to your 3D object(s).

1. Select **3D > Shader > ApplyMaterial** to insert an ApplyMaterial node into your script.
2. Connect the unnamed input of the ApplyMaterial node to your geometry (for example, a Sphere, ReadGeo, or ModelBuilder node).

   ![Diagram of ApplyMaterial node](image)

**Tip:** If you want to apply a global material to several objects, you can also connect the unnamed input to a MergeGeo node. This overrides any materials applied to the individual geometry nodes before they were merged.

3. Connect your materials (for example, a 2D texture image, a BasicMaterial node, or a Wireframe node) to the ApplyMaterial node’s `mat` input.
By default, ApplyMaterial applies the material from the `mat` input onto all incoming geometry objects.

4. If you created your geometry using ModelBuilder or imported an Alembic file using ReadGeo, you can choose to only apply the material onto a particular object in the incoming geometry. To do so, open the ApplyMaterial properties and set `filter` to `name`. This allows you to tell ApplyMaterial to ignore any geometry that doesn't match the filter on the right.

**Note:** You can also limit a material to a particular object if your geometry was created or imported using a third-party plug-in that adds a name attribute for the geometry objects.

5. To set how to filter the incoming geometry objects, set the dropdown menu next to `name` to:
   - **equals** - set the material on any objects whose name matches the string in the filter name field exactly.
   - **doesn't equal** - set the material on any objects whose name does not match the string in the filter name field exactly.
   - **contains** - set the material for any objects whose name contains the string in the filter name field.

   This can be useful when you have some structure to your object names. For example, if you have objects like `/Root/Chair/Seat`, `/Root/Chair/Back`, and `/Root/Table`, you can select `contains` and set the filter name field to `Chair` to apply the material to all parts of the chair while leaving the table alone.

   - **doesn't contain** - set the material for any objects whose name does not contain the string in the filter name field.
6. To set the filter name, type the name directly into the text entry field or use the choose button to open the **Object Name Chooser** dialog and select a filter name from a list of incoming geometry objects.

![Object Name Chooser](image)

**Tip:** You can also Ctrl/Cmd+click or Shift+click in the **Object Name Chooser** dialog to select multiple objects.

### Adjusting the Diffuse Color

The Diffuse node lets you adjust the color of the material when illuminated. The material appears darker as the surface points away from the light, as the light is not falling on it.

1. Select **3D > Shader > Diffuse** to insert a Diffuse node into your script.
2. Place the Diffuse node between your 2D texture image and your 3D object node, or connect it to an ApplyMaterial node's `mat` input.
3. In the Diffuse properties, use the **channels** dropdown menu to select the channels you wish to process.
4. Adjust the **white** slider to control the diffuse color. By default, this is in grayscale, but you can also adjust the individual `r`, `g`, and `b` values. The higher the value, the brighter the material.
Adjusting Specular Highlights

You can use the Specular node to control how bright and wide the highlights on the material seem. The location of the viewpoint is significant: the specular highlights are the brightest along the direct angle of reflection.

1. Select 3D > Shader > Specular to insert a Specular node into your script.
2. Place the Specular node between your 2D texture image and your 3D object node, or connect it to an ApplyMaterial node’s mat input.
3. In the Specular properties, use the channels dropdown menu to select the channels you wish to process.
4. Adjust the white slider to control the brightness of the specular highlight. The higher the value, the shinier the material seems.

5. To control the width of the highlights, adjust the min shininess and max shininess sliders.
6. If necessary, adjust **shininess channel** to control how the input channels are used to map the black and white values to the **min shininess** and **max shininess** parameters when a **mapSh** input is connected. Select **red** to use the red channel for the mapping, **green** to use the green channel, **blue** to use the blue channel, **luminance** to use the luminance, or **average rgb** to use the average of the red, green, and blue channels.

**Simulating Materials That Emit Light**

You can use the Emission node to simulate lamps or other sources that emit light.

1. Select **3D > Shader > Emission** to insert an Emission node into your script.
2. In the Emission properties, use the **channels** dropdown menu to select the channels you wish to process.
3. Adjust the **emission** slider to change the brightness of non-illuminated areas for the surface.
   The higher the value, the more light the material seems to emit and the brighter it appears.

**Adjusting Diffuse, Specular, and Emission Using a Single Node**

The BasicMaterial node is a combination of the Diffuse, Specular, and Emission nodes, allowing you to control all three aspects of the material from a single properties panel.

1. Select **3D > Shader > BasicMaterial** to insert a BasicMaterial node into your script.
2. Place the BasicMaterial node between your 2D texture image and your 3D object node, or connect it to an ApplyMaterial node’s **mat** input.
3. The BasicMaterial node has several map inputs you can use to mask the effect of the node. Use:
   - **mapD** to modulate the diffuse component,
   - **mapS** to modulate the specular component,
   - **mapE** to modulate the emission component, and
   - **mapSh** to modulate the shininess value.
4. In the BasicMaterial properties, use the **channels** dropdown menu to select the channels you wish to process.
5. Adjust **emission** to change the color of the light the material emits. Note that when you have an image connected to the unlabeled input of the BasicMaterial node and adjust this value, you need to look at the rendered 2D image to see the effect of your changes. Changing the emission value does not have any effect in the 3D Viewer.
6. Adjust **diffuse** to control the color of the material when illuminated.
7. Use **specular** to control how bright the highlights on the material seem.
8. Adjust **min shininess** and **max shininess** to set the minimum and maximum shininess values. If you haven’t connected an image to the **mapSh** input of the node, the average of these values is used as the shininess value for the material.

9. Select a **shininess channel** to control how the input channels are used to map the black and white values to the minShininess and maxShininess parameters when a **mapSh** input is connected. Select **red** to use the red channel for the mapping, **green** to use the green channel, **blue** to use the blue channel, **luminance** to use the luminance, or **average rgb** to use the average of the red, green, and blue channels.

Simulating Smooth, Regular Surfaces

The Phong node uses the Phong algorithm to smooth edges between faces. It provides realistic shading and highlights for smooth materials, such as skin and other organic surfaces.

1. Select **3D > Shader > Phong** to insert a Phong node into your script.
2. Place the Phong node between your 2D texture image and your 3D object node, or connect it to an ApplyMaterial node’s **mat** input.
3. The Phong node has several **map** inputs you can use to mask the effect of the node. Use:
   - **mapD** to modulate the diffuse component,
   - **mapS** to modulate the specular component,
   - **mapE** to modulate the emission component, and
   - **mapSh** to modulate the shininess value.
4. In the Phong properties, use the **channels** dropdown menu to select the channels you wish to process.
5. Adjust **color** to change the material color.
6. Adjust **emission** change the color of the light the material emits.
7. To control the color of the material when illuminated, adjust **diffuse**.
8. To control how bright the highlights on the material seem, adjust **specular**.
9. To control how shiny the material appears, adjust **shininess**.
10. To set the minimum and maximum shininess values, adjust **min shininess** and **max shininess**. If you haven’t connected an image to the **mapSh** input of the node, the average of these values is used as the shininess value for the material.
11. Select a **shininess channel** to control how the input channels are used to map the black and white values to the **minShininess** and **maxShininess** parameters when a **mapSh** input is connected. Select **red** to use the red channel for the mapping, **green** to use the green channel, **blue** to use the blue channel, **luminance** to use the luminance, or **average rgb** to use the average of the red, green, and blue channels.
Rendering a Wireframe Overlay on Your Geometry

The Wireframe node allows you to render a wireframe overlay on the surface of your geometry object or particle simulation. This can be useful, for example, if you want to:

- check that your texture projection correctly lines up with your geometry.
- create a quick render of your 3D scene to check the positioning of objects.
- create motion graphics.
- create "making of" videos.

**Note:** The Wireframe node currently only works if you are rendering your 3D scene using ScanlineRender, RayRender and PrmanRender do not support the Wireframe shader.

1. Select 3D > Shader > Wireframe to insert a Wireframe node into your script.
2. Place the Wireframe node between your 2D texture image and your 3D object node, or connect it to an ApplyMaterial node's mat input.
3. In the Wireframe properties, use the **channels** dropdown menu to select the channels you wish to process.

4. From the **operation** dropdown, select how to apply the wireframe overlay to your geometry:
   - **opaque** - display the wireframe on fully opaque black input geometry.
   - **see through** - display the wireframe on fully transparent geometry.
   - **over** - display the wireframe on top of the input shader or texture.
• **multiply** - multiply the wireframe by the input shader or texture and display it on fully transparent geometry.

• **modulate** - apply standard diffuse shading to the wireframe and display it on top of the input shader or texture. This takes into account any lights in the scene.

5. To set the width of the wireframe lines (in pixels), adjust **line width**.
6. To set the color and transparency of the wireframe lines, adjust **line color**.

![Wireframe Overlay](image)

**Line width** set to 0.5. **Line width** set to 3.

**Line color** set to white. **Line color** set to cyan.
Merging Two Shader Nodes

With the **Shader** menu’s MergeMat node, you can combine two shader nodes together, using compositing algorithms like **none**, **replace**, **over**, and **stencil**. The MergeMat node is particularly useful for combining multiple Project3D nodes, allowing you to composite 2D images projected onto the 3D geometry atop each other.

**To Merge Two Shaders**

1. Select **3D > Shader > MergeMat** to add a MergeMat (over) node after the two shader nodes you want to combine.
2. Connect the MergeMat node to the **img** input of the 3D object you want to project the images on.
3. Connect the shader nodes to the MergeMat node’s **A** and **B** inputs. **A** refers to the foreground element, and **B** to the background element.

   For example, if you wanted to combine two Project3D nodes and composite their results onto a sphere, your node tree would look something like the following:

4. From the **operation** dropdown menu, select how you want to composite the results of the two shader nodes together:
   - to only use input **B** in the composite, select **none**.
   - to only use input **A** in the composite, select **replace**.
• to composite input A over input B using a mask, select over.
• to use input B outside the mask area, select stencil.
• to use input B inside the mask area, select mask.
• to add input B to input A, select plus.
• to use input A if it is greater than input B or else use input B, select max.
• to use input A if it is less than input B or else use input B, select min.

5. For operations (such as over) that need an alpha channel (mask), select which channel to use for the alpha from the alpha channels dropdown menu.
Merging a Material with the Objects Behind

The **Shader** menu's BlendMat node sets how the pixels colored by the material is applied to combine with the pixels from objects behind. It is like the MergeMat node, but instead of blending with another material, it blends with whatever is rendered behind in the 3D scene.

![Image of BlendMat node application]

The right-hand image shows the BlendMat node applied to the checkered sphere (that has a checkered alpha channel) and the BlendMat **operation** set to **stencil**.

**To Merge a Material with the Objects Behind**

1. Select **3D > Shader > BlendMat** to add a BlendMat node after the material you want to merge with the background pixels.
2. Connect the BlendMat node to the **img** input of the 3D object you want to project the material on.
3. From the **channels** dropdown menu, select the channels you want to affect.

4. From the **operation** dropdown menu, select how you want to composite the BlendMat node's input material and the background pixels together:
   - to set the material to black, select **none**.
   - to show the material where the material and the background overlap, select **replace**.
   - to composite the material over the background pixels according to the material's alpha, select **over**.
• to show the background pixels where the material's alpha is black, select **stencil**. Where the material's alpha is white, the material is set to black.
For this to work, the BlendMat node needs to process the alpha channel, so set **channels** to **rgba**.
This operation is the opposite of **mask**.

• to show the background pixels where the material's alpha is white, select **mask**. Where the material's alpha is black, the material is also set to black.
For this to work, the BlendMat node needs to process the alpha channel, so set **channels** to **rgba**.
This operation is the opposite of **stencil**.

• to add the background pixels to the material, select **plus**.
Merging a Material with the Objects Behind

- to use the material if its pixel values are greater than the background pixels or else use the background pixels, select **max**.

- to use the material if its pixel values are less than the background pixels or else use the background pixels, select **min**.
Replacing Material Channels with a Constant Color

The FillMat node lets you replace selected material channels with a constant color. Typically, you would use this node to make one object hold out the others. When you set the FillMat color to 0, it acts as a “3D cookie cutter” and makes a black hole where the material would otherwise be.

![A sphere in front of a cube.](image1)

![The same scene with the sphere material’s rgba channels set to black using the FillMat node.](image2)

![The alpha channel after applying the FillMat node.](image3)

This is similar to using a black Constant node as the input texture. However, the advantage of using the FillMat node is that you can easily apply it to the alpha channel in addition to the rgb channels. Another advantage is that the FillMat node doesn't break the shading sequence, so you can insert it after other material nodes in your node tree.

1. Select 3D > Shader > FillMat to insert a FillMat node between the 2D image you’re using for the surface texture and the 3D object node that creates the surface.
2. In the FillMat controls, use the **channels** controls to select the channels you want to replace with a constant color.

3. Use the **color** control to select the constant color. By default, this is set to black (0).
Projecting Textures onto Objects

You can use the UVProject and the Project3D nodes to project texture images onto your 3D objects. This way, you can add detail, surface texture, or color to your geometry, making the geometry more realistic and interesting.

The UVProject node changes the uv values of the vertices whereas the Project3D node is a material shader.

Projecting Textures with the UVProject Node

The UVProject node sets the uv coordinates for the object, allowing you to project a texture image onto the object. If the object already has uv coordinates, this node replaces them.

1. Select 3D > Modify > UVProject to insert a UVProject node anywhere after the 3D object you want to modify.
2. Attach a Viewer to the node to see your changes.
3. In the node’s controls, use the display dropdown menu to select how you want to view your object in the Viewer while making changes to it.
4. Connect an Axis or a Camera node to the UVProject node’s axis/cam input. If you connect an Axis node, project the texture UV coordinates onto the object using the axis transform values (that is, translation, rotation, scale, etc.). If you connect a Camera node, do a similar projection as with the axis but also use the camera lens information, such as the aperture.
5. Adjust the following parameters:
   • From the projection dropdown menu, select the projection type. Usually, it’s best to select a type that’s close to the object’s surface shape. For example, if your object is a sphere, like a football or a planet, select spherical.
   • From the plane dropdown menu, select the projection direction: XY, YZ, or ZX to project the texture image along the z, x, or y axis. This dropdown menu is only available if you selected planar as the projection type.
   • From the projecton dropdown menu, select both, front or back depending on whether you want to project the texture on the front face of the object, its back face or both. The front face of an object is the one facing the camera and similarly the back face is the one furthest away from the camera.
   • Check view frustum culling if you want the UVProject node to affect only the vertices inside the camera view frustum. Any vertices outside the view frustum are not affected and they still keep their original UV coordinates. Uncheck if you want the node to affect all vertices.
   • To mirror the texture UV coordinates in the horizontal direction, check invert u. To mirror them in the vertical direction, check invert v.
• To scale (stretch or squash) the texture UV coordinates in the horizontal direction, adjust the \texttt{u scale} slider. To scale them in the vertical direction, adjust the \texttt{v scale} slider. The higher the value, the more the texture is stretched.

• To change the name of the attribute that’s used as the vertex’s UV coordinates to find the image pixel, enter a name in the \texttt{attrib name} field.

Projecting Textures with the Project3D Node

The Project3D node projects an input image through a camera onto the 3D object.

1. Select \texttt{3D} > \texttt{Shader} > \texttt{Project3D} to insert a Project3D node after the image you want to project. Connect a Camera node to the Project3D node’s cam input.

2. Insert a 3D geometry node (a Sphere for example) after the Project3D node.

3. Attach a Viewer to the 3D geometry node to see your changes.

4. In the node’s controls, use the \texttt{display} dropdown menu to select how you want to view your object in the Viewer while making changes to it.

5. From the \texttt{project on} dropdown menu, select to project the image on either the front facing, back facing, or both polygons.

6. To extend the input image at its edges with black, check \texttt{crop}. To extend the image with the edge colors, uncheck \texttt{crop}.

7. If you want to use ray casting to test the projection and find out which parts of it are occluded, you can use the occlusion mode dropdown. Select:
   - \texttt{none} - to disable occlusion testing.
   - \texttt{self} - to tell Project3D that only the geometry connected to it can cause occlusion.
   - \texttt{world} - to tell Project3D that other objects in the scene can cause occlusion.
Importing UDIM Patches

When applying textures to models that use regions of UV space outside the standard (0,0) - (1,1) range, it's common to use one texture for each 1x1 square. These textures can be numbered in a variety of ways. UDIM is a numbering scheme that identifies the first texture that's applied to the (0,0) - (1,1) region as 1001, with numbers increasing by one for each texture in the U direction, and by ten for each texture in the V direction.

For example, the texture for the region (1,0) - (2,1) would have UDIM 1003, and the texture for (0,1) - (1,2) would have UDIM 1011. A set of patches following the UDIM scheme might look like this:
- color.1001.tif
- color.1002.tif
- color.1003.tif
- color.1004.tif
- color.1005.tif

Where color is the channel name and 1001, 1002, ... is the UDIM value for each patch in UDIM space. In Nuke, you can import a set of patches following the UDIM scheme and quickly apply them to the surface of a 3D object. To do this:

1. Select **Image > UDIM Import** from the Toolbar.
2. Select the UDIM image or sequence to import and click **Open**.

3. To ignore an individual patch, disable the checkbox next to it.
4. To add additional patches, click **Add Files**.
5. Check the **postage stamp** checkbox to enable a thumbnail view of the patch on the Read node.
6. Check the **group nodes** checkbox to place the Read nodes for each individual patch into a Group node. The name of the Group node is based on the name of the image or sequence.
7. Click **Ok** to import the files.

This creates a Read node for every patch in the sequence and appends a UVTile node (which allows you to modify a patch’s coordinates in UV space) to each one.

![Diagram of Read nodes and UVTile nodes](image)

**Note:** If you checked **group nodes** in the UDIM import dialog, highlight the Group node and press **Ctrl+Return** (**Cmd+Return** on a Mac) to view the contents.

The UDIM value for each patch is inserted into the **udim** field in the UVTile node. This value is important as it offsets every Read node (and therefore, patch) to the correct UDIM space. A number of MergeMat nodes are used to combine the Read/UVTile node combinations.

![MultiTexture node at the end of the tree](image)

**Note:** The MultiTexture node at the end of the tree optimizes rendering when multiple MergeMat nodes are chained together. It uses the default lop vertex shader and only handles multi-texturing. This node has no fields or knobs and is intended for use only with the **UDIM Import** operation.

To apply these patches to a 3D object in your scene, simply connect the output of the last node in the tree to the **img** input of a ReadGeo node.

To move a patch to a different point in UV space, in the UVTile control panel enter a different UDIM value or disable the **udim** field and enter values in the **u** and **v** fields.
**Note:** By default the **UDIM Import** operation parses the file name based on the UDIM number. It is possible, however, to create a custom parsing function to work with your own naming conventions. Please refer to the Python Developers Guide for details on how to do this.
Lighting

The nodes under the **Lights** menu let you control the lighting in your scene. Using these nodes, you can bring objects out or push them back, create an illusion of depth, simulate the conditions in the real world, or simply alter the feeling of the scene.

Nuke features four types of light you can use in your 3D scenes: direct light, point light, spot light, and environment light. You can add these using the DirectLight, Point, Spotlight, and Environment nodes. In addition to these, there is a Light node, which lets you create direct, point, and spot lights, as well as read in lights from `.fbx` files. For more information, see *Working with Lights*.

The Light, DirectLight, Point, and Spotlight nodes all have controls that you can use to adjust how the lights cast shadows in your 3D scene. See *Casting Shadows*.

Lighting is also affected by the 3D object normals, which are used to determine how the light should bounce off a surface at any particular point. In Nuke, the Normals node allows you to manipulate object normals in order to control the diffuse and specular light contributions. See *Manipulating Object Normals*.

There's also a Relight node, which takes a 2D image containing normal and point position passes and lets you relight it using 3D lights. See *Relighting a 2D Image Using 3D Lights*. 
Working with Lights

There are several types of lights that you can add and manipulate in your 3D scene to create the required effect.

Inserting Direct Lights

A direct light is a light that emits parallel light in one direction. It appears to illuminate all objects with equal intensity, as if it was coming from a far away source. Being at an infinite distance from the objects, direct light has orientation, but no position. A real world example of a direct light is the sun. You can use direct light to simulate sunlight and moonlight, for example.

1. Select 3D > Lights > Direct to insert a DirectLight node in your script.
2. Connect the DirectLight node to the Scene node.
3. In the DirectLight node’s controls, adjust the following:
   - Drag the color slider to change the light color.
   - Drag the intensity slider to change the brightness of the light.
   - To control the direction of the light, enter values in the rotate fields.

   ![DirectLight node controls](image)

   • To adjust the settings for shadows, change values for the controls on the Shadows tab. For more information on these controls, see Casting Shadows.

Inserting Point Lights

A point light is a point in 3D space that emits light in every direction. A real world example of a point light is a light bulb. You can use point light to simulate light bulbs, lamps, and candles, for example.

1. Select 3D > Lights > Point to insert a Point node in your script.
2. Connect the Point node to the Scene node.
3. In the Point node’s controls, adjust the following:
   • Drag the **color** slider to change the light color.
   • Drag the **intensity** slider to change the brightness of the light.
   • To control how much light the object gets from the light source (based on the distance between the object and the light source), use the **falloff type** dropdown menu. A **Linear** type diminishes the light at a fixed rate as it travels from the object, whereas **Quadratic** and **Cubic** types diminish the light at an exponential rate. If you select **No Falloff**, the distance between the light source and the object does not affect the lighting.
   • To control the position of the light in the 3D space, enter values in the **translate** fields.
   • To adjust the settings for shadows, change values for the controls on the **Shadows** tab. Note, however, that the Point light doesn’t cast shadows if you’re using ScanlineRender. For more information on these controls, see **Casting Shadows**.

### Inserting Spot Lights

A spot light is a point in 3D space that emits a cone-shaped light in a given direction. A real world example of a spot light is a desk lamp.

1. Select **3D > Lights > Spot** to insert a Spotlight node in your script.
2. In the node's controls, adjust the following:
   • Drag the **color** slider to change the light color.
   • Drag the **intensity** slider to change the brightness of the light.
   • Drag the **cone angle** slider to control the spread of the light (how wide or narrow the beam is) in degrees from 0 to 180.
   • Drag the **cone penumbra angle** slider to control the softness along the edge of the area of illumination. A negative value fades inward from the circle’s edge. A positive value fades outward from the circle’s edge. The **cone falloff** should be set to zero or a low value in order to see the softness. This feature is only visible in the rendered objects and not in the 3D OpenGL Viewer.
   • Drag the **cone falloff** slider to control how concentrated the light is (that is, how much the light diminishes from the center of the circular region out to the edge). The higher the value, the more focused the light becomes. The falloff is independent of the **falloff type**.
   • To control how much light the object gets from the light source (based on the distance between the object and the light source), use the **falloff type** dropdown menu. A **Linear** type diminishes the light at a fixed rate as it travels from the object, whereas **Quadratic** and **Cubic** types diminish the light at an exponential rate. If you select **No Falloff**, the distance between the light source and the object does not affect the lighting.
   • To control the direction of the light, enter values in the **rotate** fields.
   • To control the position of the light in the 3D space, enter values in the **translate** fields.
• To adjust the settings for shadows, change values for the controls on the Shadows tab. For more information on these controls, see Casting Shadows.

Inserting Environment Lights

An environment light is a light that illuminates the objects using an image of light from a real-world environment. This image-based lighting is generated using High Dynamic Range Images (HDRI). When HDR images are created, several differently exposed images are combined to produce a single image of the surrounding environment. As a result, HDR images have a wide range of values between light and dark areas, and represent the lighting conditions of the real world more accurately.

To use environment light, you first need to shoot a real life environment as an HDR image. Using the SphericalTransform node, you then convert this image into a spherical mapped image. The sphere is used to surround the 3D objects, so that the mapped image color illuminates them.

Environment light only works with shiny object materials that can reflect the mapped image. It results in a very realistic lighting that makes it easier to integrate the objects into the environment.

1. Read an HDR image of the environment into your script.
2. Select Transform > SphericalTransform to insert a SphericalTransform node after the HDR image. You use this node to convert the HDR image into a spherical mapped image. In the node’s controls, select the Input Type and the Output Type (in this case, Sphere).
3. Select 3D > Lights > Environment to insert an Environment node in your script. Connect the SphericalTransform node to the Environment node’s map input, and the Environment node to the Scene node.
4. In the Environment node's controls, adjust the following:
   • Drag the **color** slider to change the light color.
   • Drag the **intensity** slider to change the brightness of the light.
   • From the **filter** dropdown menu, select a filtering algorithm for the map image. For more information, see *Choosing a Filtering Algorithm*.
   • To change the blur size of the map image, adjust the **blur size** slider.
Inserting Direct Lights, Point Lights, or Spot Lights

The Light node includes the DirectLight, Point, and Spotlight nodes, so you can set it to act as any of these three nodes. The advantage of using a Light node in this way is that if you want to change the light type later, you can do so without setting up a new node. For example, you might insert a direct light, but then realize that what you actually need is a spot light. If you inserted the direct light using a DirectLight node, you need to delete this node and insert a SpotLight node instead. However, if you inserted the direct light using a Light node, you can simply change the light type from directional to spot in the Light controls.

**Tip:** The node can also be used to import lights from .fbx files. This is described under Importing Lights from FBX Files.

1. Select **3D > Lights > Light** to insert a Light node into your script.
2. In the Light controls, select the light type you want to use: point, directional, or spot. The controls are enabled and disabled according to the light type you select. For example, if you chose directional light, you get the same controls that appear on the DirectLight node.

3. Adjust the controls as necessary. For information on the functions of the controls, refer to the following:
   - If you selected point as the light type, see Inserting Point Lights.
   - If you selected directional as the light type, see Working with Lights.
   - If you selected spot as the light type, see Inserting Spot Lights.

**Using the Look Input**

You can use the optional look input of the Light node, so that the light automatically rotates to face towards the connected input. You can attach a Camera, Light, or Axis node to the look input. For example, you can connect an Axis node to the look input so that the light rotates to face the axis, wherever it is moved.
If you animate a card to move along the x axis, you can attach a Camera and a Light node with the `look` inputs so that they automatically rotate and face the card as it moves. To do this, complete the following steps:

1. After animating your card, insert an Axis node.
2. Expression link the Axis node to the Card node by holding **Ctrl**, clicking on the `translate` animation button in the Card node's properties, and dragging it to the `translate` animation button in the Axis node's properties. The expression link is displayed as a green line with an arrow denoting the direction of the expression. See Linking Expressions for more information.

3. Insert a Camera node and either drag the camera to the required position in the Viewer, or use the Camera node properties to adjust the position of the camera.
4. Connect the Camera's **look** input to the Axis node.

5. Insert a Light node and drag the light to required position in the Viewer, or use the Light node properties to adjust the position of the camera.
6. Connect the Light node's **look** input to the Axis node
7. Playback your animated card and notice that the Camera and Light follow the animated card.

![Animated card with Camera and Light](image)

**Importing Lights from FBX Files**

FBX is a standard 3D file format that gives you access to 3D scenes created in other applications supporting the same format. What you generally have in an .fbx file is an entire 3D scene containing cameras, lights, meshes, non-uniform rational B-spline (NURBS) curves, transformation, materials, and so on. From this scene, you can extract cameras, lights, transforms, and meshes into Nuke. This way, you can, for example, create a light in Maya, export it in a .fbx file, and use the same light again in Nuke.

**Note:** For the FBX SDK version used in Nuke, see [Third-Party Library Versions](#).

**Tip:** If you have trouble with .fbx files, it may be because they were written with an older version of FBX. If they load very slowly, it is also possible that they are ASCII rather than binary. To get around these problems, you can use the FBX converter on the Autodesk website ([http://usa.autodesk.com/fbx/download/](http://usa.autodesk.com/fbx/download/)). It converts between various different formats, including older FBX versions, ASCII, and binary, and is available on Windows, Mac, and Linux.

You can use the Light node to read in directional, point, and spot lights from FBX scene files (for more information on these three light types, refer to [Lighting](#)). One Light node only reads in one light. Therefore, if your .fbx file contains three lights and you want to import all of them into Nuke, you need to use three Light nodes.

To import a light from an .fbx file:

1. Select **3D > Lights > Light** to insert a Light node in the place where you want to add the light in your script.
2. In the Light controls, check **read from file**. This enables the controls on the **File** tab, allowing you to read in lights from an .fbx file. It also disables all controls whose values are filled from the .fbx file. You can still view these values and use them in expressions, but you cannot modify them, because they are read from the .fbx file. Any changes you make in the .fbx file are reflected in these values of the Light node.

3. On the **File** tab, click the folder icon and browse to the .fbx file that contains the light you want to use. Click **Open**.

4. From the **animation stack** dropdown menu, select the take you want to use from the .fbx file. FBX files support multiple takes in the same file. One of the takes is usually a default take without any animation.

5. From the **node name** dropdown menu, select the light node you want to import from the .fbx file.

6. If you want to override the frame rate used in the .fbx file to sample the animation curves, enter a new frame rate (frames per second) in the **frame rate** field. Check **use frame rate** to use the rate you entered (rather than the one in the .fbx file).

7. To scale the intensity channel values read from the .fbx file, adjust the **intensity scale** slider. If the light is too dark, increase this value.

8. If you want to modify the light properties imported from the .fbx file, uncheck **read from file** on the **Light** tab and make the necessary modifications. As long as **read from file** is unchecked, your changes are kept.

9. To reload the light properties from the .fbx file, make sure **read from file** is checked and click the **reload** button on the **File** tab.
Casting Shadows

The Light, Point, DirectLight, and Spotlight nodes all have controls that you can use to adjust how the lights cast shadows in your 3D scene. The following geometry nodes also have controls that allow you to select whether they receive shadows cast by the lights and whether they themselves cast shadows on other objects:

- Card
- Cube
- Cylinder
- Sphere
- MergeGeo
- ModelBuilder
- PointCloudGenerator
- PositionToPoints
- ReadGeo
- Scene

Note: The method used to create shadows varies between different render nodes:

- **ScanlineRender uses depth mapping to create shadows.** It first renders a depth map for each light that casts a shadow. The depth map is rendered from the light's point of view, and each pixel in the depth map represents the distance from the light to the nearest surface the light illuminates in a specific direction. The depth map is then compared to the render from the point of view of the camera. If a point is farther away in the image that the camera sees than in the depth map, then that point is considered to be in shadow.

Depth map shadows are often faster to render than raytraced shadows, but may not appear as realistic.
• **PrmanRender creates shadows through raytracing.** It fires individual light rays from the camera into the scene for each pixel. When a light ray hits a surface in the scene, PrmanRender traces so-called shadow rays between the intersection point and every light source in the scene. If there are obstacles between the intersection point and the light sources, the intersection point is considered to be in shadow.

The advantage of raytracing is that it can be used to create more accurate shadows and shadows that have soft edges, much like those in the real world. However, compared to creating shadows using depth mapping, raytracing can take much longer to render.

**To Cast Shadows:**

1. Open the properties panels for the 3D objects in your scene.
2. Check the **cast shadow** or **receive shadow** box, or both.
   - **Cast shadow** tells objects in a light’s path to cast shadows onto other objects in the scene.
   - **Receive shadow** tells objects to display shadows cast by other objects in the scene. This is useful for showing occlusion in a scene.

You can also use a Scene node to define these settings for all 3D objects connected to it. In the Scene properties, set **shadow** to **override inputs** and use the **cast shadow** and **receive shadow** controls to override the corresponding controls on the individual 3D objects.
3. Make sure you've got a Shader node attached to any 3D objects that you want to receive shadows.

4. Attach a light to your scene, and check the **cast shadows** box on the **Shadows** tab.
   In our example node tree below, we are using a ScanlineRender node, but you could use a PrmanRender node instead.

![Node Tree Diagram](image)

*Note:* If you are using a point light, casting shadows is not supported in the ScanlineRender node.

5. Depending on the render node you are using, proceed to either:
   - Adjusting Shadows When Using ScanlineRender below, or
   - Adjusting Shadows When Using PrmanRender.

### Adjusting Shadows When Using ScanlineRender

1. On the **Shadows** tab of the light node properties, set **shadow mode** to:
   - **solid** - Objects seen from the light that cast shadows are considered completely solid.
   - **clipped alpha** - objects that cast shadows are considered to be transparent if the object's alpha is below the light's clipping threshold control. All other alpha values fully occlude the light.
   - **full alpha** - Shadows are calculated based on how light is reduced when it passes through non-opaque occluders.

   This affects shadows cast by objects, based on the objects’ opacity.
Casting Shadows | Adjusting Shadows When Using ScanlineRender

2. If you want the light node to output the shadow map, set output mask to the channel where you want to store this. You can enable this even if cast shadows is disabled.

3. If you set shadow mode to solid or clipped alpha, you can adjust the following:
   - **depthmap resolution** - this sets the resolution of the depth map. Larger values result in shadows with less crunchy edges and less artifacts, but require more time to process.
     
     ![Depthmap resolution](image)
     
     Depthmap resolution set to 800. Depthmap resolution set to 7400.
     
   - **samples** - this sets the number of samples for the light when generating soft shadows. If soft shadows in your scene appear dotty or noisy, try increasing this value. The higher the value, the smoother the soft shadows become.
     
     ![Samples](image)
     
     Samples set to 1. Samples set to 50.

Note: You can also fix crunchy edges by increasing the number of samples instead of increasing the depth map resolution.
• **jitter scale** - the amount of jitter used when doing percentage-closer filtering (PCF) for soft shadows. A larger **jitter scale** value results in softer, more perceptually accurate shadows. PCF works by sampling the depth map at many different positions around the same spot. The final shadow value for that spot is an average of how many of the samples were occluded or visible from point of view of the light.

• **bias** - this is a constant offset that moves the surface sample point away from surface, towards the light that is casting the shadow. If self-shadowing artifacts appear in the image, you may want to increase this value. Note, however, that if you increase the value too much, some shadows may start moving away from the base of the objects that cast them.

![Shadow artifacts.](image1)

![Increasing bias can help reduce artifacts.](image2)

• **slope bias** - this is like **bias**, but the offset is proportional to the slope of the depth map. This allows you to give a different offset to each value in the depth map, depending on the surface's slope relative to the light. For example, if the surface's slope relative to the light is shallow, the value in the depth map may be a correct approximation and no offset (or only a very small offset) is needed. If the surface's slope relative to the light is steep, the values in the depth map are less likely to be a correct approximation and a larger offset is needed.

  If increasing **bias** reduced the existing self-shadowing artifacts but introduced more artifacts in other areas of the image, you may want to bring **bias** down a little and increase **slope bias** instead. Then, tweak both values until you’re happy with the result.

In the **clipped alpha** mode, you can also adjust:

• **filter** - the type of filter to use. For more information on the available filtering algorithms, see [Choosing a Filtering Algorithm](#).

• **clipping threshold** - any surface samples with alpha values below this threshold are considered transparent. The higher the value, the more areas on objects that cast shadows are considered transparent and pass through some light.
4. If you set shadow mode to full alpha, you can adjust the following:
   • filter - the type of filter to use. For more information on the available filtering algorithms, see Choosing a Filtering Algorithm.
   • scene epsilon - an offset that moves the sampling point away from the geometry surface, towards the light that is casting the shadow. Increasing this value can reduce self-shadowing artifacts.

**Tip:** To generate accurate shadows from a Direct light, view the scene through the light (using the Viewer's camera dropdown menu) and adjust the Direct light's scale control so that the part of the scene that should cast shadows fits within the view. This ensures that none of the shadow-casting geometry is missed by the depth map.

---

### Adjusting Shadows When Using PrmanRender

1. Open the PrmanRender properties and make sure shadows is enabled.
2. Open the light node properties and go to the Shadows tab.
3. Make sure shadow mode is set to solid. The other two modes aren't relevant when using PrmanRender.
4. If you want to cast soft shadows from the light, increase sample width. This value multiplies the width of the soft area around the edge of a shadow. The higher the value, the larger the soft area.

**Sample width** set to 1. **Sample width** set to 15.

5. If you increased sample width in the previous step and the resulting soft shadows in your scene appear dotty or noisy, increase samples. This sets the number of samples for the light when generating soft shadows. The higher the value, the smoother the soft shadows become.
Casting Shadows | Adjusting Shadows When Using PrmanRender

6. If self-shadowing artifacts appear in the image, increase the **bias** value. This moves the surface sample point away from surface. Note, however, that if you increase the value too much, some shadows may start moving away from the base of the objects that cast them.
Manipulating Object Normals

Object normals are vectors that are perpendicular to the surface. They are used in lighting calculations to determine how the light should bounce off a surface at any particular point. By manipulating them, you can control the diffuse and specular light contributions.

To Manipulate Object Normals

1. Select **3D > Modify > Normals** to insert a Normals node anywhere after the 3D object whose lighting you want to adjust.
2. Connect a Camera, Axis, or light node to the Normals node’s **lookat** input.
3. In the Normals controls, open the **action** dropdown menu and select:
   - **unchanged** to make no changes.
   - **set** to assign the normals value to the normal x, y, and z fields.
   - **build** to rebuild each normal based on the surrounding vertices. Adjust the **threshold angle** slider to determine the break angle where two faces no longer constitute a smooth surface. An angle of 0 means all faces are flat, whereas 180 means all faces are smooth. A good average setting is 60.
   - **lookat** to point all normals towards the Normals node's **lookat** input.
   - **delete** to remove the named attribute from the object. For example, if you remove the N attribute, the object has no normals.
Relighting a 2D Image Using 3D Lights

The Relight node takes a 2D image containing normal and point position passes and lets you relight it using 3D point lights. Essentially bypassing the need to return to a 3D application and re-render the lighting, Relight provides a quick and interactive way to relight a 3D scene in a 2D environment.

Relight works by applying a 3D shader to a 2D image using the normal and point position passes stored in separate image channels, and lets you attach and manipulate a 3D point light (or multiple lights via a Scene node).

Note: Relight only works with Light nodes set to light type > point.

To Relight a 2D Image Using Relight

1. From the 3D > Lights menu, select Relight to add the node to your script.
2. Read in a 2D image containing normal and point position passes and connect it to the color input of the node.

Tip: If the position pass and normal vectors are contained in separate images, they can be combined using a ShuffleCopy node, which connects to Relight through the color input.

You can create normal and point position passes using the DepthGenerator node in NukeX, for example. See Generating Depth Maps.

3. In the Relight properties panel, select the channel containing the normal data from the normal vectors dropdown menu.
4. Select the channel containing the point position information from the point positions dropdown menu.
5. Connect a Light node with light type > point to the lights input, or multiple lights via a Scene node.

6. Connect the camera that was used to render the original scene to the cam input.

7. Attach a shader (a Phong node, for example) to the material input. Depending on the type of shader you attach, ensure that you've defined the necessary properties for it. For information on defining material properties, refer to Object Material Properties.

   **Note:** The camera input does not appear until the light inputs have been connected to a light or scene node, and the material inputs do not appear until the camera input has been connected.

8. Switch to the 3D Viewer and position your lights.

9. If the image in the color input contains an alpha channel and you'd like to use this as a mask to limit the effects of Relight, check use alpha.

10. If required, adjust the ambient slider to set the global ambient light level for the scene.

11. To combine the lighting information from the Relight node with the original 2D image, use a Merge node with operation set to multiply. Connect the Relight node to the Merge node's A input and your 2D image to its B input.
Cameras

Nuke supports multiple cameras in a scene, with each providing a unique perspective.

For details on how to add a camera, look through it, and edit its lens characteristics, see Working with Cameras.

In addition to using cameras to view and render 3D scenes, you can also set up cameras that project 2D images onto geometry in your scene. For more information, see Projection Cameras.

If you have created a camera in a third-party 3D application (such as Maya) and want to use it in Nuke, you can export it from your 3D application in the FBX or Alembic format and then import it into Nuke. See Importing Cameras from FBX Files and Importing Cameras from Alembic Files.

If your camera was created in Boujou, you can also use the import_boujou.tcl script to import it into Nuke. See Importing Cameras from Boujou.

You can extrapolate a 3D point's position from imported camera data using the PointsTo3D node, which can help you position geometry in a scene. See <xref> for more information.

Tip: NukeX and Nuke Studio include CameraTracker which creates animated cameras without resorting to third-party applications. See Camera Tracking for more information.
Working with Cameras

This section explains how to add a camera to your script, look through it, lock it, and edit its lens characteristics.

To Add a Camera
1. Click **3D > Camera** to insert a Camera node.
   OR
   In the 3D Viewer, select **create camera** to place a new camera at the current position and orientation in 3D space.

2. To setup the rendering camera, drag a connector from the new Camera node to the ScanlineRender node.
   OR
   To setup an additional scene camera for viewing, drag a connector from the new Camera node to the Scene node.

To Look Through a Camera
1. Press **V** to make sure you are looking through the 3D perspective view, and not one of the orthographic views.
2. From the 3D Viewer window, select the camera from the dropdown menu in the top right corner.
Note: This selection does not change the camera used for rendering. This changes only the camera to “look through” for the current 3D Viewer.

Cameras in the current data stream automatically appear in the dropdown menu of cameras you can select. To select a camera that doesn't appear in menu, double-click the Camera node to open its panel and add it to the menu.

To Lock the 3D Camera View

You can lock the 3D view to the selected camera or light. You can toggle between the unlocked and locked modes by clicking the 3D view lock button, or by pressing Ctrl/Cmd+L.

- **unlocked** - move freely in the 3D view without restrictions. The 3D view lock button is gray.

- **locked** - lock your movement to the camera or light you've selected in the dropdown menu on the right side of the 3D view lock button. The 3D view lock button is red.

To Use the Interactive 3D Camera View Mode

With the interactive 3D camera view mode you can change the camera or light values according to your movement in the Viewer. You can activate the interactive mode by Ctrl/Cmd+clicking the 3D view lock button. When the interactive mode is on, the 3D view lock button turns green. In order to activate the interactive mode, you need to have a Camera or a Light node selected in the dropdown on the right side of the 3D view lock button.

When the interactive mode is on, you can use the plus (+) and minus (-) keys to change the translate values of the camera or light you've selected. When the interactive mode is off, these keys zoom your view in and out.

To Edit a Camera’s Lens Characteristics

1. If necessary, double-click on the Camera node to display its parameters.
2. Click the **Projection** tab.
3. Drag the **focal length** slider to adjust the camera's level of magnification.
4. Drag the **near** slide to edit the position of the camera's forward clipping plane. Objects closer to the camera than this plane are not rendered.
Note: The value for the near clipping plane must always be positive to produce a sensible result.

5. Drag the far slider to edit the position of the camera's rearward clipping plane. Objects farther from the camera than this plane are not rendered.

6. Increment the window translate u (horizontal axis) and v (vertical axis) sliders to translate the camera's output along either axis.

7. Increment the window scale u (horizontal axis) and v (vertical axis) sliders to scale the camera's output on either axis.

8. Drag the window roll slider to rotate the camera's output on z.

Using the Look Input

You can use the optional look input of the Camera node, so that the camera automatically rotates to face towards the connected input. You can attach a Camera, Light, or Axis node to the look input. For example, you can connect an Axis node to the look input so that the camera rotates to face the axis, wherever it is moved.

If you animate a card to move along the x axis, you can attach a Camera and a Light node with the look inputs so that they automatically rotate and face the card as it moves. To do this, complete the following steps:

1. After animating your card, insert an Axis node.

2. Expression link the Axis node to the Card node by holding Ctrl, clicking on the translate animation button in the Card node's properties, and dragging it to the translate animation button in the Axis node's properties. The expression link is displayed as a green line with an arrow denoting the direction of the expression. See Linking Expressions for more information.

3. Insert a Camera node and either drag the camera to the required position in the Viewer, or use the Camera node properties to adjust the position of the camera.
4. Connect the Camera's **look** input to the Axis node.

5. Insert a Light node and drag the light to required position in the Viewer, or use the Light node properties to adjust the position of the camera.
6. Connect the Light node's *look* input to the Axis node

7. Playback your animated card and notice that the Camera and Light follow the animated card.
Projection Cameras

In addition to viewing and rendering a 3D scene, cameras can also project a 2D still image or image sequence onto geometry in the scene. This is similar to the front-projection systems used in practical photography, where a background image or other element is projected onto the stage and photographed with other elements.

In Nuke, a projection camera can receive camera data tracked from the original shot - or another shot - to setup a projection that is match-moved to another source.

This setup requires these nodes: a projection camera, a Scene node, a Project3D node, a geometry object node (what you’ll be projecting onto), and a 2D node with the image that you want to project.

First a Little Math...

When you create a projection camera, you need to gather some information and do a few small calculations to make sure the projection works. Here are the bits of information you need:

• Focal length of the lens that photographed the projection image.
• Resolution of scanned image.
• Scanner pitch of the film scanning device.

After you have this information, you need to do these calculations to get the horizontal and vertical aperture settings for the projection setup:

\[
\text{horiz. res.} / \text{scanner pitch} = \text{horizontal aperture} \\
\text{vertical res.} / \text{scanner pitch} = \text{vertical aperture}
\]

So, for example, if your image resolution is 720 x 486 and the scanner pitch is 20, then these are the results:

\[
720 / 20 = \text{horizontal aperture} = 36 \\
486 / 20 = \text{vertical aperture} = 24.3
\]

Generally, for most professional projects, you can get the lens focal length from the camera report for the shot(s). If that is not available, you may be able to extrapolate lens information by running the shot through a 3D tracking application, such as Boujou, Syntheyes, or RealViz.
Setting Up the Projection Camera Script

Once you have the horizontal and vertical aperture and the lens focal length for the image you want to project, you can complete the projection camera setup.

**To Add a Projection Camera**

1. Select **3D > Camera** to add a new camera to your script and rename the node to identify it as a projection camera.
2. Select **3D > Shader > Project3D** to add a Project3D node to the script.
3. Connect the 2D image (i.e., Read node) to the Project3D node.
4. Connect the projection camera to the Project3D node.
5. Connect the Project3D node to the geometry node that should receive the 3D projection.
6. Double-click the projection camera node to load its parameters.

7. Click the **Projection** tab in the camera's panel and then enter the information you gathered for **focal length**, **horiz aperture**, and **vert aperture**.
When you are finished, view the 3D scene to check the placement of the projection. The next section explains how to preview the 2D and 3D elements together to check the results of the composite.

To View a 3D Scene over a 2D Background Image

1. Select the Scene node and press 1 to display its output to the Viewer.
2. If necessary, press Tab to toggle the Viewer to 3D mode.
3. Select the rendering camera object or node and press H to look through it.
4. Select the node with the 2D image you want to see in the Viewer, and then press Shift+2.
   The Shift+2 keystroke connects the image to the Viewer (assigning the next available connection, number 2), and also sets up the compare wipe.
5. Select the desired option from the Viewer composite dropdown menu, such as over, under, minus, and wipe).

This last step superimposes the two elements in the Viewer. The crosshair (shown below) is the control that lets you adjust the location and angle of the wipe for the comparison.
Comparing a 3D scene over a 2D image.
Importing Cameras from FBX Files

FBX is a standard 3D file format that gives you access to 3D scenes created in other applications supporting the same format. What you generally have in an .fbx file is an entire 3D scene containing cameras, lights, meshes, non-uniform rational B-spline (NURBS) curves, transformation, materials, and so on. From this scene, you can extract cameras, lights, transforms, and meshes into Nuke. This way, you can, for example, create a camera in Maya, export it in a .fbx file, and use the same camera again in Nuke.

Note: For the FBX SDK version used in Nuke, see Third-Party Library Versions.

Tip: If you have trouble with .fbx files, it may be because they were written with an older version of FBX. If they load very slowly, it is also possible that they are ASCII rather than binary. To get around these problems, you can use the FBX converter on the Autodesk website (http://usa.autodesk.com/fbx/download/). It converts between various different formats, including older FBX versions, ASCII, and binary, and is available on Windows, Mac, and Linux.

Importing Cameras from an FBX File

The Camera node lets you read in the standard FBX cameras (Producer Perspective, Producer Top, Producer Bottom, Producer Right, Producer Left, Producer Front, Producer Back) and any other cameras.

Using one Camera node, you can only import one camera from a .fbx file. If you need to import several cameras, you need to use one Camera node per camera.

1. Select 3D > Camera to insert a Camera node in the place where you want to add the camera in your script.
2. In the Camera controls, check read from file. When this is checked, the controls on the File tab are enabled, and you can use them to read in a camera from an .fbx file. Any controls whose values are read in from the .fbx file are disabled. You can still view these values and use them in expressions but, as long as read from file is checked, you cannot modify them. Modifying the
values in the .fbx file, however, affect the disabled values in the Camera controls, because these are reloaded from the .fbx file every time the node is instantiated.

3. To read in a camera from an .fbx file, click the folder icon on the File tab. Navigate to the .fbx file and select Open.

4. From the animation stack dropdown menu, select the take you want to use from the .fbx file. FBX files support multiple takes in one file. Usually, one of the takes is a default take with no animation.

5. From the node name dropdown menu, select the camera node you want to import from the .fbx file.

6. In the frame rate field, define a frame rate (frames per second) to sample the animation curves. To use this rate rather than the one defined in the .fbx file, check use frame rate.

7. To have the camera rotation values calculated using the look up vector and look at position, check compute rotation. If you don't check this, Nuke uses the rotation channel from the .fbx file instead of computing a new one. The rotation values are always computed when there is a look at target.

8. If you want to modify the camera properties imported from the .fbx file, uncheck read from file on the Camera tab and make the necessary modifications. As long as read from file is unchecked, your changes are kept.

9. To reload the camera properties from the .fbx file, make sure read from file is checked and click the reload button on the File tab.
Importing Cameras from Alembic Files

You can import cameras from Alembic files (.abc file format) into a Nuke scene. During the import, Nuke allows you to control which nodes in the Alembic scene get loaded by using an import dialog. If there is only one item in the Alembic file, it loads automatically.

For more information on Alembic, see http://code.google.com/p/alembic/

**Tip:** In addition to cameras, you can also import meshes (or NURBS curves/patch surfaces converted to meshes), point clouds, and transforms from Alembic files.

To learn how to export files in the Alembic (.abc) format, refer to Exporting Geometry, Cameras, Lights, Axes, or Point Clouds for more information.

To Import Cameras from an Alembic File:

1. Click **Image > Read** or press **R** on the Node Graph. The **Read File(s)** dialog displays.
2. Select the Alembic file you want to import from the file location, then click **Open**. The Alembic import dialog displays. By default, all items are selected in the Scene Graph when the import dialog is opened, as shown below:

   ![Alembic Scene Graph Example](image)

   Selected parent items are indicated by a yellow circle, and selected child items by a yellow bar (these turn orange when you select them in the list). Unselected items do not have an indicator next to them.
3. To import specific items, you must first deselect the root item by clicking on the yellow circle. This de-selects the root and any child items. Then, select specific items in the scenegraph by clicking on the blank space where the circles had been, as shown below:
Alternatively, you can right-click on an item and select:

- **Select as parent** - to select this item and make it a parent to other items. This allows you to create a separate Nuke node for this item (and any child items underneath it) in the next step.
- **Select as child** - to select this item and make it a child to the nearest parent item up the tree.
- **Deselect** - to deselect this item (that is, not import it from the scene).

You can also select multiple items by pressing **Ctrl/Cmd** or **Shift** while clicking them.

4. Do one of the following:
   - Click **Create all-in-one node** to create one Nuke node for everything that's selected, regardless of whether the items are selected as parent or child.
   - Click **Create parents as separate nodes** to create one Nuke node for each parent item (yellow circle) in the tree. This node contains all the child items (yellow bars) under the parent.

Nuke creates ReadGeo, Camera, and Axis nodes as necessary, depending on what you selected to import from the scene.

5. On the Camera nodes, you can adjust the following:
   - From the **animation stack** dropdown menu, select the take you want to use from the .abc file. Alembic files support multiple takes in one file.
   - From the **node name** dropdown menu, select the camera you want to import from the .abc file.
   - In the **frame rate** field, define a frame rate (frames per second) to sample the animation. To use this rate rather than the one defined in the .abc file, check **use frame rate**.
   - If you want to modify the camera properties imported from the .abc file, uncheck **read from file** on the Camera tab and make the necessary modifications. As long as **read from file** is unchecked, your changes are kept.
   - To reload the camera properties from the .abc file, make sure **read from file** is checked and click the **reload** button on the File tab.
   - To have the camera rotation values calculated using the look up vector and look at position, check **compute rotation**. If you don't check this, Nuke uses the rotation channel from the .abc file instead of computing a new one. The rotation values are always computed when there is a look at target.

**Tip:** To load specific items from an Alembic file, you can also create a ReadGeo, Camera, or Axis node, check **read from file**, and click the folder icon on the **File** tab to browse to an .abc file.
Importing Cameras from Boujou

Nuke is shipped with a script called import_boujou.tcl, which lets you load in cameras created with Boujou.

To Import a Camera from Boujou:

1. Save the Boujou camera solve as a .txt file.
2. In Nuke, click on a content menu button and select Script Editor. The Script Editor opens in the pane whose content menu you used.

3. In the input pane of the Script Editor (that is, the lower pane), enter nuke.tcl("import_boujou"). Click the Run the current script button on the top of the Editor, or press Ctrl+Return (Cmd+Return on a Mac).
4. In the File Browser that opens, navigate to the .txt file you saved in step 1.

A Camera, a ScanlineRender, and a Group node are loaded into Nuke. The Group node contains cylinders to represent points from Boujou.

**Tip:** You can also open Nuke's Boujou Text File Browser by doing the following:
1. Press x on the Node Graph to open the Nuke script command dialog.
2. In the dialog, check Tcl (if it's not already checked).
3. In the command field, enter import_boujou.
4. Click OK.

These steps can be used to replace the first three steps in the above instructions.
Locating a 3D Point from an Animated Camera

Animated cameras created in third-party applications, such as Maya, or Nuke's CameraTracker contain enough parallax information to locate a 3D point in a 2D scene using the PointsTo3D node. PointsTo3D uses three reference frames in an image sequence to extrapolate the location of a 2D point in 3D space.

Connecting the PointsTo3D Node

1. Read in your image sequence. See Loading Image Sequences for more information.
2. Import your animated camera as described under Cameras.

Tip: NukeX and Nuke Studio include CameraTracker, which creates animated cameras without resorting to third-party applications. See Camera Tracking for more information.

3. In the toolbar on the left of the interface, navigate to Transform > PointsTo3D to add a PointsTo3D node.
4. Connect the image sequence and Camera to the PointsTo3D node and then connect the Viewer.

Setting Reference Frames

1. In the Properties panel, set Camera type to free move.
2. Scrub through the sequence and pick out three frames where the feature is clearly visible.
3. Scrub to the first reference frame where your feature is visible and then drag the pointA Viewer widget from the bottom-left of the Viewer to the feature's location.
4. Click set frame.

The 2D coordinates and reference frame number are inserted into the Point A controls.

Tip: You can adjust the 2D point using the x and y controls in the properties for pixel-perfect positioning.

5. Scrub to the second reference frame where your feature is visible and then drag the pointB Viewer widget from the bottom-left of the Viewer to the feature's location.
6. Click set frame.
7. Scrub to the third reference frame where your feature is visible and then drag the pointC Viewer widget from the bottom-left of the Viewer to the feature's location.
8. Click set frame.

You can now see coordinate and frame information for all three reference frames in the properties.

Calculating the 3D Position

1. Click Calculate to display the range dialog.
2. Click the Frame range dropdown to select the range bounds and the frames to calculate, if required.
3. To calculate the position from the image sequence using a lower resolution, enable Use proxy. This can speed up the calculation, but the results may not be as accurate.
4. To stop the calculation if an error occurs during the process, disable Continue on error.
5. Click OK.
Nuke runs through the sequence, tracking the 2D position against the three reference frames to determine the corresponding 3D point.

Once the calculation is complete, the **point2d** Viewer widget is placed on the tracked point in 2D space.

6. Press **Tab** in the Viewer to switch to the 3D perspective view. The calculated 3D point is highlighted in 3D space.

![The 2D points used to calculate the 3D position.](image1)

![The resulting position in the 3D Viewer.](image2)

You can use the calculated 3D point to help you place elements in the scene, or click **generate axis** to add an Axis node to control all elements in the scene together. See [Parenting to Axis Objects](#) for more information on axes.
Transforming Geometry, Cameras, and Lights

Transform operations include moving, scaling, rotating the objects in your 3D scene. When an object node is active, you can enter specific transform settings in the node parameters (see Transforming from the Node Properties Panel), or directly manipulate the object with the transform handles displayed in the 3D Viewer (see Using the Transform Handles). Transformations occur from the location of the object’s pivot point (see Transformations and the Pivot Point).

Nuke allows you to transform several objects together or have one object always face another (see Parenting to Axis Objects and Using the TransformGeo Node).

You can also link transform parameters to imported track or camera data (see Applying Tracks to an Object), or control the transforms with animation curves.

Using FBX or Alembic files, you can import transforms from third-party 3D applications, such as Maya (see Importing Transforms from FBX Files and Importing Transforms from Alembic Files).
Using the Transform Handles

Transform handles appear when a 3D object with transform capabilities is loaded into the Properties Bin. The colors of the handles correspond to the axes available in 3D space: red transforms the x-axis, green transforms the y-axis, and blue transforms the z-axis.

To Move an Object with the Transform Handles

• Drag an object to move it on any axis.

  - Shift+drag to constrain movement to one axis.

To Rotate an Object with the Transform Handles

• Ctrl+click (Mac users Cmd+click) between the rotation rings and drag to rotate the object on any axis.

  • Click on a specific axis ring and drag to constrain the rotation to one axis.
To Scale an Object with the Transform Handles

- **Ctrl+Shift**+click (Mac users **Cmd+Shift**+click) and drag on the central yellow square to scale uniformly on all three axes.

- **Ctrl+Shift**+click (Mac users **Cmd+Shift**+click) and drag on a single axis square to scale on that axis.
Transforming from the Node Properties Panel

The transform handles (see Using the Transform Handles) are a convenient way to move objects around in the 3D workspace, but when you want more precision, you should enter values directly into the object's node panel. The panel also includes transform and rotation order options, which are not available within the 3D Viewer.

The following assumes you've already loaded the object's parameters into the Properties Bin.

To Set Transformation Options

• From the **transform order** dropdown menu, select an option to define the order by which transformations are executed (s signifies scale, r, rotation; and t, translation).
• From the **rotation order** dropdown menu, select an option to define the axial order by which rotations are executed.

To Transform an Object from Its Panel

• To move the object along one or more axes, increment or decrement the **translate x**, **y**, and **z** fields.
• To rotate the object, increment or decrement the **rotate x**, **y**, and **z** fields.
• To scale the object on all axes simultaneously, increment or decrement the **uniform scale** field.
• To scale the object asymmetrically (on x, y, or z), increment or decrement the **scale x**, **y**, and **z** fields.
• To skew the object (warp it by rotating its local axes), increment or decrement the **skew x**, **y**, and **z** fields to rotate the corresponding axis (and associated object vertices).
Transformations and the Pivot Point

When you make changes to an object's position, scaling and rotation, these occur from the location of the object's origin point or pivot. By default, the pivot point is located at the intersection of the object's local axes.

You can offset the pivot point and move it anywhere you like - you can even move it outside of the object. Subsequent local transformations then occur relative to the new pivot point location.

To Move the Pivot Point

1. Double-click on the object node to display its parameters.
2. Change the values of the pivot x, y, and z fields to move the local axis in any direction.

The object's graphical overlay points to the location of the pivot point with a line. All subsequent local transformations occur relative to this pivot point.

Once you've defined the location of an object's pivot point, you can use the object's transform parameters to translate, rotate, scale, and skew the object relative to the pivot point.
Parenting to Axis Objects

An axis object works as a null object by adding a new transformational axis to which other objects may be parented. Even when objects already have their own internal axes, it’s sometimes useful to parent in a separate axis.

For example, the Axis node has been parented to the other objects in the scenes (the two image planes and the camera). The result is an axis object that globally controls the scene. Rotating it, for example, rotates all objects in the scene, as the figure below shows.

![Rotating an entire scene with an axis object.](image)

**Tip:** To move several objects together, you can also merge them using a MergeGeo node (see Merging Objects) and then control them using a TransformGeo node (see Using the TransformGeo Node).

To Add an Axis Object

1. Click 3D > Axis to insert an Axis node.
2. To create the parent relationships, connect the Axis node to all object nodes you wish to control with the Axis transformation controls.
To create a nested transformational hierarchy, chain additional Axis nodes to the first one you inserted. For example, you could create a hierarchy of three Axis nodes to control rotation, scale, and transform.

Creating a nested transformational hierarchy.

In the above example, Axis3 (rotation) is connected to Axis2 (scale). Axis2 is connected to Axis1 (transform), and Axis1 is connected to the TransformGeo node(s) that you want to affect. With the Axis nodes connected in this manner, their transformation data ripples down the chain and is added to the settings of the TransformGeo node.
Using the Look Input

You can use the optional **look** input of the Axis node, so that the axis automatically rotates to face towards the connected input. You can attach a Camera, Light, or Axis node to the **look** input. For example, you can connect an Camera node to the **look** input so that the axis rotates to face the camera, wherever it is moved. See Working with Lights or Working with Cameras for an example.
Using the TransformGeo Node

The TransformGeo node allows you to move, rotate, scale, and perform other transformations on several objects merged together with a MergeGeo node. It also lets you connect geometry objects to an Axis node. By doing so, you can move all the connected objects together by using the Axis transformation controls. All you need to do is insert a TransformGeo after each geometry object, connect the Axis node to the TransformGeo nodes' axis input, and adjust the transform controls of the Axis node. For more information, see Parenting to Axis Objects.

Another use of the TransformGeo node is to have the rotation of one object depend on the position of another so that the first object is always rotated to face or “look at” the second one. For example, you can have a sphere object always facing a cylinder object, regardless of the cylinder's position. When the cylinder is moved to a new position, the sphere is automatically rotated to face it.

To Have One 3D Object Always Face Another

1. Select the 3D object node (for example, a sphere) that you want to face another object.
2. Select 3D > Modify > TransformGeo to insert a TransformGeo node.
3. Select the object you want the first object to face (for example, a cylinder), and insert a TransformGeo after this node, too.
4. Connect the first TransformGeo node into the look input of the second TransformGeo node.
5. Open the controls of the first TransformGeo node and go to the **Look** tab.

6. From the **look axis** dropdown menu, select the axis around which the object rotates to face the other object:
7. Use the rotate X, rotate Y, and rotate Z check boxes to select the axes the object rotates around. For the first object to truly face the second, you need to check all three check boxes.

8. Adjust the look strength slider to define the extend of the rotation. The smaller the value, the less the object is rotated. Setting the value to 0 produces no rotation.

9. If you want to use an alternate scheme to calculate the rotation, check use quaternions. This may be useful for smoothing out erratic rotations along the selected look axis.

If you now adjust the second TransformGeo node's transform controls, you'll notice that the first object automatically rotates to face the second object. For more information on how to adjust the transform controls, see Using the Transform Handles and Transforming from the Node Properties Panel.
Applying Tracks to an Object

Nuke can import channel files and apply the motion data to the transformation parameters of any camera or object. The most common purpose for this is to simulate a practical camera move or move objects along a defined path.

Channel files contain a set of Cartesian coordinates for every frame of animation in a given shot. This information is calculated by 3D tracking software, such as 3D-Equalizer, Maya, or Boujou, and then exported as channel files.

To apply a channel file to an object

1. Double-click on an object or camera node to display its parameters.
2. Click import chan file. The file navigation dialog appears.
3. Navigate to the channel file, then click OK.
4. Nuke reads in the channel data and displays a status message about the number of data frames imported. You'll also notice the object's translation parameters turn green to indicate these parameters are now controlled by animation data. Scrub the Viewer and you'll notice the object or camera now moves according to the transformation data imported from the channel file.

**Note:** You can use the export chan file button to export as a chan file any animated translation parameters which you've applied to given object. This is a useful method of sharing setups between artists.

**Tip:** You can also use channel files to import cameras created in other applications into Nuke. However, as the chan file format is not a standard file format, you may need a file format converter to export chan files from other applications.
Importing Transforms from FBX Files

FBX is a standard 3D file format that gives you access to 3D scenes created in other applications supporting the same format. What you generally have in an .fbx file is an entire 3D scene containing cameras, lights, meshes, non-uniform rational B-spline (NURBS) curves, transformation, materials, and so on. From this scene, you can extract cameras, lights, transforms, and meshes into Nuke. This way, you can, for example, create a transform in Maya, export it in a .fbx file, and use the same transform again in Nuke.

**Note:** For the FBX SDK version used in Nuke, see Third-Party Library Versions on page 2021.

**Tip:** If you have trouble with .fbx files, it may be because they were written with an older version of FBX. If they load very slowly, it is also possible that they are ASCII rather than binary. To get around these problems, you can use the FBX converter on the Autodesk website (http://usa.autodesk.com/fbx/download/). It converts between various different formats, including older FBX versions, ASCII, and binary, and is available on Windows, Mac, and Linux.

Importing Transforms from an FBX File

The Axis node reads in transforms, markers and nulls (locators) from FBX files. You can use it to import one transform, marker, or null per Axis node.

1. Select 3D > Axis to insert an Axis node in your script. Connect the Axis node to a Scene node.
2. In the Axis controls, check read from file. This enables the controls on the File tab, allowing you to import transforms from an .fbx file. It also disables controls whose values are filled in from the .fbx file. As long as read from file is checked, you cannot modify these values. You can, however, view them and use them in expressions. The values are reloaded from the .fbx file every time the node is instantiated, so any changes you make in the .fbx file's values are reflected in the Axis controls.
3. On the **File** tab, click the folder icon to open the File Browser. Navigate to the `.fbx` file that contains the transform you want to use. Click **Open**.

4. From the **animation stack** dropdown menu, select the take you want to use from the `.fbx` file. FBX files support multiple takes, one of which is usually a default take with no animation.

5. From the **node name** dropdown menu, select the transform, marker, or null you want to import from the `.fbx` file.

6. If you do not want to use the frame rate from the `.fbx` file for sampling the animation curves, in the **frame rate** field, enter a new value (frames per second). To override the frame rate defined in the `.fbx` file and use the one you defined here, check **use frame rate**.

7. If you want to modify the transform properties imported from the `.fbx` file, uncheck **read from file** on the **Axis** tab and make the necessary modifications. As long as **read from file** is unchecked, your changes are kept.

8. To reload the transform properties from the `.fbx` file, make sure **read from file** is checked and click the **reload** button on the **File** tab.
Importing Transforms from Alembic Files

You can import transforms from Alembic files (.abc file format) into a Nuke scene. During the import, Nuke allows you to control which nodes in the Alembic scene get loaded by using an import dialog. If there is only one item in the Alembic file, it loads automatically.

For more information on Alembic, see http://code.google.com/p/alembic/

**Tip:** In addition to transforms, you can also import meshes (or NURBS curves/patch surfaces converted to meshes), point clouds, and cameras from Alembic files.

To learn how to export files in the Alembic (.abc) format, refer to Exporting Geometry, Cameras, Lights, Axes, or Point Clouds for more information.

To Import Transforms from an Alembic file:

1. Click **Image** > **Read** or press **R** on the Node Graph.
   The **Read File(s)** dialog displays.
2. Select the Alembic file you want to import from the file location, then click **Open**.
   The Alembic import dialog displays. By default, all items are selected in the Scene Graph when the import dialog is opened, as shown below:

   ![Alembic Import Dialog]

   Selected parent items are indicated by a yellow circle, and selected child items by a yellow bar (these turn orange when you select them in the list). Unselected items do not have an indicator next to them.
3. To import specific items, you must first deselect the root item by clicking on the yellow circle. This de-selects the root and any child items. Then, select specific items in the scenegraph by clicking on the blank space where the circles had been, as shown below:
Alternatively, you can right-click on an item and select:

- **Select as parent** - to select this item and make it a parent to other items. This allows you to create a separate Nuke node for this item (and any child items underneath it) in the next step.
- **Select as child** - to select this item and make it a child to the nearest parent item up the tree.
- **Deselect** - to deselect this item (that is, not import it from the scene).

You can also select multiple items by pressing Ctrl/Cmd or Shift while clicking them.

4. Do one of the following:

   - Click **Create all-in-one node** to create one Nuke node for everything that’s selected, regardless of whether the items are selected as parent or child.
   - Click **Create parents as separate nodes** to create one Nuke node for each parent item (yellow circle) in the tree. This node contains all the child items (yellow bars) under the parent.

   Nuke creates ReadGeo, Camera, and Axis nodes as necessary, depending on what you selected to import from the scene.

5. On Axis nodes, you can adjust the following:

   - From the **animation stack** dropdown menu, select the take you want to use from the .abc file. Alembic files support multiple takes in one file.
   - From the **node name** dropdown menu, select the transform, marker, or null you want to import from the .abc file.
   - In the **frame rate** field, define a frame rate (frames per second) to sample the animation. To use this rate rather than the one defined in the .abc file, check **use frame rate**.
   - If you want to modify the transform properties imported from the .abc file, uncheck **read from file** on the Axis tab and make the necessary modifications. As long as **read from file** is unchecked, your changes are kept.
   - To reload the transform properties from the .abc file, make sure **read from file** is checked and click the **reload** button on the File tab.

   **Tip:** To load specific items from an Alembic file, you can also create a ReadGeo, Camera, or Axis node, check **read from file**, and click the folder icon on the File tab to browse to an .abc file.
Adding Motion Blur to the 3D Scene

To create more realism for a 3D scene, you'll want to add motion blur to it based on the movement of your 3D camera. This can be done in two ways:

- Adjust the `samples` value in the ScanlineRender or RayRender node's Properties panel. The image is sampled multiple times over the shutter period. This way is the most accurate, but also the slowest, because the full rendering calculation is done multiple times for each pixel. See Adding Motion Blur Using a Renderer.

- Use the VectorBlur node to generate motion blur based on motion vectors. This way is faster to render, as it is only affected by the pixel resolution of the final image rather than the complexity of your 3D scene. See Adding Motion Blur Using VectorBlur.
Adding Motion Blur Using a Renderer

To add motion blur using ScanlineRender or RayRender:

1. In the node’s controls, go to the MultiSample tab.
2. Increase the samples value to sample the image multiple times over the shutter time. The higher the value, the smoother the result, but the longer the render time.

   ![Samples set to 5.](image1)
   ![Samples set to 15.](image2)

3. In the shutter field, enter the number of frames the shutter stays open when motion blurring. For example, a value of 0.5 would correspond to half a frame. Increasing the value produces more blur, and decreasing the value less.

   ![Shutter set to 20.](image3)
   ![Shutter set to 40.](image4)

4. Adjust the shutter offset using the shutter offset dropdown menu. The different options control when the shutter opens and closes in relation to the current frame value. Select:
   - centered - to center the shutter around the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 29.5 to 30.5.
   - start - to open the shutter at the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 30 to 31.
• **end** - to close the shutter at the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 29 to 30.

• **custom** - to open the shutter at the time you specify. In the field next to the dropdown menu, enter a value (in frames) you want to add to the current frame. To open the shutter before the current frame, enter a negative value. For example, a value of **-0.5** would open the shutter half a frame before the current frame.

5. If you're using ScanlineRender, you can add randomness to the distribution of samples in time so they don't produce regularly spaced images, by adjusting **randomize time**. This affects all samples except the first and the last one. The larger the value, the larger the time difference between the samples.

6. If you're using ScanlineRender, set **sample diameter** to the diameter of the circle that the samples for each pixel are placed in for antialiasing. The larger the value, the more pixels are jittered.

7. If you're using ScanlineRender, adjust the **focus diameter** to randomly orbit the camera about a point at the **focal distance** in front of it for each sample to produce depth-of-field effects from multiple samples.

8. If rendering has become very slow, you can approximate multi-sample rendering and reduce render times by using **stochastic samples**. This sets the number of samples, per pixel, to use in stochastic estimation (0 is disabled). Lower values result in faster renders, while higher values improve the quality of the final image.

   Note that rather than setting **stochastic samples** to a very large value, you may want to leave it at a relatively low value and increase **samples** instead.

9. To distribute the samples uniformly over the shutter time, enable **uniform distribution**. This gives the same importance to all samples over the shutter time, and produces more accurate results for stochastic multi-sampling.

For more information on the ScanlineRender and RayRender nodes, see *Rendering a 3D Scene*.
Adding Motion Blur Using VectorBlur

Nuke’s VectorBlurr node generates motion blur by blurring each pixel into a straight line, using the values from the motion vector channels (u and v channels) to determine the direction of the blur. Compared to generating motion blur using the ScanlineRender node’s MultiSample controls, this is less accurate but faster to render.

You can create the necessary motion vectors for use with VectorBlur in several ways:

• Use the ScanlineRender or RayRender nodes. This is usually more accurate than using a MotionBlur3D node (described below), and works well with both static and moving cameras as well as both linear and non-linear camera movement. See To use VectorBlur with ScanlineRender or To use VectorBlur with RayRender.

• Alternatively, if your 3D scene is static or nearly so and the camera movement over the shutter time is nearly linear, you can also use the Filter > MotionBlur3D node. However, note that MotionBlur3D only uses camera information to produce motion vectors. If you have moving objects in your scene but the camera is static, MotionBlur3D isn’t able to produce any output. See To use VectorBlur with MotionBlur3D.

• Several third-party 3D applications can also produce motion vector information as two-channel, floating point images that you can use with VectorBlur. If possible, you should unpremultiply these images. See To use VectorBlur with Third-Party Motion Vectors.

To use VectorBlur with ScanlineRender

1. In your 3D scene, open the ScanlineRender properties and go to the Shader tab.
2. Make sure motion vectors is set to one of the following (not off):
   • classic - Render motion vectors the classic (pre-Nuke 6.1) way. This option is only provided for backwards compatibility, and isn't always accurate.
   • velocity - Store the velocity of every single pixel in the motion vector channels (pre-Nuke 7.0 way). This option is only provided for backwards compatibility. In order to have the same behavior as Nuke 6.3, set samples to 1.
   • distance - For every pixel, store the distance (in pixels) between samples in the motion vector channels. This is the recommended option that usually produces the best results. It also allows the VectorBlur node to produce curved vector blur where interpolation between two frames is made according to a curve rather than linearly.
3. Use motion vector channels to select where to store the generated motion vectors.
If you view the channels you selected, you should see they now contain motion vectors.

A render of a 3D scene. Motion vectors produced by ScanlineRender.

4. On the **MultiSample** tab, leave **samples** set to 1. You can increase this value later if non-linear movement cannot be approximated sufficiently well using VectorBlur’s linear approach.

5. Select **Filter > VectorBlur** to insert a VectorBlur node after ScanlineRender.

6. In the VectorBlur properties, use the **channels** dropdown menu to select the channels to blur.

7. Select the motion vector channels from the **uv channels** dropdown menu. These should be the same channels as the channels you chose to create in step 3.

8. Set the **mv presets** dropdown to set which renderer format to apply to the motion vectors. Each preset contains a scale and offset specific to the renderer, so that the vectors are in the format expected by Nuke.

9. Use the **blur uv** dropdown to blur the motion vectors themselves, before blurring the image, to smooth the results between regions with very different motion. Choose either:
   - **uniform** - applies a small uniform blur to soften edges in the UV map,
### Adding Motion Blur Using VectorBlur

- **linear** - applies a linearly-weighted blur to blend between regions with different motion. This has a stronger effect than the **uniform** option.

**Tip:** The **uniform** option can be used to create blur similar to pre-Nuke 10 results.

10. Set **shutter offset** to 0. This means the shutter opens at the current frame.
11. Select the **blur type** using the dropdown.
12. To adjust the amount of blur, adjust the **motion blur** controls.

![The original image.](image1) ![Motion blur produced by VectorBlur.](image2)

### To use VectorBlur with RayRender

1. In your 3D scene, open the RayRender properties and go to the **AOVs** tab.
2. Enable **output AOV** and then use the **motion vector** dropdown to select where to store the generated motion vectors.
   - If you view the channels you selected, you should see they now contain motion vectors.

   ![A render of a 3D scene.](image3) ![Motion vectors produced by ScanlineRender.](image4)

3. On the **MultiSample** tab, leave **samples** set to 1. You can increase this value later if non-linear movement cannot be approximated sufficiently well using VectorBlur’s linear approach.
4. Select **Filter > VectorBlur** to insert a VectorBlur node after RayRender.

![A node tree with RayRender and VectorBlur.](image)

5. In the VectorBlur properties, use the **channels** dropdown menu to select the channels to blur.
6. Select the motion vector channels from the **uv channels** dropdown menu. These should be the same channels as the channels you chose to create in step 2.
7. Set the **mv presets** dropdown to set which renderer format to apply to the motion vectors. Each preset contains a scale and offset specific to the renderer, so that the vectors are in the format expected by Nuke.
8. Use the **blur uv** dropdown to blur the motion vectors themselves, before blurring the image, to smooth the results between regions with very different motion. Choose either:
   - **uniform** - applies a small uniform blur to soften edges in the UV map, or
   - **linear** - applies a linearly-weighted blur to blend between regions with different motion. This has a stronger effect than the **uniform** option.

   **Tip:** The **uniform** option can be used to create blur similar to pre-Nuke 10 results.

9. Set **shutter offset** to 0. This means the shutter opens at the current frame.
10. Select the **blur type** using the dropdown.
11. To adjust the amount of blur, adjust the **motion blur** controls.
To use VectorBlur with MotionBlur3D

1. In your 3D scene (which should be static or nearly static), open the ScanlineRender properties, go to the Shader tab, and set motion vectors to off.
   This tells ScanlineRender not to produce motion vectors, which is what you want if you are going to use MotionBlur3D to generate the vectors.
2. Select Filter > MotionBlur3D to insert this node and connect it to the ScanlineRender node's output.
3. Connect the rendering camera to the cam input of the MotionBlur3D node. Note that this has to be a moving camera. The camera movement over the shutter time should be linear or nearly linear.
4. In the MotionBlur3D properties, use Output UV to select where to store the generated motion vectors.
   If you view the channels you selected, you should see they now contain motion vectors.
5. Select Filter > VectorBlur to insert this node and connect it to the MotionBlur3D node.
6. In the VectorBlur properties, select the motion layer from the **uv channels** dropdown menu.

7. Use the **blur uv** dropdown to blur the motion vectors themselves, before blurring the image, to smooth the results between regions with very different motion. Choose either:
   - **uniform** - applies a small uniform blur to soften edges in the UV map, or
   - **linear** - applies a linearly-weighted blur to blend between regions with different motion. This has a stronger effect than the **uniform** option.

   **Tip:** The **uniform** option can be used to create blur similar to pre-Nuke 10 results.

8. Select the **blur type** using the dropdown.

9. To adjust the amount of blur, adjust the **motion blur** controls.

10. To adjust the length of the blur, adjust the **Shutter** setting in the MotionBlur3D **Properties** panel.
To use VectorBlur with Third-Party Motion Vectors

1. Select Filter > VectorBlur to insert a VectorBlur node after a 2D image that contains motion vector channels.
2. In the VectorBlur properties, use the channels dropdown menu to select the channels to blur.
3. Select the motion vector channels from the uv channels dropdown menu.
4. Set the mv presets dropdown to apply the presets associated with the selected renderer’s default settings, if present.
5. If your motion vectors have been premultiplied, check uv alpha and specify the channel used to premultiply the image.
6. Select the blur type and the required amount of blur using the motion blur controls.
Exporting Geometry, Cameras, Lights, Axes, or Point Clouds

You can export geometry, cameras, light, axes and point clouds into an FBX (.fbx) or Alembic (.abc) file using the WriteGeo node.

1. Create a WriteGeo node and attach it to a Scene node.
2. In the WriteGeo controls, select fbx or abc in the file type dropdown, and check the objects to export:
   - geometries - to write the scene geometries into the file.
   - cameras - to write the scene cameras to the file.
   - lights - to write the scene lights into the file (.fbx only).
   - axes - to write the scene axes into the file.
   - point clouds - to write the scene point clouds into the file.
3. If you're writing FBX files:
   - Check ascii file format if you don't want to write out a binary .fbx file.
   - Check animate mesh vertices if you want the mesh vertices animated and keyframes created at every frame. The animated meshes use vertex point caches for the data. A directory with _fpc appended to the file name is created to contain the point caches. By default the animated meshes check box is off so the point cache directory is not created.
4. If you're writing Alembic files, select the storage format to use when writing the file:
   - HDF - A storage format that maintains backwards compatibility.
   - Ogawa - A storage format that offers faster file reading and smaller files.
5. Browse to the destination you want to save the .fbx or .abc file in the file field and give it a name.
6. Check the limit to range box if you want to disable the node when executed outside of a specified frame range. In the frame range fields, enter the range of frames you want to make executable with this Write node.
7. Select the views to export, such as left and right for stereoscopic projects.
8. Click Execute.
9. If necessary, you can change the frame range you want to include in the file using the pop-up dialog and then click OK.
   A progress bar displays, and the selected file is saved to the specified directory.
Rendering a 3D Scene

The 3D Viewer displays the scene using an OpenGL hardware render. When you build a scene, Nuke renders high-quality output from the perspective of the camera connected to the render node. The rendered 2D image is then passed along to the next node in the compositing tree, and you can use the result as an input to other nodes in the script.

Choosing a Render Node

Nuke ships with two render nodes, ScanlineRender and RayRender, which are suited to different tasks. ScanlineRender, as the name suggests, renders row-by-row advancing down the picture. RayRender traces a path from the camera, or virtual eye, to the light source pixel-by-pixel.

**Note:** NukeX and Nuke Studio also include Pixar’s PhotoRealistic RenderMan®, another ray renderer. See PrmanRender for more information.

The ScanlineRender and RayRender nodes have the same inputs and share some of the same controls in their respective Properties panels, but they both have different strengths and weaknesses:

- ScanlineRender generally produces faster results, but is less accurate with reflection and refraction.
- ScanlineRender supports Deep workflows downstream by injecting a **deep** channel into the node tree. See Using ScanlineRender to Generate Deep Data.
- RayRender generally produces very accurate reflection, but at the cost of processing time.
- RayRender does not currently support Deep workflows or render sprite-based particles from NukeX’s Particles system.

As a general rule, if you can afford to wait for a render and don’t need Deep data or particles that rely on sprites, use RayRender.

Rendering Out a Scene

To render a 3D scene into 2D space you need three things: your scene, a render camera, and a render node. When set up correctly, the output of the render node is passed down the node tree for compositing as normal.

To render out a scene:

1. Set up your 3D scene as normal, including geometry, materials, lights, etc.
2. Add a render node and connect the scene to the `obj/scn` input.
3. Make sure the render camera is connected to the render node's `cam` input. See Cameras for more information.
4. Add a background to the `bg` input, if required. You can use the `bg` input to composite a background image into the scene and to determine the output resolution.

**Note:** If the `bg` input is not used, the render node output defaults to the `root.format` or `root.proxy_format` defined in the Project Settings.

5. Toggle the Viewer back to 2D.
6. Connect the output of the render node to the appropriate 2D nodes in your script. See the Nuke Reference Guide for more information on the different render node controls.

### To Add Motion Blur to the 3D Scene

You can use the properties on the MultiSample tab to add motion blur to your 3D scene. For more information, see Adding Motion Blur to the 3D Scene.
Stereoscopic Scripts

The title of this chapter is slightly misleading, as Nuke isn't actually limited to stereoscopic views, but rather provides multi-view support for as many views as you need. The views do not have to be stereo pairs, but since that is the most obvious application, this chapter mainly deals with stereoscopic projects.

Quick Start

In many ways, Nuke lets you work on stereoscopic material just like you would on any other images. However, there are also a few stereo-specific settings and nodes that you need to be aware of when compositing stereoscopic material. The following teaches you how to set up your stereo project, read in and view your images, use the stereo nodes, and render the final output.

Here's a quick overview of the workflow:

1. The first step in working on stereo footage in Nuke is to set up views for them in your project settings (you can open up the project settings by pressing S over the Node Graph). Press Set up views for stereo on the Views tab to do this. For more information, see Setting Up Views for the Script.

2. You can then load your stereo footage into Nuke, either by entering a variable in the file name field or by using a JoinViews node. For more information, see Loading Multi-View Images.

3. In the Viewer, you can select which view to display with the views buttons. You can also display the views side by side or mix them together with the SideBySide and MixViews nodes. For more information, see Displaying Views in the Viewer.

4. Sometimes you need to make changes to one view, while the other one remains as it is. In these cases you need to either split off a view using the View menu or, in case of RotoPaint, use the right-click options in the stroke/shape list to split off views. For more information, see Selecting Which Views Display Changes.

   If you need to perform totally different actions on the two views, you can separate one view for processing with the OneView node. See Performing Different Actions on Different Views.

5. If you have a disparity field available, you can also use the correlate options in the RotoPaint node and the View menu to have changes made to one view automatically reproduced in the other. For more information, see Reproducing Changes Made to One View.

6. With the ShuffleViews node you can rearrange your views, and with the Anaglyph node you can convert your footage into a red and cyan anaglyph footage. For more information, see Swapping Views and Converting Images into Anaglyph.
7. Sometimes you need to re-adjust the convergence, or the inward rotation of your left and right view cameras. This changes the point in the image that appears at screen depth when viewed with 3D glasses. For more information, see Changing Convergence.

8. Finally, you can preview a view of your stereo project by flipbooking it and rendering it out. For more information, see Previewing Stereoscopic Images.
Setting Up Views for the Script

You can import your footage and let Nuke create the views automatically or set up views in advance in the project settings. This allows you to process the individual views separately or both views together, and see the effect of your changes on each view.

If you are likely to need the same views in several projects, you may want to save the views you created in a template.nk script file. For more information on how to do this, see Template Scripts.

Creating Views Automatically

1. Read in your multi-view files as normal.
   A warning dialog is displayed.
   
   ![](warning_dialog.png)

2. Click **Add Views**, **Replace Views**, or **No**:
   - **Add Views** - add the views in the incoming clip to those that exist in the project.
   - **Replace Views** - replace all existing project views with those in the incoming clip.
   - **No** - import the clip and display only the first view in the file, retaining any existing views in the project.

   You can now access the views in your project from the **view** dropdown menu of certain nodes’ controls. You’ll also notice that each view has its own button in the Viewer controls.

Creating and Managing Views Manually

1. Select **Edit > Project Settings** or press **S** in the Node Graph.
2. Go to the **Views** tab. The exiting views are listed in the **views** field.
3. If you want to remove the view called **main** and add views called **left** and **right**, click the **Set up views for stereo** button.

**Note:** If the script is already set up for multiple views, **Set up view for mono** is displayed. Clicking **Set up view for mono** deletes all existing views and adds a single **main** view.

The two new views are assigned colors, red and green by default. To change the colors, double-click on the color field and select another color from the color picker.

If you check **Use colors in UI?**, these colors are used in Node Graph connections, split views indicators on nodes, and Viewer and ShuffleViews node controls to make it easier to differentiate between views.
4. If you want to add other new views, click the + button. A new view is added automatically.
5. To rename the view, double-click its name in the view list and enter the new name.
6. Repeat these steps as necessary until you've got the views you want. You can assign colors to all views by double-clicking the area on the right of the view name.

7. To delete an unnecessary view, select the view from the list and click the - button.
8. To move a view around in the list of views, click the up and down arrows above the views panel.
9. To select the view that's displayed whenever you load the project, set the hero dropdown to the appropriate view.

   You can now access the views in your project from the view dropdown menu of certain nodes' controls. You'll also notice that each view has its own button in the Viewer controls.
If you created many views, you may want them to appear in a dropdown menu rather than as separate buttons in the Viewer and node controls. To switch to using dropdown menus, disable **View selection uses buttons?** on the **Views** tab of the **Project Settings** panel.
Loading Multi-View Images

Once you have set up the views, you are ready to read your images into Nuke. To make things easier, the images you read in should have the view name or the first letter of the view name in the filename, for example `filename.left.0001.exr`, `filename.l.exr`, or `lefteyefilename.0001.cin`.

If you are using `.exr` files, your files can contain both the input for the left eye and the input for the right eye, as `.exr` files support multiple views in a single file. With any other file types, you need to have separate files for the left and right inputs.

To Read Images in

1. Select Image > Read.
2. Navigate to the files containing the images intended for either the left or right eye (or in the case of `.exr` and `.mov` images, both eyes), and select Open.
3. Do one of the following:

   • If the images you want to read in contain a view name or the initial letter of one (for example, `left`, `right`, `l` or `r`) in their file names, replace this with the variable `%V` or `%v` in the file field of the Read node’s controls. Use `%V` to replace an entire view name (for example, `left` or `right`), and `%v` to replace an initial letter (for example, `l` or `r`). When a variable is used, Nuke reads in the missing inputs and combines all inputs into a single output.

   For example, if you read in `image.left.cin` and changed the name to `image.%V.cin`, Nuke would read in both `image.left.cin` and `image.right.cin` with the same Read node, provided that views called `left` and `right` existed in your project settings. Both input images would be combined into a single output.

   Note: Mac and Linux operating systems can be case-sensitive or case-insensitive. If your OS is case-sensitive, you’ll need to make sure you use the correct case when naming your left and right views, as the `%v` variable can only retrieve the case used in the view name.

   You can also use the `%V` and `%v` variables at a directory level. For example, let’s say you have set up views called `testleft`, `testmiddle` and `testright`, and you have the following directories and files:

   mydirectory/testleft/image.testleft.cin
   mydirectory/testmiddle/image.testmiddle.cin
   mydirectory/testright/image.testright.cin

   If you now read in `image.testleft.cin` and changed the pathname to `mydirectory/%V/image.%V.cin`, all three inputs would be read in with the same Read node.
• If the images you want to read in do NOT contain a view name or the initial letter of one (for example, left, right, l or r) in the file names and are not stereo .exr files, insert a Read node for each input and combine them into a single output using the JoinViews node (see below for instructions on how to do that).

• If the images you want to read in are in the stereo .exr file format, you do not need to do anything. However, remember that not all .exr files contain multiple views. If you are using files that are not, follow the instructions in the first two points.

• If the images you want to read in are in the multi-view .mov file format, you need to open the Read node's properties and disable First track only.

You'll notice that the .mov Read node in the Node Graph is now marked with √ to denote multiple views. You can now switch between views using the buttons above the compositing Viewer.

Combining Views from Different Files into a Single Output

1. Select Image > Read to read in your image sequences containing the different views.
2. To insert a JoinViews node, select Views > JoinViews.
3. Connect the inputs of the JoinViews node into the appropriate Read nodes. There should be an input for each view you have created in the project settings. The inputs are labeled with the name of the view.

   If you have assigned colors to the views and checked Use colors in UI? on the Views tab of your project settings, the connecting arrows reflect the view colors. If this does not happen and the arrows are black, you may have connected the inputs the wrong way around. Check that you have connected each Read node to the correct input of the JoinViews node.
Nuke combines the inputs into a single output.
Displaying Views in the Viewer

You can only display the views that exist in your project settings. To see a list of these views, or to add or delete views, select Edit > Project Settings and go to the Views tab. For more information, see Setting Up Views for the Script.

To Display a Particular View

1. Add a Viewer into your script if you haven't already done so.
2. On top of the Viewer controls, do one of the following:
   • If you have checked View selection uses buttons? in the project settings, click the button of the view you want to display. For example, click the right button (assuming you have a view called right in your script).

   ![Viewer button selection example]

   • If you haven't checked View selection uses buttons? in the project settings, select the view you want to display from the dropdown menu.

   ![Viewer dropdown menu example]

Note: Nuke lists the views in .exr files in the order they appear in the clip's header, so a view named 'left' may not always be the first view displayed above the Viewer. If your views do not appear in the correct order, you can rearrange them in the Project Settings > Views tab. See Setting Up Views for the Script for more information.

Tip: You can also press the ; (semicolon) and ’ (forward single quote) keys to move between different views in the Viewer.
To Display Two Views Next to Each Other

1. Add a Viewer into your script if you haven't already done so.
2. If necessary, combine your views into a single output using the JoinViews node. For more information on how to do this, see Loading Multi-View Images.
3. Select Views > Stereo > SideBySide to insert a SideBySide node in an appropriate place in your script.
4. In the SideBySide node's controls, select the two views you want to display from the view1 and view2 dropdown menus. View1 is displayed on the left and view2 on the right.
5. If you want to display one view on top of another rather than next to it, check vertical. View1 is displayed above view2.
6. If you want to swap the views around in the Viewer, click the swap button.
The Viewer displays the two selected views simultaneously, so you can easily compare them.

To Display a Blend Between Two Views

1. Add a Viewer into your script if you haven't already done so.
2. If necessary, combine your views into a single output using the JoinViews node. For more information on how to do this, see Loading Multi-View Images.
3. Select Views > Stereo > MixViews to insert a MixViews node into your script. This node displays a blend between two views in the Viewer, allowing you to check how elements in these views are aligned.
4. In the MixViews controls, use the views buttons or dropdown menus to select the two views to blend between.
5. To control the blend between the views, adjust the **mix** slider. Setting the slider to 0 or 1 displays only one of the views. Values between 0 and 1 produce different blends between the views.
Selecting Which Views Display Changes

By default, Nuke applies any changes you make to all views of the processed node. To apply changes to a particular view only (for example, the left view but not the right), you must first do one of the following:

• In the case of most nodes, split the view off in the node's controls.
• In the case of RotoPaint nodes, select the view you want to process from the view dropdown menu in the node's controls.

These methods are useful, for example, when you want to perform the same operation on both views but use different values for each.

Splitting Views Off

Nuke allows you to split views off in order to apply changes to the existing views separately.

1. Insert a process node (for example, ColorCorrect) in the appropriate place in your script.
2. If you haven't already done so, attach a Viewer to the node. From the Viewer's controls, select the view you want to make changes to.

3. Open the node's controls.
4. Click the view button next to the control you want to adjust. From the menu that opens, select Split off [view name]. For example, to apply changes to a view called left, select Split off left. You can also split all the node's controls by selecting Split all knobs from the right-click menu.
An eye appears on the view button and the node gets a small green dot on it in the Node Graph to indicate that views have been split off.

If you have assigned colors to the views and checked **Use colors in UI?** in your project settings, dots also appear on the node to indicate which views have been split off. For example, if you are using red for the left view and split off that view, a red dot appears on the node.

Any changes you now make using the control in question are only applied to the view you chose to split off. Changes to controls that have not been split off are still applied to all views.

**To Show Separate Values for Each View**

Once you have split off a view, you can apply changes to the existing views separately. Simply click on the small arrow on the left side of a control you have split off. This divides the control so that you can see the values for each view.

Adjusting the split control for only the current view and for all views separately.
To Unsplit Views

1. In the node’s controls, click the view button.
2. From the menu that opens, select Unsplit [view]. For example, to unsplit a view called left, you’d select Unsplit left.
3. Repeat step 2 for all views you want to unsplit.

The view is unsplit, and all changes you made after splitting it off are lost.

Selecting the View to Process When Using the RotoPaint Node

To select the view to process:

1. Open the RotoPaint node’s controls.
2. From the view dropdown menu, select the view you want to process. To apply changes to all views at the same time, select all the views separately.
3. If you selected to process just one view, make sure you are viewing the selected view in the Viewer when making your changes.
Performing Different Actions on Different Views

In case you need to perform totally different actions on the two views, you can add a OneView node to separate one view for processing.

To Extract a View for Processing

1. Select **Views > OneView** to insert a OneView node in an appropriate place in your script.
2. In the OneView node’s controls, select the view you want to make changes to from the **view** dropdown menu.

All views are extracted, and any changes you make are only applied to the view you selected (regardless of which view you are displaying in the Viewer).

To make changes to a different view, select it from the OneView node’s **view** dropdown menu.

To merge views from two separate streams, select **Views > JoinViews** to combine the views (or delete the OneView node from your script).

If you need to extract all views, process them individually, and then merge them together, use the **Split and Join** menu item. This menu item is actually a combination of the OneView and JoinViews nodes. It first extracts all the views you have set up in your project settings and then merges them back together. It’s no different to using several OneView nodes together with a JoinViews node, but speeds up work, because you don’t need to add each node in a separate go. To use the menu item, select **Views > Split and Join**.

For example, if you have created views called **left** and **right** in your project settings and use a **Split and Join** menu item after your Read node, you get the following node tree:
You can then add any necessary nodes, such as color corrections, between the OneView and JoinViews nodes.
Reproducing Changes Made to One View

When rotoscoping, creating paint effects, or doing other operations dependent on image locality, you can have changes made to one view automatically reproduced in the other. This applies to the RotoPaint node, Roto node and any nodes, groups, or gizmos that have controls for x and y coordinates.

To reproduce changes made with the above nodes, groups, or gizmos, you need a disparity field that maps the location of a pixel in one view to the location of its corresponding pixel in the other view. You can create a disparity field using the O_DisparityGenerator plug-in, which is included in the Ocula plug-in set, or a 3D application. Once you have the disparity field, you can store it in the channels of an .exr file or use the ShuffleCopy node to add the disparity channels in the data stream where you need them.

For more information on reproducing paint strokes, Bezier shapes, or B-spline shapes, see Reproducing Strokes/Shapes in Other Views.

Reproducing X and Y Values

Whenever there are values in any x or y control in Nuke for one view, you can automatically generate the corresponding values for the other view. This is true for both nodes and gizmos. For example, you can use a Tracker node to track something in one view, and then have the track's x and y position generated for the other view automatically.

1. Make sure there is a disparity field upstream from the image sequence you are manipulating. If the image sequence is an .exr file, the disparity field can be included in its channels. Otherwise, you can use a ShuffleCopy node or Ocula's O_DisparityGenerator plug-in to add it in the data stream.
2. Insert a node that has an x and y control after the image sequence you are manipulating.
3. Attach a Viewer to the node you added in the previous step, and make your changes in one view.
4. From the View menu next to the x and y controls, select Correlate [view] from [view] using disparity, for example Correlate right from left using disparity. This generates the corresponding x and y values for the other view.
If you have Foundry's Ocula plug-ins installed, you can also select **Correlate [view] from [view]** with Ocula. This way, extra refinements are made when creating the corresponding x and y values, and the results may be more accurate.

5. If you want to adjust the x and y values further, you need to adjust both views independently. Adjustments you make to one view are not automatically generated for the other.
Swapping Views

You can rearrange the views in your script using the ShuffleViews node. For example, you can swap the left and right views around in the pipeline, so that Nuke uses the left input for the right eye and vice versa.

To Rearrange Views

1. Select **Views > ShuffleViews** to insert a ShuffleViews node in an appropriate place in your script.
2. In the ShuffleViews controls, click **add** as necessary.
3. Use the buttons or dropdown menus to select which view to replace with which. For example, to swap the left and right views around, you need to make the following selections:
   • On one row, select **left** under **get**, and **right** under **from** (“get left from right”). The left view is now replaced with the right view.
   • On another row, select **right** under **get**, and **left** under **from** (“get right from left”).
     The right view is replaced with the left view.

If there aren’t enough rows of buttons or dropdown menus on the ShuffleViews node’s properties panel, click the **add** button to add a row.

To remove unnecessary rows in the ShuffleViews node’s controls, click the **delete** button next to the row you want to remove.
Converting Images into Anaglyph

You can use the Anaglyph node to convert your inputs into anaglyph images, which produce a 3D effect when viewed with 2-color anaglyph glasses.

To Convert Your Images into Anaglyph

1. Select Views > Stereo > Anaglyph to insert an Anaglyph node in an appropriate place in your script.
2. Use the views controls in the Anaglyph properties panel to select which views you want to use for the left and the right eye.

   Nuke converts the input images into grayscale anaglyph images. The left input is filtered to remove blue and green, and the right view to remove red.

3. To add color into the images, drag right on the amtcolor slider, or insert a value between 0 (grayscale) and 1 (colored) into the amtcolor input field.

   If the images include areas that are very red, green, or blue, adding more color into them may not produce the best possible results.
4. To invert the colors and use the red channel from the right input and the blue and green channels from the left, check the *(right=red)* box.

5. To control where the images appear in relation to the screen when viewed with anaglyph glasses, enter a value in the **horizontal offset** input field. To have the images appear in front of the screen, you would usually enter a negative value. To have the images appear further away, you would usually enter a positive value. (This is not the case if you have swapped the left and right views around.)

---

**Tip:** If you like, you can register the Anaglyph node as a Viewer Process. This way, you always have it as a viewing option in the Viewer’s Viewer Process dropdown menu and can apply it to the current Viewer without having to insert the node in the Node Graph. Do the following:

1. Create a file called *menu.py* in your plug-in path directory if one doesn’t already exist. For more information on plug-in path directories, see *Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts*.
2. To register the Anaglyph node as a Viewer Process, save the following in your *menu.py*:
   ```python
   ```
4. To apply the Anaglyph Viewer Process, select it from the Viewer Process dropdown menu in the Viewer controls.
5. To adjust the Anaglyph Viewer Process controls, select **show panel** from the Viewer Process dropdown menu.

For more information on Viewer Processes, see *Using the Viewer Controls*. 
Changing Convergence

The ReConverge node lets you shift *convergence* (the inward rotation of the eyes or cameras) so that any selected point in the image appears at screen depth when viewed with 3D glasses. This point is called the *convergence point*. It is the point where the lines of sight from the two cameras meet.

Changing convergence moves the point where the lines of sight from the two cameras meet.

At the convergence point, the different views in the image are aligned and appear at screen depth when viewed with 3D glasses. Anything behind the convergence point appears behind the screen, while anything in front of it seems to pop out of the screen. This is illustrated in the figure below.

Convergence controls where elements in the image appear in relation to the screen when viewed with 3D glasses.
Changing convergence changes the perceived depth of the images. It moves all the elements in the image backwards or forwards a fixed distance while keeping the distance between them the same. This is illustrated in the figure below, where the gray rectangles represent elements depicted in a stereo image.

Changing convergence changes the perceived depth of the images.

Often, the element of an image that appears closest to the audience is used as the convergence point. However, to make an element in your image jump out of the screen, you need to converge on something behind this element.

To calculate the convergence shift, the ReConverge node needs a disparity field that maps the location of a pixel in one view to the location of its corresponding pixel in the other view. To create the disparity field, you can use the O_DisparityGenerator plug-in, which is part of the Ocula plug-in set. Alternatively, you can create the disparity field in a 3D application. Once you have the disparity field, you can store it in the channels of an .exr file or use the ShuffleCopy node to add the disparity channels in the data stream where you need them.

Note that the ReConverge node only shifts views horizontally, not vertically.

**To Change the Convergence Point of a Stereo Image**

1. Make sure there is a disparity field upstream from the image sequence whose convergence you want to change. If the image sequence is an .exr file, the disparity field can be included in its channels. Otherwise, you can use a ShuffleCopy node or Ocula's O_DisparityGenerator plug-in to add it in the data stream.
2. From the Toolbar, select **Views > Stereo > ReConverge** to insert a ReConverge node after the image sequence whose convergence you want to adjust.
3. Attach a Viewer to the ReConverge node.
4. To better view the effect of the ReConverge node, insert an Anaglyph node (**Views > Stereo > Anaglyph**) between the ReConverge node and the Viewer.
5. Make sure the ReConverge properties panel is open. You should see the convergence point overlay in the Viewer. Drag the point on top of the point you want to appear at screen level when viewed with 3D glasses. The convergence shifts to this location. You can also move the convergence point by entering the point's x and y coordinates in the Convergence upon fields.

6. By default, the ReConverge node moves the right view to achieve the convergence shift. However, if you like, you can use the Mode dropdown menu in the ReConverge controls to move the left view instead (select shift left) or move both views equally (select shift both).

7. If necessary, adjust the offset for convergence (in pixels) in the ReConverge controls. To bring all elements of your image forward from the screen level, enter a positive value in the Convergence offset field. To move all elements further away, enter a negative value.

It is also possible to use the same element as the convergence point throughout the image sequence. You can, for example, have the same actor always appear at screen depth. To converge on the same element throughout the sequence, link the ReConverge node with a Tracker node.

Using the Same Convergence Point Throughout a Sequence

1. Insert a Tracker node after the image sequence whose convergence you want to adjust.
2. Track the point that you want to appear at screen level throughout the sequence. For more information on how to use the Tracker node, refer to Tracking and Stabilizing.

3. When you have the track animation data, apply it to the ReConverge node’s Converge upon control via a linking expression. The easiest way to do this is to Ctrl/Cmd+drag the animation button next to the tracks list to the animation button next to the Converge upon control.
Previewing Stereoscopic Images

You can preview stereo images using the Viewer stereo modes or by flipbooking the sequence using Nuke's default flipbook.

Previewing Using the Viewer Stereo Modes

The Viewer stereo modes allow you to see both views at once, in either anaglyph or interlaced per scanline. To enable of Viewer stereo mode, do the following:

1. Right-click in the Viewer to display the context-sensitive menu.
2. Navigate to **Stereo Modes** and select the required mode:
   - **Single** - the default mode, shows only the view selected using the buttons above the Viewer.
   - **Anaglyph** - produces a 3D effect when viewed with 2-color anaglyph glasses.
   - **OpenGL Stereo** - displays both views at once on a 3D monitor for review purposes. See Displaying OpenGL Stereo in Comp Viewers for more information.
   - **Interlace** - shows both views, but alternately per scanline.

![Anaglyph Stereo Mode](image1)

![Interlaced Stereo Mode](image2)

Displaying OpenGL Stereo in Comp Viewers

The Viewer **OpenGL Stereo** mode allows you to see both views at once on a 3D monitor for review purposes. OpenGL Stereo is only supported on NVIDIA Quadro series GPUs and AMD Radeon Pro series GPUs on Windows and Linux OS.

**Note:** OpenGL Stereo mode is not supported on Mac due to limitations in macOS and not supported with AMD GPUs on Linux.
Enabling OpenGL Stereo Output

Windows

To enable NVIDIA GPU stereo output:
1. Right-click on the desktop and select NVIDIA Control Panel.
2. Navigate to **3D Settings > Manage 3D Settings > Stereo - Enable**.

3. Proceed to **Switching to OpenGL Stereo Output**.

To enable AMD GPU stereo output:
1. Double-click the AMD taskbar icon and select **Advanced Settings**.
2. Navigate to the AMD Pro Settings and check **Enable Quad Buffer Stereo**.
3. Select either Auto-Stereo (Horizontal Interleaved) or Auto-Stereo (Vertical Interleaved) and click Apply.
4. Proceed to Switching to OpenGL Stereo Output.

Linux

To enable NVIDIA GPU stereo output:
1. Open a command prompt and enter:
   ```
   nvidia-xconfig --stereo=3
   ```
   **Tip:** For more information on the `nvidia-xconfig` utility, please see the `man` page: `man nvidia-xconfig`

2. Proceed to Switching to OpenGL Stereo Output.

   **Note:** OpenGL Stereo mode is not supported with AMD GPUs on Linux.

**Switching to OpenGL Stereo Output**

1. Right-click in the Viewer to display the context-sensitive menu.
2. Navigate to Stereo Modes and select OpenGL Stereo.
   If you select OpenGL Stereo mode before enabling your GPU settings, a warning is displayed.
The first time you select OpenGL Stereo, a warning message is displayed.

Tip: You can disable the warning by enabling Don't show again and clicking OK.

OpenGL Stereo is displayed in the Comp Viewer.

Flipbooking Stereo Sequences

To flipbook stereo images, do the following:
1. Select the node that you want to flipbook.
2. Select Render > Flipbook Selected from the main menu bar (or press Alt+F).
   You can also press the Flipbook this Viewer button at the bottom-right of the Viewer. This flipbooks the nodes that are connected to the Viewer.
   A dialog opens.
3. Check that your settings are correct in the dialog. The settings in the dialog are almost identical to the settings when you flipbook non-stereo sequences. For more information about these settings, see Flipbooking Sequences.

4. In the **frame range** field, enter the frame range you want to preview (for example, 1-35 or 1-8 10 12-15).

5. Using the **View** control, select the view you want to flipbook, either **left** or **right**.

6. Click **OK**.

   Nuke renders the selected view as a temporary sequence using the frame range and resolution defined in the script's settings. This may take a few moments.

7. After the render is complete, Nuke launches **Flipbook** Viewer and loads in the temporary sequence.

   You can play it back and view it using **Flipbook** Viewer controls.
Rendering Stereoscopic Images

You can render several views using a single Write node. When using the stereo extensions for the .exr file format, Nuke writes the output of both views into a single file. With any other file types, the views are written into their respective files.

Rendering EXR Files

To render .exr files:
1. Select **Image > Write** to insert a Write node in an appropriate place in your script.
2. In the Write node’s controls, select **exr** from the **file type** dropdown menu.
3. From the **views** dropdown menu, select the view(s) you want to render, for example **left, right**.
4. Adjust any other Write controls as necessary and click **Render**. Nuke prompts you for the frames to render.
   Nuke writes the selected views into a single file.

Rendering Other File Formats

To render files that are not in the .exr file format:
1. Select **Image > Write** to insert a Write node in an appropriate place in your script.
2. In the Write nodes’ controls, select the file type of your images from the **file type** dropdown menu.
3. When entering names for the rendered image sequences, you can use the variable `%V` (with a capital V) to represent the words **left** and **right** (or any other full view names) in the file names, for example **filename.%V.####.exr**. To represent the letters **l** and **r** (or the first letters of any views), use the variable `%v` (with a lower-case v) instead. When rendering, Nuke then fills this in with **left, right, l, or r**, and renders all views you specify in the next step.
4. Adjust any other Write controls as necessary and click **Render**. Nuke prompts you for the frames to render as well as the views to execute (assuming you have set up several views in the project settings).

Nuke renders several views, but writes them into separate files. If you did not specify a view in the file names (using either the name of the view, its first letter, or a variable), you can only render one view.
Note: For command line renders, you can pass the `-view` argument with a list of view names to render, separated by a comma. If you do not specify a `-view` argument, Nuke renders all views.
Deep Compositing

Deep compositing is a way of compositing digital images using data in a different format to standard "flat" compositing. As the name suggests, deep compositing uses additional depth data. This reduces need for re-rendering, produces high image quality, and helps you solve problems with artifacts around the edges of objects.

About Deep Compositing

A standard 2D image contains a single value for each channel of each pixel. In contrast, deep images contain multiple samples per pixel at varying depths and each sample contains per-pixel information such as color, opacity, and camera-relative depth.

For example, creating holdouts of objects that have moved in the scene has previously required re-rendering the background, and problems frequently occurred with transparent pixels and anti-aliasing. With Nuke's Deep compositing node set, you can render the background once, and later move your objects to different places and depths, without having to re-render the background. Any transparent pixels, with motion blur for example, are also represented without flaw, so working with deep compositing is not only faster, but you also get higher image quality.

Quick Start

With Nuke's deep compositing node set, you can:
• Read in your deep image with the DeepRead node. See Reading in Deep Footage.
• Merge deep data with the DeepMerge, see Merging Deep Images.
• Generate holdout mattes from a pair of deep images using the DeepHoldout node. See Creating Holdouts.
• Flatten deep images to regular 2D or create point clouds out of them. See Creating Deep Data.
• Sample information at a given pixel using the DeepSample node. See Sampling Deep Images.
• Crop, reformat, and transform deep images much in the same way as you would a regular image, using the DeepCrop, DeepReformat and DeepTransform nodes. See Cropping, Reformatting, and Transforming Deep Images.
Reading in Deep Footage

You read in deep images to Nuke with a DeepRead node, which is rather like reading in any other images with the Read node. Nuke allows you to import deep images in two formats:

- DTEX (generated from Pixar's PhotoRealistic RenderMan® Pro Server).
- Scanline OpenEXR 2.2, or above (tiled OpenEXR files are not supported).
Importing DTEX Files

Before importing DTEX files, you need to set up Pixar's RenderMan Pro Server 20, or earlier, on your computer.

Do the following:

1. Install RenderMan Pro Server on your computer, and set up the necessary environment variables that enable Nuke to work with it. For details, have a look at Setting Up RenderMan Pro Server and PrmanRender. Note that you don't need a RenderMan Pro Server license to work with deep images in Nuke, just installing the software is enough.

2. Create a DeepRead by clicking Deep > DeepRead.

3. Navigate to your .dtex image, and click Open. For more information about the Read node controls, see the Managing Scripts chapter.

4. By default, Nuke tries to automatically detect the .dtex file type by looking at the subimage name. If the name is either Deep Shadow or ends with (or is) deepopacity, Nuke treats the file as a deep opacity file. However, if you have manually changed the subimage name when rendering the file, Nuke may not be able to detect the file type correctly. If this is the case, set the type dropdown menu to one of the following:
   - **deepopacity** - This forces Nuke to treat the file as an accumulated deep opacity file, corresponding to a RenderMan Display Driver configuration of:
     ```
     Display "Filename.dtex" "deepshad" "deepopacity"
     ```
   - **alpha** - This forces Nuke to treat the file as the newer point-sampled alpha or color, corresponding to a RenderMan Display Driver configuration of either:
     ```
     Display "Filename.dtex" "deepshad" "a" or
     Display "Filename.dtex" "deepshad" "rgba"
     ```

Importing Scanline OpenEXR Files

To import scanline OpenEXR files:

1. Create a DeepRead by clicking Deep > DeepRead.

2. Navigate to your deep .exr image, and click Open. For more information about the Read node controls, see the Managing Scripts chapter.

3. By default, the exr prefix is attached to metadata keys to make them distinct from other metadata in the tree. If you'd rather read metadata in "as is" without attaching a prefix, enable do not attach prefix.
Creating Deep Data

You can create deep data in Nuke by:

- sampling a regular 2D image sequence at multiple frames to create several samples for each pixel in a single deep frame. See Converting a 2D Image Sequence to a Deep Frame Using Input Frames.
- converting a regular 2D image to a deep image with a single sample for each pixel at the depth defined by the depth.Z channel. See Converting a 2D Image to a Deep Image.
- recoloring depth samples using a regular 2D color image. See Recoloring Depth Data.
- adding a ScanlineRender node to a 3D scene and connecting a Deep node downstream. See Using ScanlineRender to Generate Deep Data.

Converting a 2D Image Sequence to a Deep Frame Using Input Frames

You can use the DeepFromFrames node to create depth samples from input frames.

1. Connect the DeepFromFrames node to your footage. The deep image is created by placing each frame at increasing depths.
2. To adjust the results, use the controls in the properties panel:
   - `samples` - the number of samples to create per pixel in the output deep image.
   - `frame range` - the range of frames to use for one deep image. For example, with the default samples value (5) and frame range value (1-9) DeepFromFrames sample at times 1, 3, 5, 7 and 9.
   - `premult` - check to premultiply the samples.
   - `split alpha mode` - select `additive` to perform a straight division by the number of samples or `multiplicative` to split the alpha so that it can be returned to its original value if flattened later on (using the DeepToImage node, for example). If you select `additive`, the alpha can't be returned to its original value.
   - `zmin` - the depth to assign to the first sample of each deep pixel output, corresponding to the first frame in the range.
   - `zmax` - the depth to assign to the last sample of each deep pixel output, corresponding to the last frame in the range.
Creating Deep Data | Converting a 2D Image to a Deep Image

A simple setup for creating a deep fog element.

Converting a 2D Image to a Deep Image

Using the DeepFromImage node, you can convert a regular 2D image to a deep image with a single sample for each pixel at the depth defined by the depth.Z channel.

1. Connect DeepFromImage to the footage you want to convert to a deep image.
2. Use the premult input box in the properties panel to select whether you want the input channels to be premultiplied or not.
3. Uncheck the keep zero alpha box if you want to drop any samples with a zero alpha value from the resulting deep image. By default, the box is checked and the resulting deep image contains the zero alpha samples.
4. You can also specify the depth using the z control in the properties panel. In that case, check the specify z box to indicate you don't want to use a depth channel from the input.

Recoloring Depth Data

Use the DeepRecolor node to merge deep buffer files that only contain opacity for each sample with a standard 2D color image. DeepRecolor spreads the color at each pixel of the input 2D image across all the samples of the corresponding pixel in the deep input.

1. Connect your deep source to the depth input of the DeepRecolor node, and your 2D image to the color input. You might want to add an unpremultiply node between your color input and the DeepRecolor if your 2D image is premultiplied.
2. In the properties panel, you can select which channels you want to use from the color input image.
In the example below, DeepRecolor takes an unpremultiplied .exr image and uses it to color the .dtx file's deep samples.

3. If at this point the alpha from the final high-quality flat render doesn't match the alpha represented by the deep samples (for example, as a result of the compression that usually happens to deep files on disk or some change to the shader), you can check target input alpha. This means the color input's alpha is distributed amongst the deep samples, so that the final resulting alpha after flattening of the deep data matches the color input's alpha.

If you leave target input alpha unchecked, Nuke distributes the color to each sample by unpremultiplying by the color image's alpha and then remultiplying by the alpha of each sample. In this case, the alpha from DeepRecolor may not match the alpha from its color input.

Using ScanlineRender to Generate Deep Data

The ScanlineRender node outputs deep data if there is a Deep node downstream.

1. Create a 3D scene and attach a ScanlineRender node to it to render the scene as a 2D image.

   Note: Deep compositing only supports the over blend mode. As a result, if there is a BlendMat node in the 3D scene, its operation always appears to be set to over when converted to Deep.

2. Add a node from the Deep menu downstream from ScanlineRender.
3. If you don't want deep samples with an alpha value of 0 to contribute to the output, open the ScanlineRender properties and make sure that **drop zero alpha samples** is enabled.

4. Adjust the rest of the ScanlineRender properties as usual. For example:
   - If you see any aliasing artifacts in the render, go to the **MultiSample** tab and increase **samples**. This increases the number of deep samples per pixel.
     Alternatively, you can set **antialiasing** to **low**, **medium**, or **high** on the **ScanlineRender** tab.

   ![Samples set to a low value. Samples set to a high value.](image)

   - If you want to add motion blur to your 3D scene, increase the **samples** value to sample the image multiple times over the shutter time.
     In the **shutter** field, enter the number of frames the shutter stays open when motion blurring. If rendering becomes very slow, you can approximate multi-sample rendering and reduce render times by increasing **stochastic samples**.
For more information on the ScanlineRender properties, see Rendering a 3D Scene.

**Tip:** You can use a DeepToPoints node after ScanlineRender to create a 3D point cloud that represents the motion in the scene. For more information on DeepToPoints, see Creating 2D and 3D Elements from Deep Images.
Merging Deep Images

Use the DeepMerge node to merge the samples from multiple deep images, so that each output pixel contains all the samples from the same pixel of each input.

1. Connect the data you want to merge to the DeepMerge node’s numbered inputs.
2. In the DeepMerge properties, make sure operation is set to combine.
3. You can check the drop hidden samples box in the properties panel to not include samples that are completely occluded by nearer samples whose alpha value is one.

![Image of DeepMerge node diagram]

Merging two DeepRecolor results.

4. You can filter out samples using the drop zero threshold control. Increasing the value removes more samples with very small alpha values, such as those caused by floating point inaccuracy.
5. The metadata from control allows you to control which input's metadata is passed down the node tree.

Note: When metadata from is set to All and there are keys with the same name in both inputs, keys in B override keys in A.

6. Enable compute occluded samples if you want Nuke to calculate occlusion using the values of the holdout samples in front of samples from main. This is a more accurate representation of occlusion at depth, but can take longer to process.

For example:

<table>
<thead>
<tr>
<th>Depth</th>
<th>M0</th>
<th>H0</th>
<th>H1</th>
<th>H2</th>
<th>M2</th>
<th>H3</th>
<th>M4</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>INF</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

M = main sample
H = holdout sample

M₀ remains unchanged since there are no holdout samples before it. M₂ is affected by the combined H₀, H₁, and H₂ holdout samples and M₄ is affected by all holdout samples.
Creating Holdouts

You can create holdouts using either the DeepHoldout or the DeepMerge nodes. The primary difference is whether you want the node to produce a flattened output image or a deep output image after the holdout.

Creating a Flattened Holdout with the DeepHoldout Node

The DeepHoldout node removes or fades out samples in the main input that are occluded by samples in the holdout input. The result is a flattened image of your main input image, with holes where objects in the holdout input image have occluded parts of it. If you need a deep output image, then using the holdout operation in the DeepMerge node may be a preferable option.

To create a holdout using the DeepHoldout node:
1. Connect the deep image you want to remove or fade parts from to the main input.
2. Connect the deep image with the occluding parts to the holdout input.
3. You can now view the result, which is a holdout with red, green, blue and alpha channels. Note that the output image is flattened.

![A holdout of blue buildings with ball shapes held out.](image)

4. Enable compute occluded samples if you want Nuke calculate occlusion using the values of the holdout samples in front of samples from main. This is a more accurate representation of occlusion at depth, but can take longer to process.

For example:

```
<table>
<thead>
<tr>
<th>Depth</th>
<th>0</th>
<th>Depth</th>
<th>INF</th>
</tr>
</thead>
<tbody>
<tr>
<td>M0</td>
<td>H0</td>
<td>H1</td>
<td>H2</td>
</tr>
<tr>
<td>M2</td>
<td>H3</td>
<td>M4</td>
<td></td>
</tr>
</tbody>
</table>
```
M = main sample
H = holdout sample

M0 remains unchanged since there are no holdout samples before it. M2 is affected by the combined H0, H1, and H2 holdout samples and M4 is affected by all holdout samples.

5. You can filter out samples using the drop zero threshold control. Increasing the value removes more samples with very small alpha values such as those caused by floating point inaccuracy.

Creating a Deep Holdout with the DeepMerge Node

Like the DeepHoldout node, the holdout operation of the DeepMerge node removes or fades out samples in input B that are occluded by samples in input A. However, unlike the DeepHoldout node, DeepMerge doesn’t flatten the data but produces a deep output image.

To create a holdout using the DeepMerge node:
1. Connect the deep image you want to remove or fade parts from to input B.
2. Connect the deep image with the occluding parts to input A.
3. In the DeepMerge properties, set operation to holdout.
4. You can filter out samples using the drop zero threshold control. Increasing the value removes more samples with very small alpha values, such as those caused by floating point inaccuracy.
5. The metadata from control allows you to control which input’s metadata is passed down the node tree.

![Note: When metadata from is set to All and there are keys with the same name in both inputs, keys in B override keys in A.]

6. Enable compute occluded samples if you want Nuke to calculate occlusion using the values of the holdout samples in front of samples from main. This is a more accurate representation of occlusion at depth, but can take longer to process.

For example:

<table>
<thead>
<tr>
<th>Depth</th>
<th>Depth</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>INF</td>
</tr>
<tr>
<td>M0</td>
<td>H0</td>
</tr>
<tr>
<td>H1</td>
<td>H2</td>
</tr>
<tr>
<td>M2</td>
<td>H3</td>
</tr>
<tr>
<td>M4</td>
<td></td>
</tr>
</tbody>
</table>

M = main sample
H = holdout sample

M0 remains unchanged since there are no holdout samples before it. M2 is affected by the combined H0, H1, and H2 holdout samples and M4 is affected by all holdout samples.
You can now view the result, which is a holdout with red, green, blue, and alpha channels. Note that the output image is still a deep image.
Creating 2D and 3D Elements from Deep Images

You can create a 2D image or a 3D point cloud from a deep image.

Creating a 2D Image from a Deep Image

You can use the DeepToImage node to flatten an image, in other words merge all the samples in a deep image into a regular 2D image.

1. Connect the node to a deep image (or a DeepMerge with merged deep data) you want to flatten.
2. In the properties panel, the volumetric composition box is checked by default, but if you uncheck it, Nuke only calculates the front depth of each sample and assumes the samples do not overlap. If you uncheck this, the calculation takes less time, but if you have overlapping samples in your deep image, the resulting image might not represent every pixel as expected.

Creating a Point Cloud from a Deep Image

You can use the DeepToPoints node to transform the deep pixel samples into points in 3D space that you can see in Nuke's 3D view, much like a point cloud. This node is useful for position reference.

1. Connect the DeepToPoints node's deep input to the deep image you want to view in 3D. If you have a camera that you want to look at the point cloud through, connect it to the camera input.
2. Change to 3D view (by pressing Tab) to view the results.
3. In the properties panel, you can use the Point detail slider to adjust the density of the cloud.
   
   ![Point detail comparison](image)
   
   **Point detail** set to 0.005.  
   **Point detail** set to 0.05.
4. Adjust Point size to change the size of the points. You can also use 3D object selection on the point cloud, and for example snap on your DeepToPoints results. (For more information on 3D selection, see 3D Selection Tools.)
Point size set to 2.

Point size set to 6.
Modifying Deep Data

Nuke allows you to color correct deep images as well as modify them using expressions.

Color Correcting Deep Images

The DeepColorCorrect node applies the color correction to each sample at each pixel.

There are control sets for adjusting shadows, midtones and highlights, as well as a master set for adjusting all these at once. You can use the lookup curves on the Ranges tab to control the range of the image that is considered to be in the shadows, midtones, and highlights. For more information about the basics of color correcting, see Working with Color.

Tip: Make mattes by setting the gain value for your alpha channel to 0 and setting the offset value for the alpha channel to 1 in the range you want a matte for.

Adjusting the Effect of Deep Color Correction

On the Masking tab you can set the point among the deep samples where the effect of your color correction starts and finishes.

1. Check the limit_z box to activate the zmap tool.
2. Adjust the trapezoid so that the A delimiter marks the depth where you want the color correction to start, B and C mark the length of the full effect and delimiter D indicates where the effect stops. Zmap tool's y axis, therefore, indicates the amount of the effect, and the x axis is the range of your depth samples.
3. Use the mix control to adjust the overall mixing between the color corrected result and the original image. Zero value is the original image, and value 1 is the full color correction result.

Modifying Your Deep Images with Expressions

You can use the DeepExpression node to run Nuke expressions on deep data. Use the controls in the properties panel:

1. There are four fields for temporary expressions, just like in the normal Expression node. These can be useful if you need to use a long expression in several fields and want to assign that
expression temporarily to a variable. Enter the variable name to the left of the equals (=) sign, and the expression to the right. You can then use the variable name to represent the entire expression in the expression fields for the channels.

2. In the **chans0 - chans3** dropdowns you can then specify which channels you want to create expressions for. This adds or removes expression fields below.

3. You can then enter your expressions for the different channels in the channel expression fields.

For more information about expressions, have a look at **Expressions**.
Cropping, Reformatting, and Transforming Deep Images

You can crop, reformat and transform deep images much in the same way as you would a regular image, using the corresponding deep nodes.

**Note:** Remember that since the samples at each pixel can be located at arbitrary depths, resampling during transforming may produce unexpected results since there may not be samples at the same depth in adjacent pixels.

### Cropping Deep Images

You can use the DeepCrop node to clip your deep image, much like the normal Crop node:

1. Connect the DeepCrop node to the deep image you want to crop.
2. Adjust the crop box in the Viewer in X and Y directions to define your crop area. Alternatively, define your crop area using the `bbox` fields in the properties panel. If you want to keep the depth samples outside the crop box, you can check the **keep outside bbox** box.
3. Use the `znear` and `zfar` controls in the properties panel to crop samples in depth. If you don’t want to use either of these controls, you can disable them by unchecking the **use** box next to them. If you want to keep your depth samples outside of the z range defined by these controls, you should check the **keep outside zrange** box.

### Reformatting Deep Images

DeepReformat is the Reformat node for deep data. You can use it to set your deep image’s dimensions, scale, and so on. To reformat your deep image:

1. Connect the DeepReformat node to the deep image you want to resize.
2. In the **type** dropdown, select:
   - **to format** - sets the output width and height to the selected format. Select the format in the **output format** dropdown. If the format does not yet exist, you can select **new** to create a new format from scratch. The default setting, **root.format**, resizes the image to the format indicated on the **Project Settings** dialog.
Cropping, Reformatting, and Transforming Deep Images

- **to box** - sets the output width and height to dimensions you define in pixels. Enter values in the **width**, **height** and **pixel aspect** fields to specify the dimensions.
- **scale** - sets the output width and height to a multiple of the input size. Use the **scale** slider to define the factor. The scale factor is rounded slightly, so that the output image is an integer number of pixels in the direction selected under **resize type**.

3. You can specify what kind of resize you want in the **resize type** dropdown. Select:
   - **none** - to not resize the original.
   - **width** - to scale the original until its width matches the output width. Height is then scaled in such a manner as to preserve the original aspect ratio.
   - **height** - to scale the original so that it fills the output height. Width is then scaled in such a manner as to preserve the original aspect ratio.
   - **fit** - to scale the original so that its smallest side fills the output width or height. The longest side is then scaled in such a manner as to preserve the original aspect ratio.
   - **fill** - to scale the original so that its longest side fills the output width or height. The smallest side is then scaled in such a manner as to preserve the original aspect ratio.
   - **distort** - to scale the original so that both sides fill the output dimensions. This option does not preserve the original aspect ratio, so distortions may occur.

4. Check the **center** box to define whether the input pixels should be resampled to the new size or centered in the output. If you don’t center, the lower left corners of the input and output are aligned.

5. To further adjust your image’s layout, you can check the respective boxes for:
   - **flip** - to swap the top and bottom of the image.
   - **flop** - to swap the left and right of the image.
   - **turn** - to turn the image 90 degrees.
   - **black outside** - to set pixels outside the format black.
   - **preserve bounding box** - to preserve pixels outside the output format rather than clipping them off.

### Transforming Deep Samples

You can use the DeepTransform node to reposition the deep samples.

1. Connect the node to the deep footage you want to transform.
2. Use the **translate x**, **y**, and **z** controls to translate your samples.
3. Scale the samples’ **z** depth using the **zscale** control. Values above 1 increase the depth, whereas values below 1 decrease it.
4. If you connect a mask to the node’s **mask** input, you can use it to regulate how much of an effect the depth transformation has in different parts of the frame.
Sampling Deep Images

You can use the DeepSample node to sample any given pixel in a deep image. The Deep Sample node gives you the depth data as figures.

1. Connect the DeepSample node to another Deep node.
2. Position the pos indicator over the pixels you want to sample in the Viewer.
3. View the deep sample information in the sample table on the DeepSample properties panel.
4. You can also toggle the accumulate box to select whether you want to see the individual sample values of the sample pixel (unchecked), or the final composited value (checked).
Writing Deep Data

You can write out deep images in the scanline OpenEXR 2.2, or above, format using the DeepWrite node, which shares a lot of controls with the standard Write node. Do the following:

1. Select Deep > DeepWrite to insert a DeepWrite node into your script.
2. In the properties panel, click the file or proxy field’s folder icon and browse to the directory where you want to store the deep image.
3. After the path, type a name for the deep image, including the .exr extension, and then click OK. If you’re rendering an image sequence, include the frame number variable (for example, ####) in the name.
4. Use the datatype dropdown menu to select the bit depth for the rendered file: 16 bit half or 32 bit float.
5. Set compression to the compression type to apply to the rendered file.
6. From the metadata dropdown menu, select what metadata is included with the rendered file:
   - no metadata - No custom attributes are created, and only metadata that fills required header fields is written out.
   - default metadata - The optional time code, edge code, frame rate, and exposure header fields are also filled using metadata values.
   - default metadata and exr/*
   - all metadata except input/*
   - all metadata
7. By default, unknown metadata keys have the prefix nuke attached to them when they are written into the file. If you’d rather have them written into the file “as is”, without the prefix, check do not attach prefix.
8. Adjust the rest of the controls as necessary. For more information on them, see Output (Write) Nodes.
9. Click the Render button.
   Nuke prompts for a frame range, defaulting to the range in the frame range fields.
10. If necessary, change the start and end frames, and then click OK.
    Nuke writes the deep data to a scanline OpenEXR 2.2, or above, file (tiled OpenEXR files are not supported).
Working with File Metadata

The Read node's **Metadata** tab and the nodes in the **Metadata** menu of the Toolbar let you work with information embedded in your images. This section gives instructions on their usage, teaching you to view, compare, edit, and render metadata.

Metadata

Metadata is a set of information about an image embedded in the image file. This information may include the image's original bit depth, width, and height, for example. It can be attached to the file by the camera used to shoot the images, and/or edited later.

When Nuke loads an image, it reads in the metadata embedded in the image. The metadata is then passed down the node tree so you can view and use it at any point in your script. For example, you can reference metadata via expressions. You can also edit or delete existing metadata, add new metadata to a file, and write the resulting metadata out to files.

**Note:** Metadata for QuickTime files does not show gamma or bit depth.

**Note:** The Read and Write node's **timecode** and **edge code** fields have been removed from the Properties panel, but the metadata is still available using the **input/timecode** and **input/edgencode** keys. See Viewing Metadata for more information.

**Tip:** When using a Merge node, you can choose which input's metadata to pass down the tree. In the Merge controls, set **metadata from** to either **A** or **B**.

As well as the **Metadata** tab in the Read node, the **MetaData** menu of Nuke's Toolbar features five nodes that help you work with file metadata:

- ViewMeta data lets you inspect the metadata passed down by the input node. See Viewing Metadata.
- CompareMeta data lets you compare the metadata between two inputs and view the differences. See Comparing Metadata Between Inputs.
- ModifyMeta data lets you edit existing metadata in the input stream, add metadata into the stream, and delete metadata from the stream. See Modifying Metadata.
• CopyMetaData lets you copy metadata from one input to another and filter metadata to exclude some of it. See Copying and Filtering Metadata Between Inputs.
• AddTimeCode lets you add a timecode to the metadata passed down by the input node. See Adding a Time Code to Metadata.
Viewing Metadata

The simplest way to view file metadata is by clicking the Metadata tab in the Properties panel of a standard Read node. All the available metadata is displayed, along with a simple search function.

To filter the lists of metadata, use the search metadata for field. For example, if you enter f in the search metadata for field, only the keys and values that include the letter f are displayed. By default, the search is done within both keys and values. If you want to limit the search to either keys or values only, set within to keys only or values only. For example, you can view metadata specific to the input's file format by entering the file format (for instance, dpx/) in the search metadata for field and setting within to keys only.

Note: When observing the creation time (input/ctime) of an image, Windows generally differs from Linux and Mac. This is due to the different way in which the operating systems deal with file creation times.

Once you know which keys exist on the input, you can reference them in expressions. See Accessing Metadata Using Tcl Expressions.

You can also view metadata using the ViewMetaData node:

1. Select MetaData > ViewMetaData to insert a ViewMetaData node after the node whose metadata you want to inspect.
2. In the ViewMetaData properties, you can see a list of the metadata embedded in the image. This is divided into keys and their values.
Comparing Metadata Between Inputs

To compare metadata between two inputs:

1. From the Toolbar, select **MetaData > CompareMetaData** to add a CompareMetaData node after the two nodes whose metadata you want to compare.
2. Connect the nodes you want to compare to the A and B inputs of the CompareMetaData node.
   
   A list of keys where there is a difference between the two inputs is shown in the CompareMetaData properties.
Modifying Metadata

There are several ways to modify metadata in Nuke.

**To Add Metadata**

1. Select `MetaData > ModifyMetaData` to insert a ModifyMetaData node after the node whose metadata you want to add a new key to.
2. In the ModifyMetaData controls, click on the plus (+) button. A placeholder appears in the metadata box.
3. Double-click on the placeholder under `key`.

   The **Pick metadata key** dialog opens.

4. In the field at the bottom of the dialog, enter a name for the new key you want to add to the metadata. Click **OK**.

5. Double-click on the placeholder under `data` and enter a value for the new key.
   
   The new key and its value are added to the metadata being passed through.

**To Edit Metadata**

1. Select `MetaData > ModifyMetaData` to insert a ModifyMetaData node after the node whose metadata you want to edit.
2. In the ModifyMetaData controls, click on the plus (+) button.
A placeholder appears in the metadata box.

3. Double-click on the placeholder under **key**.
   The **Pick metadata key** dialog opens.

4. Pick the key whose name or value you want to edit and click **OK**.
   The key is added to the ModifyMetaData properties.

5. In the ModifyMetaData properties, double-click on the key or its value and edit the information as required.

### To Remove Metadata

1. Select **MetaData > ModifyMetaData** to insert a ModifyMetaData node after the node whose metadata you want to edit.

2. In the ModifyMetaData properties, click on the plus (+) button. A placeholder is added to the metadata list.

3. Double-click on the placeholder under **action** and select **remove** from the menu that opens.

4. Double-click on the placeholder under **key**.

   ![ModifyMetaData node with remove option selected]

   The **Pick metadata key** dialog opens.

5. From the list of existing keys, select the key you want to remove and click **OK**.
To Edit the List of Actions in the ModifyMetaData Properties

• To perform a new action, click on the plus (+) button.
• To cancel an existing action, select it from the list and click on the minus (-) button. Note that this only affects the ModifyMetaData actions, and does NOT delete keys from the metadata embedded in the input image.
• To move an item up in the list, select it and click on the up arrow button.
• To move an item down in the list, select it and click on the down arrow button.
Copying and Filtering Metadata Between Inputs

To copy metadata from one input to another and/or filter metadata:

1. Select **MetaData > CopyMetaData** to insert a CopyMetaData node into your script.
2. Connect:
   - the **Image** input to the node whose image you want to pass down the tree.
   - the **Meta** input to the node whose metadata you want to copy to the output.
3. Set **metadata from** to one of the following:
   - **Image+Meta** - to add the metadata from the **Meta** input to the metadata from the **Image** input. If the inputs share any common metadata keys, the values taken from the Meta input override those taken from the Image input.
   - **Meta only** - to only use the metadata from the Meta input.
   - **Meta+Image** - to add the metadata from the **Image** input to the metadata from the **Meta** input. If the inputs share any common metadata keys, the values taken from the Image input override those taken from the Meta input.
   - **Image only** - to only use the metadata from the Image input. This produces the same result as not using a CopyMetaData at all: both the image and metadata are taken from the Image input. However, this option can be useful if you want to filter the metadata to exclude some of it (see the next step for how to do this).
4. To filter the metadata taken from the inputs, use the **copy only** fields under **Meta filtering** and/or **Image filtering**. For example, if you enter **f** in the **copy only** field under **Meta filtering**, only the keys and values that include the letter **f** are copied from the Meta input. By default, the search is done within both keys and values. If you want to limit the search to either keys or values only, set **within** to **keys only** or **values only**. For example, you can copy metadata specific to the input's file format by entering the file format (for instance, **dpx/**) in the **copy only** field and setting **within** to **keys only**.
Adding a Time Code to Metadata

1. Select **MetaData > AddTimeCode** to insert an AddTimeCode node into your node tree. A time code is added to the metadata being passed through. By default, the time code is 01:00:00:00 on the first frame. It is updated throughout the frame range according to the input clip’s playback speed, which in turn is controlled by the **fps** (frames per second) parameter in the Project Settings. If you change the **fps** value in the Project Settings, the time code in the metadata is updated to reflect the change.

![AddTimeCode Node](image)

Nuke can also deal with drop frames, such as when a clip’s frame rate is 29.97 or 59.94. Instead of the default HH:MM:SS:FF time code format, use the format HH;MM;SS;FF, delimited by ; (semicolon).

**Note**: Using semicolon delimiters with non-drop frame time codes displays an error in the Viewer.

2. If you don't want the time code on the start frame to be 01:00:00:00, enter a new time code in the **startcode** field.
3. If you want to specify the playback speed manually rather than get it from the metadata and project settings, uncheck **get FPS from metadata** and enter a new value in the **fps** field.
4. If you want to specify a different start frame than the first frame, check **use start frame?** and enter a new value in the **start frame** field.

If you want to display the time code on the image, insert a Text node after the AddTimeCode node and enter **[timecode]** in the **message** field. For more information on referencing metadata via expressions, see **Accessing Metadata Using Tcl Expressions**.
Rendering Metadata

When rendering with the Write node, Nuke lets you write out metadata into the following file formats: .exr, .cin, .dpx, and .jpg. You cannot write out metadata into any other formats.

When rendering metadata into an .exr file, you can use the metadata dropdown menu in the Write node controls to specify what to write out:

- **no metadata** - Do not write out any metadata, except for metadata that fills required header fields (for example, file name and bbox).
- **default metadata** - Write out the time code, edge code, frame rate, and exposure.
- **default metadata and exr/** - Write out the time code, edge code, frame rate, exposure, and anything in exr/.
- **all metadata except input/** - Write out all metadata, except anything in input/.
- **all metadata** - Write out all metadata.

When rendering any other file format than .exr, Nuke writes whatever metadata the file format header is known to support. Therefore, what is written out varies according to the file format.
Accessing Metadata Using Tcl Expressions

You can access metadata via Tcl expressions in the following ways:

• To get a list of all keys in the incoming metadata, use the expression `[metadata]`. For example, if you add a Text node after an image and enter `[metadata]` in the `message` field, a list of all the keys in the incoming metadata appears on the image. The values of the keys are not displayed.

To get a list of all keys and values, use the expression `[metadata values]`.

• To get the value of a particular key in the incoming metadata, use the expression `[metadata key]`. Replace `key` with the name of the key whose value you want to use. For example, to display the name and location of the image file on the image, add a Text node after the image and enter `[metadata input/filename]` in the `message` field.

• To get a filtered list of keys in the incoming metadata, use the expression `[metadata keys filter]`. Replace `filter` with whatever you want to use to filter the list. You can use asterisks (*) as wildcards in your filter to substitute zero or more characters in the key names. For example, to get a list of all the keys with the letter `f` in them, use the expression `[metadata keys *f*]`. To get a list of all the keys starting with `input/`, use the expression `[metadata keys input/*]`.

By default, the keys are listed on separate lines. To change this, you can use `-s "separator"` to have the keys separated by a separator of your choice. Replace `separator` with whatever you want to appear between the different keys. For example, to get a list of all the keys starting with `input/` and separated by spaces, you can use `[metadata -s " " keys input/*]`. To get the same list separated by commas, use `[metadata -s ", " keys input/*]`.

By default, if you attempt to access metadata that does not exist in the stream, Nuke returns an empty string. To make this error instead, use the `-e` flag before other parameters.

For more information on using expressions, see the Expressions chapter.
Accessing Metadata Using Python

You can also access metadata using the Python programming language. For more information, see the Nuke Python documentation (Help > Documentation).
Audio in Nuke

In many compositing projects it's vital to be able to key visual changes to cues on the audio track that goes with the picture. You can use Nuke's AudioRead node to read in an audio file, view it in the Curve Editor and Dope Sheet in order to line up keyframes of your composition with the waveform of the sound. You can then flipbook the audio with your footage to preview your comp with sound.

Quick Start

You can load audio files into Nuke using the AudioRead node, in much the same way as you read in images with the Read node. You can read in uncompressed .wav and .aiff files and flipbook them with your footage for playback.

Here's a quick overview of the workflow:
1. Read in an audio file. See Reading Audio Files into the Node Graph.
2. Display an audio waveform for your audio clip and access its animation curve in the Curve Editor or the Dope Sheet. See Creating and Editing Audio Curves.
3. When you're done, you can flipbook your script to view and listen to the results. See Flipbooking the Audio Track.
Reading Audio Files into the Node Graph

You can drag and drop audio clips from the Project tab to the Node Graph, if the clip is already in Nuke.

Otherwise, use the AudioRead node to read in an audio file:

1. To create an AudioRead node, click Other > AudioRead in the Nuke Toolbar.
   The AudioRead node doesn't have to be connected to other nodes.

Simple AudioRead node setup.
Tip: You can also load an audio file by creating a normal Read node and navigating to a supported audio file.

2. In the AudioRead properties, use the file control to navigate to the audio file you want to read in. You can read in uncompressed .wav and .aiff files.
3. Use the time range fields to enter the start and end times in seconds for the audio in Nuke.
4. In the file time range fields, enter the start and end times in seconds of the audio file read in. These are automatically set to the values in the file, but you can change them to trim the data used in Nuke.
5. If you want to discard your changes and reload the audio file, click reload.
6. Use the ratesource menu to select the source for the sample rate:
   • file - reads the rate from the audio file.
   • custom - lets you specify a custom sample rate in the rate field.
Creating and Editing Audio Curves

Once you have read in an audio file (see Reading Audio Files into the Node Graph), you can display an audio waveform for your audio clip and access its animation curve in the Curve Editor or the Dope Sheet.

Creating a Keyframe Curve

In the curves section of the AudioRead properties panel, you can generate curves out of the audio data:

1. Set the keyframe interval you want to use when creating the curves in the key interval field. For example, if you enter 3, keyframes are created to every third frame of the input footage.
2. Click generate to generate the audio data as a curve that you can use in the Curve Editor and Dope Sheet.
3. View the left and right stereo levels on the current frame in the left and right fields and adjust if necessary. Any changes are reflected on the curve automatically.

Modifying the Audio Curve in the Curve Editor and Dope Sheet

When you're working with your audio curve in the Curve Editor or Dope Sheet, there are a few right-click options that you can use to adjust how the clip's waveform displays:

1. Right-click in the Curve Editor or Dope Sheet and select View > Audio.
2. Then select Source and check the box for either ProjectDefault or an AudioRead node depending on which one you want to view. If you've only got one AudioRead, it is the project default.
3. If you're working with a stereo clip with more than one audio channel, you can select your audio channel by ticking the appropriate box under Channel.
4. Select a style in which you want your waveform to be drawn by selecting one of the DrawStyle options:
   • Off - to draw no audio waveform.
   • Behind - to draw a waveform behind the animation curves.
   • Below - to draw a waveform below the animation curves.
Audio waveform in the Curve Editor.
Flipbooking the Audio Track

When you're done, you can proceed to flipbooking your results:

1. Click the *flipbook this Viewer* button in the Comp Viewer.
2. In the *Flipbook* dialog, select the AudioRead file you want to use in the *Audio* dropdown.
3. Click *OK*. View and listen to your clip in Flipbook.
Previews and Rendering

Nuke supports a fast, high-quality internal renderer, with superior color resolution and dynamic range without a slowdown in the workflow.

About Rendering in Nuke

These are some of the key features of Nuke's rendering engine:

• Multi-threaded rendering to take advantage of multiple processors in its calculations.
• Scanline (as opposed to buffer-based) rendering allows you to immediately see portions of render output.
• Calculations performed with 32-bit precision, using linear light levels.

This chapter teaches you how to use the renderer's various features to preview a script's output and generate its final elements. You'll also learn how to preview using Flipbooking and Capture, and check output on an external broadcast video monitor.

Quick Start

Here's a quick overview of the workflow:

1. With the Viewer, you can preview your footage and use the ROI button to focus on a particular part of it. For more information, see Previewing Output.
2. You can then flipbook your clip. A quick way of doing this is to click the Flipbook button in the Viewer, set the frame range and other settings in the Flipbook dialog, and click OK to flipbook and automatically launch the Flipbook Viewer. See Flipbooking Sequences.
3. Save out low resolution .jpg sequences using Capture to share your work, such as for peer review purposes. Click the Capture button in the Viewer, set the frame range and other settings in the Capture dialog, and click OK.
4. If you've read in an audio clip with the AudioRead node, you can flipbook that with your footage just by selecting the right AudioRead node in the Audio dropdown. For more information, see Audio in Nuke.
5. To check the final result in correct video colorspace and pixel aspect ratio, preview your footage on an external broadcast video monitor. See Previewing on an External Broadcast Video Monitor.
6. When you're happy with your preview results, you can render out your clip. To do this, you need to connect at least one Write node to your clip, and then set the render properties in the properties panel. You can specify your render format in the `filename` field, and use the `frame` control to offset your frame numbers if necessary. For more information, see Output (Write) Nodes.

7. If you have more than one Write node connected to your node tree, you can render out all of them, or select the ones you want to render. You can then click `Render > Render all` or `Render selected` in the menu bar to start the renders. For more information, see Output (Write) Nodes.
Previewing Output

This section explains how to preview individual frames in a Nuke Viewer window (see Previewing in a Nuke Viewer), how to render a flipbook for a sequence of frames (see Flipbooking Sequences), and how to preview output on an external broadcast video monitor (see Previewing on an External Broadcast Video Monitor).
Previewing in a Nuke Viewer

When you connect a Viewer to a given node's output (by selecting the node and pressing a number key), Nuke immediately starts rendering the output in the Viewer using all available local processors.

Keep in mind the following tips in order to speed up this type of preview rendering:
• First, if you don't need to evaluate the whole image, zoom into the area of interest. Nuke then renders only the portion of scan lines visible within the Viewer.
• Alternatively, you can use the Viewer's region of interest (ROI) feature to render only a portion of the image, while seeing that result in the context of the whole image.

To Enable the ROI Render Feature
1. Press $\text{Alt+W}$ over the Viewer. The Viewer’s $\text{ROI}$ button turns red, indicating that the feature is enabled.
2. Drag on the Viewer to draw the region of interest. The Viewer now renders only the pixels within the region.

To Edit the Position or Size of Current ROI
1. Click the $\text{ROI}$ button so that it turns red. The overlay for the current ROI appears in the Viewer.
2. To reposition the ROI:
   Using the crosshair in the middle of the ROI, drag the ROI to the desired location.
3. To resize the ROI:
   Drag any corner or side of the ROI until you achieve the desired size.

To Disable the ROI Render Feature
Click the Viewer's $\text{ROI}$ button. It turns gray, signaling that it is off. The Viewer now renders all of the visible image.
Flipbooking Sequences

Flipbooking a sequence refers to rendering a range of images (typically at proxy resolution), then playing them back in order to accurately access the motion characteristics of added effects.

You have a few options for flipbooking within Nuke:

- You can enable automatic disk caching of rendered frames, then play these frames back using Nuke's native Viewer. This option does not let you define a specific playback rate.
- You can render out temporary image sequences using the default flipbooking tool, a RAM-buffering playback utility, which is displayed in its own Viewer and plays back sequences at the defined frame rate.

If you have purchased a license for HieroPlayer, you can flipbook using HieroPlayer rather than the default flipbook that ships with Nuke.

- You can also set up an external flipbooking application in Nuke using Python. For more information, see the Nuke Python documentation (Help > Documentation).

The Nuke Viewer automatically saves to disk a version of every frame it displays. When you play through sequences in the Viewer, it reads, where possible, from this cache of pre-rendered images, making real-time play back possible (depending, of course, on image resolution and your hardware configuration). You can define the location and size of the Viewer cache in the Preferences.

Depending on what viewer buffer bit depth has been set to in the Viewer settings, the cache can contain 8-bit (byte), 16-bit (half-float), or 32-bit (float) image data. This offers a trade-off between speed and quality. Half-float and float modes provide higher precision than byte, but are also slower to process.

Setting the Viewer Cache Location and Size

1. Click Edit > Preferences to display the Preferences dialog.
2. In the Performance > Caching section, there is a Disk Caching – temp directory field. Use this to enter the path name of the directory in which you want to store the flipbook images (for example, c:/temp).
3. Using the comp disk cache size control, specify the number of gigabytes you want to allow the image cache to consume.
4. Click OK in the bottom-right corner of the Preferences dialog to update preferences and then restart Nuke.

The Viewer now caches each frame it displays in the directory specified. When you click the playback buttons on the Viewer, or drag on the scrub bar, Nuke reads in images from this cache.
Note that the cached images have unique names reflecting their point of output location in the script. This means that you can cache images from multiple nodes in the script without overwriting previously cached images. For more information on caching, see Image Caching.

To Use HieroPlayer as Nuke's Flipbook

If you have purchased a license for HieroPlayer, you can flipbook using HieroPlayer rather than the default flipbook that ships with Nuke:

1. Copy the `flipbook_hiero_launch.py` and `hiero_flipbook.py` file from:
   `<installation_folder>/pythonextensions/site-packages/hiero/examples/`
to your `.nuke` folder.
2. Launch Nuke and open the Script Editor.
3. Type:
   ```python
   import hiero_flipbook
   ```
   and press `Ctrl/Cmd+Return`.
4. Open the required comp and launch the flipbook by clicking the ▶ icon under the Viewer or by pressing `Alt+F` on a selected node.
5. Select HieroPlayer from the Flipbook dropdown and proceed as described below.

Flipbooking a Sequence

To flipbook an image sequence, do the following:

1. Select the node that you want to flipbook the output of.

   **Note:** If you select a Write node in the step above, you must first click its Render button in order to manually render its output to the destination defined in the file field. This step is necessary only in the case of Write nodes.

2. Select Render > Flipbook selected (or press `Alt+F`).
   Alternatively, you can click the Flipbook this Viewer button ▶ at the bottom-right of the Viewer.
   
   This flipbooks the nodes that are connected to the Viewer.
   A Flipbook dialog opens.
3. Check that your settings are correct in the dialog. The default values are copied from the Viewer you currently have active. You can change them if necessary:
   • Flipbook - set the flipbooking application you want to use.
   • Take settings from - set which Viewer should be used to draw default values from.
• **Enable ROI** - Check to define your region of interest.
• **Channels** - select which layer to display in the flipbook result.
• **Frame range** - set the frame range you want to flipbook.
• **Use proxy** - check to use proxy mode.
• **Render in background** - check to render in the background. If you check this, you can also set **Thread limit** and **Memory limit** controls. The former limits the number of threads that Nuke uses in the background and the latter limits the amount of cache memory that Nuke uses.

**Note:** If you're rendering multiple sequences in the background, this can take up more than the total RAM on your machine. When running background renders of any type, you need to make sure they don't require more RAM all together than what's available on the machine, otherwise you may experience problems such as hanging.

• **Delete existing temporary files** - Check to delete any existing temporary files with the same file name before flipbooking.
• **LUT** - select the LUT appropriate for viewing. By default, the flipbook renders your files with a linear colorspace and attempt to pass a LUT file to the flipbook.
• **Burn in the LUT** - If you check this box the flipbook files are rendered with the LUT applied. If you uncheck it, the flipbook is displayed using it's equivalent LUT (based on the LUT's name). If you have an equivalent LUT available in the flipbook program, then it's better not to check the **Burn in the LUT** box. This way, when you measure pixel values in the flipbook application they match what you get in the Nuke Viewer.
• **Audio** - if you want to flipbook an audio file with your clip, select the AudioRead node you need in this dropdown. For more information on audio files in Nuke, see [Audio in Nuke](#).
• **Continue on error** - check to keep rendering even if an error occurs during the process.
• **Views** - check the view(s) to output, if you're working in a multi-view comp.

**Note:** The **Views** control is only available in multi-view comps. See [Stereoscopic Scripts](#) for more information.

• **Buffer** - set which buffer you want include.

4. Click **OK**.

Nuke renders as a temporary sequence the output of the selected node using the frame range and resolution defined in the script's settings. This may take a few moments.

After the render is complete, Nuke launches Flipbook Viewer and loads in the temporary sequence. You can play it back and view it using Flipbook Viewer controls.
If you flipbooked a stereo comp, you can right-click in the Flipbook and choose **Stereo Modes** to view your flipbook in various configurations:

- **Side by Side** - displays the views side by side at the correct aspect ratio, and adds selection controls above the Viewer.
- **Squeezed Side by Side** - displays the views side by side and squeezed to fit the format horizontally, and adds selection controls above the Viewer.
- **Squeezed Above by Below** - displays the views above and below each other and squeezed to fit the format vertically, and adds selection controls above the Viewer.
- **Interlace H** - displays the views interlaced horizontally, and adds selection controls above the Viewer.
- **Interlace V** - displays the views interlaced vertically, and adds selection controls above the Viewer.
- **Checkerboard** - displays the views using an alternating checkerboard pattern (one pixel from left and one pixel from right), and adds selection controls above the Viewer.
- **Anaglyph** - displays the views simultaneously using a red hue for left and green hue for right, and adds selection controls above the Viewer.
- **Flicker** - displays both views alternately, and adds selection controls above the Viewer.

### Capturing the Viewer

You can capture the contents of the Viewer for a quick real-time flipbook, also known as a playblast, and save the content out to **.jpg** for review. This is very useful for quickly checking the animation in your 3D scene in real-time without having to do a full scanline render. Playblast flipbooks the Viewer 'as is', including 2D and 3D scenes, handle and transform overlays (such as roto shape outlines), wipes, and so on.
The contents of a Viewer containing 3D and 2D information, including the wipe handle, captured in .jpg format.

To capture the Viewer contents:
1. Click the **Capture** button under the Viewer, to the right of the flipbook button. The **Capture** dialog displays.

![Capture dialog](image)

2. Select the required **Flipbook** tool using the dropdown.
3. If you have more than one Viewer in the script, select the required Viewer.
4. Check that your settings are correct in the dialog. The default values are copied from the Viewer you currently have active. You can change them if necessary:
   - **Frame range** - set which Viewer you want to draw default values from, and set the frame range that you want to flipbook.
   - **Use proxy** - check to use proxy mode.
   - **Render in background** - check to render in the background. If you check this, you can also set **Thread limit** and **Memory limit** controls. The former limits the number of threads that Nuke uses in the background and the latter limits the amount of cache memory that Nuke uses.
   - **Delete existing temporary files** - disable this if you want to retain previously cached files.
   - **Continue on error** - check to keep rendering even if an error occurs during the process.
   - **Buffer** - select the buffer you want to capture.
   - **Customise write path** - Check this to manually set the location (in the Write path field below) where the .jpg files are stored.
   - **No flipbook** - Enable this if you don’t require the selected flipbooking tool to display the frame range.
5. Click OK to capture the contents of the Viewer.
   The frame range is cached as .jps to either the default location set in the Preferences dialog, or – if enabled – the directory specified in the Write path control.
# Previewing on an External Broadcast Video Monitor

To check the final result in correct video colorspace and pixel aspect ratio, you can preview the current Viewer image on an external broadcast video monitor. This option requires additional hardware, such as a monitor output card or a FireWire port.

Our monitor out architecture interfaces directly with the AJA and BlackMagic device drivers, which are unified across their respective hardware lines, meaning all current supported cards for the versions detailed in [Third-Party Library Versions](#) should work. We've tested the following AJA and Blackmagic hardware:

<table>
<thead>
<tr>
<th>AJA Card:</th>
<th>KONA LHi</th>
<th>KONA 3G</th>
<th>KONA 4</th>
<th>KONA iOXT</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Formats</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SD</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>HD</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>2K</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>UHD</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td></td>
</tr>
<tr>
<td>4K</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>BNC</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td></td>
</tr>
<tr>
<td>HDMI</td>
<td>●</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td><strong>Stereoscopic Support</strong></td>
<td>No</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td><strong>Platforms</strong></td>
<td>Win, Mac, Linux</td>
<td>Win, Mac, Linux</td>
<td>Win, Linux</td>
<td>Mac 10.9 and 10.10</td>
</tr>
<tr>
<td><strong>Drivers</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### AJA Card:

<table>
<thead>
<tr>
<th></th>
<th>KONA LHi</th>
<th>KONA 3G</th>
<th>KONA 4</th>
<th>KONA iOXT</th>
</tr>
</thead>
<tbody>
<tr>
<td>• Windows -</td>
<td>• Windows -</td>
<td>• Windows -</td>
<td>Mac -</td>
<td>Mac -</td>
</tr>
<tr>
<td>12.5.1</td>
<td>12.5.1</td>
<td>12.5.1</td>
<td>12.5.1</td>
<td>12.5.1</td>
</tr>
<tr>
<td>• Mac -</td>
<td>• Mac -</td>
<td>• Linux -</td>
<td>Linux -</td>
<td></td>
</tr>
<tr>
<td>12.5.1</td>
<td>12.5.1</td>
<td>12.5.2.7</td>
<td>12.5.2.7</td>
<td></td>
</tr>
</tbody>
</table>

**Note:** The following should be taken into account when using the Monitor Out functionality with AJA cards:

If you’re running AJA cards on Linux, you can contact [www.aja.com/support](http://www.aja.com/support) to obtain the correct drivers.

12-bit monitor output is only supported with dual connection cards, that is cards with two physical connections, not dual links combining two separate streams of data.

Hiero is unable to send out the right eye separately using the 2nd output cable of KONA 3G cards. Instead, both views are sent through the 1st output and can be viewed using the side-by-side, anaglyph, and interlacing options.

### Blackmagic Card:

<table>
<thead>
<tr>
<th>Blackmagic Card:</th>
<th>DeckLink SDI</th>
<th>DeckLink HD Extreme 2</th>
<th>DeckLink Extreme 3D+</th>
<th>Intensity Pro 4K</th>
</tr>
</thead>
<tbody>
<tr>
<td>Formats</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SD</td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
</tr>
<tr>
<td>HD</td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
</tr>
<tr>
<td>2K</td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
</tr>
<tr>
<td>UHD</td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
</tr>
<tr>
<td>4K</td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
</tr>
<tr>
<td>BNC</td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
</tr>
<tr>
<td>HDMI</td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
<td><img src="false" alt="Yes" /></td>
</tr>
</tbody>
</table>

**Stereoscopic Support**

| No | No | Yes | No |

**Platforms**
<table>
<thead>
<tr>
<th>Blackmagic Card:</th>
<th>DeckLink SDI</th>
<th>DeckLink HD Extreme 2</th>
<th>DeckLink Extreme 3D+</th>
<th>Intensity Pro 4K</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Win, Mac, Linux</td>
<td>Win, Mac, Linux</td>
<td>Win, Mac, Linux</td>
<td>Win, Mac, Linux</td>
</tr>
</tbody>
</table>

**Drivers**

<table>
<thead>
<tr>
<th></th>
<th>Desktop Video 10.7 to 10.8.4</th>
<th>Desktop Video 10.7 to 10.8.4</th>
<th>Desktop Video 10.7 to 10.8.4</th>
<th>Desktop Video 10.7 to 10.8.4</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Blackmagic Card:</th>
<th>DeckLink Studio 4K</th>
<th>DeckLink 4K Extreme</th>
<th>DeckLink 4K Extreme 12G</th>
</tr>
</thead>
</table>

**Formats**

- **SD**: Yes
- **HD**: Yes
- **2K**: Yes
- **UHD**: Yes
- **4K**: Yes
- **BNC**: Yes
- **HDMI**: Yes

**Stereoscopic Support**

- Yes (Both views through one output, so the **Full Resolution** option is not available.)
- Yes
- Yes

**Platforms**

- Win, Mac, Linux
- Win, Mac, Linux
- Win, Mac, Linux

**Drivers**
Nuke can output images to external broadcast monitors in either 8- or 10-bit RGB color modes. 10-bit color is automatically selected when the Viewer’s `gl buffer depth` setting is `half-float` or `float`, provided it is supported by the monitor output card. In all other cases, 8-bit color is used.

**Note:** Selecting a monitor output mode with the phrase **10-bit** in its description only outputs true 10-bit color if a `gl buffer depth` of either `half-float` or `float` is selected. Half-float and float modes are considerably slower to process, so it is recommended to stick with `byte` mode for monitor output unless 10-bit color is specifically required.

### To Preview Output on an External Broadcast Video Monitor

1. Press **S** on the Viewer to open the Viewer settings.
2. From the **monitor output device** dropdown menu, select the external device you want to use and check **enable monitor output**. All available devices are automatically detected and listed in this menu.
3. From the **monitor output mode** dropdown menu, select the display mode for the device you selected in the previous step. The available options depend on the device you are using. By default, the most recently used display mode is selected.
4. Press **Ctrl/Cmd+U**, navigate to **Viewer > Toggle Monitor Out**, or click on the monitor output button in the Viewer overflow dropdown.

From now on, any output connected to the Viewer is sent to the monitor output device you selected.
The monitor output image is always full frame, unzoomed (1:1 ratio), and unpanned, regardless of what the Viewer it is linked to is set to. This means that slightly mismatching formats (for example, 640x512 / 640x448 for PAL/NTSC) are not rescaled to fit the monitor.

If you save your Nuke script with monitor output enabled, this setting is saved with the script. The next time you open the same script, monitor output is enabled.

To Disable Viewing on an External Broadcast Video Monitor

- Click on the monitor output button in the Viewer controls.
- Select Viewer > Toggle Monitor Output from the menu bar.
- Press Ctrl/Cmd+U. This can be particularly useful if you are using a monitor output device that can draw things on the screen you are using to display the Nuke window (such as the Digital Cinema Desktop Preview device installed with Apple Final Cut Pro).
- Press S on the Viewer to open the Viewer settings, and uncheck enable monitor output.
Rendering Output

Nuke can render images locally on your workstation (see Output (Write) Nodes) or it can be setup to render images on a network render farm (see Using the Frame Server on External Machines). Before rendering, make sure that you are using the appropriate file name syntax (see File Name Conventions for Rendered Images) and verify that your project settings have the correct output format and proxy format selected (see Render Resolution and Format).

By default, if you choose to render frame 15, the resulting file is numbered accordingly, for example image.0015.rgb. However, you can change this behavior via expressions, specified start frames, and constant offsets (see Changing the Numbering of Rendered Frames).

Sometimes, you may want to render an image just to read the rendered image back in (see Using a Write Node to Read in the Rendered Image). Because reading the output from a file is faster than calculating its output by processing the node tree upstream, this can speed up large projects.

**Note:** Scripts that require Nuke to load a large number of files concurrently (for example, by having hundreds of Read nodes followed by TimeBlurs) may exceed the number of available file handles per process, causing problems when rendering scripts.

Nuke itself supports up to 2048 file handles on all systems; however, you may need to increase the file handle limit on your system.

On Mac, you can increase the default limit of 256 (depending on your version) by entering the following command from the Terminal and then running Nuke from the same Terminal session:

```bash
ulimit -Sn 2048
```
Render Resolution and Format

Before rendering a script, it’s important to check what is the currently active mode: the full-size or the proxy mode. Nuke executes all renders at the currently active scale. Thus, when a script is rendered in proxy mode, processing is done at the proxy scale and image output goes to the file name in the Write node’s proxy field. If you do not specify a proxy file name, the render fails with an error. It never resizes the proxy image, and it does not write the proxy image over the full-size one.

To view and change the proxy resolution for the current script file, select Edit > Project Settings from the menu bar, or press S with the mouse pointer over the Node Graph or the Properties Bin.

![Changing the output resolution under project settings.]

From the Project Settings properties panel, you can select a new render format from the dropdown menu of predefined resolutions, and toggle proxy rendering. You can also select the new option under either full size format or proxy format or use the proxy scale fields to define custom render resolutions for the composite. When rendering in proxy mode, use the dropdown menu on the right to select whether to use the resolution defined under proxy format or proxy scale. Also check that you have set read proxy file to what you want - this setting controls how Read nodes select the file to read (full res file or proxy) in the proxy mode. For more information on these settings, refer to Setting Up Your Script.
Output (Write) Nodes

With the correct resolution and format selected, you then insert Write nodes to indicate where you want to render images from the script.

![Diagram of a compositing tree with Write nodes]

Inserting Write nodes for rendering.

One Write node is usually placed at the bottom of the compositing tree to render the final output. However, Write nodes have both input and output connectors, so they may be embedded anywhere in the compositing tree.

You can execute renders for a single Write node or all Write nodes in your compositing script.

To Render a Single Write Node

1. Select the node in the script from which you want to render an image.
2. Select Image > Write (or press W over the Node Graph). Nuke attaches a Write node and opens its properties panel.
3. Connect a Viewer to the Write node you want to render and verify that the correct resolution is displayed for output. If necessary, press Ctrl/Cmd+P to toggle between full-res and proxy resolution. The displayed output resolution is used for rendering.
4. In the properties panel, click the file or proxy field's folder icon (depending on whether you want to render high res or low res images) and browse to the directory where you want to store the rendered sequence. For instructions, see Using the File Browser.

5. After the path, type a name for the rendered image and then click OK. If you're rendering an image sequence, include the frame number variable (for example, ####) in the name. See To Render Selected or All Write Nodes in the Script below for examples of valid file names with the frame number variable.

6. If necessary, adjust the following controls:
   • Using the channels dropdown menu and checkboxes, select the channels you want to render.
   • Using the frame dropdown menu and input field, set the relation between the currently processed frame and the numbering of the frame written out. For more information, see Changing the Numbering of Rendered Frames.
   • check read file if you want the output of the Write node to be produced by reading the rendered file back in rather than by processing the upstream node tree. For more information on this and the missing frames control, see Using a Write Node to Read in the Rendered Image.
   • From the colorspace dropdown menu, select which lookup table to use when converting between the images’ color space and Nuke's internal color space.
   • From the file type dropdown menu, select the file format for the rendered images. If you don't specify a file format, Nuke uses the extension in the file name to figure out the format.
• Check the **limit to range** box if you want to disable the node when executed outside of a specified frame range. In the **frame range** fields, enter the range of frames you want to make executable with this Write node.

7. In the Write node properties, click the **Render** button.

8. In the Render dialog, adjust the render settings if necessary:
   - **Frame range** - set the frame range you want to render.
   - **Use proxy** - check to use proxy mode.
   - **Render using frame server** - enable this checkbox if you want to use Nuke's frame server to render the frames specified in the current render task. When the frame server is disabled, Nuke can only perform one render task at a time. Executing another render displays a warning and the current render is paused until you acknowledge the message:

   ![Warning Message](image)

   See [Rendering Using the Frame Server](#) for more information.

   - **Continue on error** - check to keep rendering even if an error occurs during the process.
   - **Views** - set which views to include in the render. See [Selecting Which Views to Render](#) for more information.

9. Click **OK**.

**Tip:** When specifying the frame range to render, you can enter complex frame ranges into the frame range prompt dialog. For example, if you enter **1-5 8 10 15 22-25**, it only renders those frames. You can also increment rendered frames using something like **10-50x10**, which resolves to only frames 10, 20, 30, 40, and 50 from the range.

 Likewise, you can specify multiple ranges on the command line, for example:

```plaintext
nuke -F 1-5 -F 8 -F 10 -F 15 -F 20-40x5 -x myscript.nk
```

**Tip:** When rendering with the Write node, you can force a certain data type by adding the data type and a colon before the file path. For example, you can enter **ftiff:C:\Temp\test.tif** as the file path to render a file whose data type is ftiff and extension `.tif`.
To Render Selected or All Write Nodes in the Script

1. Connect a Viewer to a Write node you want to render and verify that the correct resolution is displayed for output.
2. If necessary, press Ctrl/Cmd+P to toggle between full-res and proxy resolution. The displayed output resolution is used for rendering.
3. If you want, you can change the order in which your Write nodes are rendered by giving them custom render order numbers in the render order field.
4. Do one of the following:
   • With the desired Write node selected, select Render > Render selected (or press F7).
   • Select Render > Render all (or press F5).
5. In the Render dialog, adjust the render settings if necessary. The default values are drawn from the Viewer you have active.
   • Frame range - set the frame range you want to render.
   • Use proxy - check to use proxy mode.
   • Render using frame server - disable this checkbox if you don't want to use Nuke's frame server to render the frames specified in the current render task. If you disable the frame server, Nuke can only perform one render task at a time. Executing another render displays a warning and the current render is paused until you acknowledge the message:

   ![Warning Message]

   See Rendering Using the Frame Server for more information.
   • Continue on error - check to keep rendering even if an error occurs during the process.
   • Views - set which views to include in the render. See Selecting Which Views to Render for more information.
6. Click OK.

**Tip:** When specifying the frame range to render, you can enter complex frame ranges into the frame range prompt dialog. For example, if you enter "1-5 8 10 15 22-25", it only renders those frames. Likewise, you can specify multiple ranges on the command line, for example: nuke -F 1-5 -F 8 -F 10 -F 15 -F 22-25 -x myscript.nk
You can see the progress of your render in the status window that appears. When the render is complete, the rendered images are added to the directory you specified.

**Tip:** When rendering with the Write node, you can force a certain data type by adding the data type and a colon before the file path. For example, you can enter `ftiff:C:\Temp\test.tif` as the file path to render a file whose data type is `ftiff` and extension `.tif`.

### Selecting Which Views to Render

The Write nodes in a script determine which views are available at render time, but the Render dialog can be used as a filter to limit the views rendered from those available. Write nodes and the Render dialog default to all available views.

For example, if your script contained two Write nodes calling for the main view and left and right views, but the Render dialog is set to render main, then only the main view is rendered.

Conversely, if the Write nodes are calling for the left and right views, but the Render dialog is set to render main, then no views are rendered.

A slightly more complex example is shown in the image.

In the example above, the left-hand dialog is set to render only main and the right-hand dialog is set to render only cam1 and cam2.
So, in the case of the first render, Write1 renders main, Write2 renders nothing, and Write3 renders main.

In the case of the second render, Write1 renders cam1 and cam2, Write2 renders cam1, and Write3 renders nothing.

Notes on Rendering QuickTime Files

If you are rendering .mov files, you can:

• choose the QuickTime codec from the codec dropdown menu.

**Note:** If you're using the Avid DNxHD codec, Avid AVDn, avoid setting the pixel format control to r408 as there is a known issue within the codec causing frames to darken with each frame progression in the sequence.

• Set the encoder library used to write the file:

**Note:** Depending on the codec in use, this control may be read only. For example, Apple ProRes 4444 always uses mov64, but Animation allows you to choose mov32 or mov64.

• mov32 - uses the full range of QuickTime codecs, but can be slow to process due to extra complexity during decode.

• mov64 - uses its own packing and unpacking and streams decode/encode for extra processing speed, but only supports a sub-set of QuickTime codecs.

**Note:** Nuke defaults to the fastest decoder for the codec used in the file - if you're reading in a type supported by the mov64 sub-set, Nuke defaults to that reader. Otherwise, the fallback mov32 reader is used.

• fps - set the playback frames per second for the output file.

• audio file - Allows you to specify a separate audio file to include in the output. Either enter the filepath manually or click the browse button to locate the audio file.

• audio offset - Sets the start time of any audio file specified in the audio file control. The unit of measure is specified using the units control. Negative values cause the audio to start before the video and vice versa.

• write timecode - When enabled, Nuke writes the timecode into the .mov metadata, where available.
**Advanced QuickTimes Options**

You can adjust advanced codec options by opening the **Advanced** dropdown. The options available vary, depending on whether you're using the **mov32** or **mov64** encoder.

<table>
<thead>
<tr>
<th>Control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>mov32 encoder</strong></td>
<td></td>
</tr>
<tr>
<td>codec options</td>
<td>Click to display an advanced <strong>Compression Settings</strong> dialog.</td>
</tr>
<tr>
<td>fast start</td>
<td>When enabled, MOVs are playable while still down loading.</td>
</tr>
<tr>
<td>use format aspect</td>
<td>When enabled, the rendered .mov uses the same pixel ratio as the input.</td>
</tr>
<tr>
<td></td>
<td>When disabled, the codec determines the pixel aspect to use.</td>
</tr>
</tbody>
</table>

**Note:** Codecs writing PAL and NTSC should be allowed to determine the ratio during render, but formats that otherwise expect 1:1 pixel ratios may require this override.

<table>
<thead>
<tr>
<th>ycbcr matrix</th>
<th>Sets the way RGB is converted to YCbCr. <strong>Rec 601</strong> and <strong>Rec 709</strong> follow the ITU.BC specifications, whilst <strong>Nuke Legacy</strong>, <strong>Nuke Legacy Mpeg</strong>, and <strong>Nuke Legacy YUVS</strong> are retained for backwards compatibility. <strong>Format-based</strong> sets the color matrix to <strong>Rec 601</strong> for formats with a width below 840 pixels and <strong>Rec 709</strong> for formats with a width of 840 pixels or above. This setting is only available when you’re working with a YCbCr-based pixel type.</th>
</tr>
</thead>
</table>
| pixel format             | Lists pixel formats supported by the current codec. The pixel format defines the type and layout Nuke requests from QuickTime:  
  • Pixel colorspace - either **RGB(A)** or **YCbCr(A)**. This defines whether QuickTime or Nuke's QuickTime reader does the conversion between colorspace. For a Y'CbCr pixel type, choosing an **RGB(A)** colorspace means Nuke relies on QuickTime to do the RGB to Y'CbCr conversion. Choosing a **YCbCr(A)** colorspace means that Nuke is responsible for the conversion, and so a |
### Control Description

- **pecific ycbcr matrix can be used (this is recommended).
- **Pixel bit depth - 8-bit, 16-bit, and so on.** This sets the encoding depth used when decompressing the frames. A large bit depth gives higher accuracy at the cost of speed and memory usage.
- **Pixel layout - 422, 444, 4444, and so on.** This defines how the chroma channels in the buffer are arranged. 444 buffers have lower spatial chroma sampling than 422, so they are generally preferred when available. For all cases, Nuke unpacks the sub-sampled buffer to full resolution.
- **Range - either Biased or empty.** For RGB(A) types, the values are full range (from 0 to 1). For YCbCr(A) types, the values are in video range by default, offering headroom at both ends of the scale. If this is set to Biased, then headroom is only available at the top end.

- **(4cc).** This is the pixel type 4cc, as defined by the QuickTime API.

  This setting defaults to the best format accepted by the codec.

<table>
<thead>
<tr>
<th>Control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>write nclc</td>
<td>When enabled, write the nclc data in the colr atom of the video sample.</td>
</tr>
<tr>
<td>write gamma</td>
<td>When enabled, write the gama data in the gama atom of the video sample.</td>
</tr>
<tr>
<td>write prores</td>
<td>When enabled, write the prores data in the prores header of the video sample.</td>
</tr>
</tbody>
</table>

### mov64 encoder

- **bitrate**
  
  Sets the target bitrate that the codec attempts to reach, within the limits set by the bitrate tolerance and quality min/max controls.

  **Note:** The bitrate control is only enabled for certain codecs, such as MPEG-4 - Video.

- **bitrate tolerance**
  
  Sets the amount that the bitrate can vary from the bitrate setting. Setting this tolerance too low will result in renders failing.

  **Note:** The bitratetolerance control is only enabled for certain codecs, such as MPEG-4 - Video.
<table>
<thead>
<tr>
<th>Control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>quality min</td>
<td>Sets the quality range within which the codec can vary the image to achieve the specified bitrate. Higher ranges can introduce image degradation.</td>
</tr>
<tr>
<td>quality max</td>
<td><img src="image" alt="Note: The quality min/max controls are only enabled for certain codecs, such as MPEG-4 - Video." /></td>
</tr>
<tr>
<td>gop size</td>
<td>Sets how many frames can be placed together to form a compression GOP (group of pictures).</td>
</tr>
<tr>
<td></td>
<td><img src="image" alt="Note: Use caution with this control as large alterations can stop other applications reading the rendered file." /></td>
</tr>
<tr>
<td></td>
<td><img src="image" alt="Note: The gop size control is only enabled for certain codecs, such as MPEG-4 - Video." /></td>
</tr>
<tr>
<td>b frames</td>
<td>Sets the maximum number of B frames that can be consecutive in the rendered file. The default, 0, does not impose any maximum number of B frames in the output.</td>
</tr>
<tr>
<td></td>
<td><img src="image" alt="Note: The b frames control is only enabled for certain codecs, such as MPEG-4 - Video." /></td>
</tr>
<tr>
<td>write nclc</td>
<td>When enabled, write the nclc data in the colr atom of the video sample.</td>
</tr>
</tbody>
</table>

Nuke writes the selected colorspace and pixel format, along with some other information, into the metadata of the file. If you then read the rendered file in, Nuke reads that metadata and is able to pick the correct defaults for the file. To see the file metadata yourself, use a ViewMetaData node.

When writing QuickTime files, some users have reported receiving the following error at the end of rendering:

*Failed to flatten movie data: the movie is open in another application.*

This is because the output file has been opened by some other process. This can be the Finder on Mac trying to show a preview, an OS file system search trying to index the file, or a virus checker, for
example. The workaround is to turn off **Fast Start** in the Write node controls to skip the flattening process that is affected.

## Notes on Rendering OpenEXR Files

Nuke supports multi-part OpenEXR 2.2 files, which allow you to store your channels, layers, and views in separate parts of the file. Storing the data this way can make loading `.exr` files faster, as Nuke only has to access the part of the file that is requested rather than all parts. However, for backwards compatibility, you also have the option to render your `.exr` files as single-part images.

To set how the data is stored in your rendered `.exr` file, open the Write properties and set **interleave** to:

- **channels, layers and views** - Write channels, layers, and views into the same part of the rendered `.exr` file. This creates a single-part file to ensure backwards compatibility with earlier versions of Nuke and other applications using an older OpenEXR library.

- **channels and layers** - Write channels and layers into the same part of the rendered `.exr` file, but separate views into their own part. This creates a multi-part file and can speed up Read performance, as Nuke only has to access the part of the file that is requested rather than all parts.

- **channels** - Separate channels, layers, and views into their own parts of the rendered `.exr` file. This creates a multi-part file and can speed up Read performance if you work with only a few layers at a time.
Rendering Using the Frame Server

The Frame Server reduces render times by sharing work over the number of render processes specified in the Preferences. The Frame Server logs renders in the Background Renders panel, including information such as script name, Write node, frame range, progress, and whether or not the render produced errors. The Frame Server does not display a progress bar for render tasks, but you can disable Render using frame server in the Render dialog to use the process from older versions of Nuke.

The Frame Server is disabled by default, but you can enable it in the Preferences > Performance > Threads/Processes or on a per render basis in the Write node's Render dialog.

**Note:** Local Frame Server processes use ports 5558-5662.

Rendering Write nodes, either from a node's Properties or from the Render menu, displays the Render dialog. See Output (Write) Nodes for more information on initiating renders. If the Frame Server is enabled, clicking OK in the Render dialog starts the selected renders in the background, allowing you to keep working.

The Background Renders tab is not displayed as part of the workspaces that ship with Nuke, but you can enable Performance > Threads/Processes > focus background renders in the Preferences to open the panel automatically when the Frame Server is in use.

**Tip:** You can also add the panel manually by right-clicking the button in the top-left of any pane, and then selecting Windows > Background Renders.
The **Background Renders** panel includes information such as script name, Write node, frame range, progress, and whether or not the render produced errors.

**Frame Server Preferences**

You can control how the Frame Server behaves using the **Performance > Threads/Processes** tab in the **Preferences**:

- **render using frame server (Nuke)** - when enabled, the Frame Server is always used for rendering.
- **focus background renders** - when enabled, rendering using the Frame Server automatically opens the **Background Renders** panel, or if it is already open, shifts focus to the panel.
- **frame server processes to run** - sets the number of render processes available to the Frame Server.
Note: You must restart Nuke if you change the number of processes available.

- **export renders** - allows you to set limitations on the resources available for render processes:
  - **limit renderer (more responsive ui)** – select this option to make the user interface more responsive during rendering. It tells Nuke to use 2 threads to transcode and to use 25% of RAM to cache. Using this option is likely to result in slower renders.
  - **no renderer limits (fastest transcoding)** – select this option to ensure that renders happen as quickly as possible. This option may result in a less responsive user interface during rendering.
  - **customize render limits** – select this option to manually configure the number of threads used and cache memory available during renders.

Using the Frame Server on External Machines

Although Nuke is capable of rendering frames internally, running the Frame Server on an external machine can accelerate the process considerably by sharing work across a network of machines.

Note: The Frame Server requires a Nuke license (nuke_i) on the main workstation, but only a Nuke render license (nuke_r) on the slave machines.

If you want to use an interactive license (nuke_i) on the slave machines, add the **useInteractiveLicense** argument to the **runframeserver.py** command described below.

Configuring the Frame Server on External Machines

Nuke's Frame Server can be set up on an external machine (or a number of machines) to render from your Nuke Studio session. To do this, you need to run the **runframeserver.py** script on the external machines, found inside the Python site-packages, with specific command line arguments.

Warning: In order for everything to work smoothly, you need to ensure that both your external slave machines and main Nuke session can read and write files to a shared location, such as an NFS share.

Depending on platform this can be done by manipulating your default **umask** setting, but be aware that this alters the permissions of the created files.
Additionally, Macs and certain Linux distributions, such as RHEL, can not function as the main workstation if the firewall is blocking the communication port 5560. You can configure the firewall to allow certain ports through the firewall using the `iptables` command, but use caution when doing so. For example:

```
sudo iptables -I INPUT -p tcp --dport 5560 --syn -j ACCEPT
```

Please refer to the documentation on firewalls for your particular platform for more information.

The Frame Server uses a number of worker processes on the external machine, each of which requires allocated resources, such as threads, memory, and so on. There are a number of arguments that you must pass to `runframeserver.py` for the server to work correctly:

- **--numworkers** - this is the number of concurrent Nuke processes that are launched when you run this server render node.
- **--nukeworkerthreads** - the number of threads that each worker is allocated. This is similar to setting the `-m` argument when running Nuke from the command line.
- **--nukeworkermemory** - the amount of memory, in MB, allocated to each frame server worker.
- **--workerconnecturl** - the TCP port address of the main workstation you want to serve. For example:

  tcp://bob:5560

  where *bob* is the resolved hostname of a machine you wish to serve. You can also use an IP address.

  **Tip:** To ensure that you're entering a valid URL, try using the `ping` command to see if you get a response.

- **--nukepath** - the path to the Nuke application on the slave workstation.

  **Tip:** On Windows, if there are spaces in the file path, remember to place the path in quotes. For example, `--nukepath="C:\Program Files\Nuke11.3v6\Nuke11.3.exe"

On a Linux slave machine, an example command prompt entry running from the install directory might look like this:

```
./python ./pythonextensions/site-packages/foundry/frameserver/nuke/runframeserver.py --numworkers=2 --nukeworkerthreads=4 --nukeworkermemory=8096 --workerconnecturl=tcp://bob:5560 --nukepath=./Nuke11.3
```
On a Windows slave machine, an example command prompt entry running from the install directory might look like this:

```bash
python.exe pythonextensions\site-packages\foundry\frameserver\nuke\runframeserver.py --numworkers=2 --nukeworkerthreads=4 --nukeworkermemory=8096 --workerconnecturl=tcp://bob:5560 --nukepath=Nuke11.3.exe
```

In the examples, we specify that the slave uses two Nuke workers, with four threads and 8 GB RAM each, and are slaved to the main Nuke workstation running on `bob`.

**Tip:** If your slave machines run a different OS than your main Nuke machine, you can use the `--remap` command line argument to convert file paths between them. The host file path is read first followed by the slave file path. Nuke expects all file paths to use `/` (forward slash) between directories. For example:

```bash
--remap "P:/,/mnt/renders/
```

converts host paths beginning with `P:` (Windows style) to slave paths beginning with `/mnt/renders/` (Linux style).

You can check that the Frame Server and workers are connected by running the following lines in the Script Editor on the main workstation:

```python
from hiero.ui.nuke_bridge.FnNsFrameServer import frameServer
print [worker.address for worker in frameServer.getStatus(1).workerStatus]
```

Successful connections should report something similar to the following in the output panel:

`['Worker 0 - henry.local - 192.168.1.11', 'Worker 0 - bob.local - 192.168.1.111', 'Worker 1 - henry.local - 192.168.1.11']`

Where `henry.local` is the name of the remote slave, and `bob.local` is the name of the main Nuke session.

**Note:** If the workers cannot contact the Frame Server, an exception is printed in the Script Editor's output panel.

### Frame Server Logs

Broker and Worker logging can help diagnose Frame Server issues. The logs are written to `NUKE_TEMP_DIR/logs` by default, and take the form:

- `broker.log`
- `worker-0.log`
worker-1.log
worker-2.log

**Note:** Running the Frame Server using Python, as described above, always writes log files to the specific OS temporary directory. For example, on Windows C:\temp is used.

**Tip:** You can use the FRAMESERVER_LOG_DIR environment variable to force Frame Server logs into a different location. See for more information.
File Name Conventions for Rendered Images

There is no parameter in the Write node to specify output format. Instead, format is indicated by a prefix or an extension when you type the file name. Here is the appropriate syntax:

```
<prefix>:/<path>/<name>.<frame number variable>.<extension>
```

The optional `<prefix>` can be any valid extension. `<path>` is the full path name to the directory where you want to render. The `<frame number variable>` is usually entered as `####`, with `####` indicating frame numbers padded to four digits.

You can change the padding by substituting `####` with any number of hash marks. For example, two-digit padding would be `##`, three-digit would be `###`, and five-digit would be `#####`.

With these conventions in mind, suppose you want to save an image sequence called “final_comp_v01” to the TIFF16 format. Here are examples of names that work in the Write node:

```
tiff16:/<path>/final_comp_v01.####.tiff
/<path>/final_comp_v01.####.tiff16
tiff16:/<path>/final_comp_v01.####
```

All extensions supported by Nuke may be used for the prefix. See Appendix C: Supported File Formats on page 2007 for a complete list of recognized extensions.

When a prefix is used, it takes precedence over the format represented by an extension. In fact, the prefix makes the extension unnecessary.

You could, for example, enter `exr:/<path>/###` as the file name, and this would create an OpenEXR image sequence with frame numbers only, padded to three digits.

Writing Versions of Rendered Images

You can write out versions of a file using the `Alt+Up/Down` arrow keys. Versions must written in the following format in order for Nuke recognize them:

```
<path>/final_comp_v01.####.tiff
<path>/final_comp_v02.####.tiff
<path>/final_comp_v03.####.tiff
```
Changing the Numbering of Rendered Frames

By default, when you render an image sequence, Nuke assumes an exact relation between the currently processed frame and the numbering of the frame written out. Therefore, if you choose to render frame 15, the resulting file is numbered accordingly, for example image.0015.rgb. However, the frame parameter on the Write node lets you change this behavior. For instance, if you have a sequence that runs from image.0500.rgb to image.1000.rgb, you may want to render it so that the frame numbering in the resulting files runs from image.0001.rgb to image.0501.rgb. You can do so via expressions, specified start frames, and constant offsets. Each method is described below.

Using Expressions

1. Select Image > Write to insert a Write node into your script.

2. In the Write properties panel, click the file folder icon, then navigate to the directory path where you want to save the rendered image sequence. Enter a name for the image sequence.

3. Set frame to expression. Enter an expression in the field on the right. The expression changes the relation between the currently processed frame and the numbering of the frame written out. The resulting file name for the current frame is displayed on the Write node in the Node Graph.

For example, if your clip begins from frame 500 and you want to name that frame image.0001.rgb rather than image.0500.rgb, you can use the expression frame-499. This way, 499 frames are subtracted from the current frame to get the number for the frame written out. Frame 500 is written out as image.0001.rgb, frame 501 is written out as image.0002.rgb, and so on.
Another example of an expression is \( \text{frame}^2 \). This expression multiplies the current frame by two to get the number of the frame that’s written out. At frame 1, image.0002.rgb is written out; at frame 2, image.0004.rgb is written out; at frame 3, image.0006.rgb is written out; and so on.

### Specifying a Frame Number for the First Frame in the Clip

1. Select **Image > Write** to insert a Write node into your script.

2. In the Write properties panel, click the file folder icon, then navigate to the directory path where you want to save the rendered image sequence. Enter a name for the image sequence.

3. Select **start at** from the frame dropdown menu. Enter a start frame number in the field on the right. This specifies the frame number given to the first frame in the sequence. The numbering of the rest of the frames is offset accordingly.

![Write node in NUKE](image.png)

For example, if your sequence begins from frame 500 and you enter 1 in the field, frame 500 is written out as image.0001.rgb, frame 501 as image.0002.rgb, and so on. Similarly, if you enter 100 in the field, frame 500 is written out as image.0100.rgb.

### Offsetting All Frame Numbers by a Constant Value

1. Select **Image > Write** to insert a Write node into your script.

2. In the Write properties panel, click the file folder icon, then navigate to the directory path where you want to save the rendered image sequence. Enter a name for the image sequence.

3. From the frame dropdown menu, select **offset**. Enter a constant offset in the field on the right. This constant value is added to the current frame to get the number for the frame that’s written out.
For example, if your clip begins from frame 500 and you want to render this first frame as image.0001.rgb rather than image.0500.rgb, you can use \(-499\) as the constant offset. This way, 499 is subtracted from the current frame to get the number for the frame that’s written out. At frame 500, image.0001.rgb is written out; at frame 501, image.0002 is written out, and so on.
Using a Write Node to Read in the Rendered Image

You can use a Write node to both render an image and read the rendered image back in. Because reading the output of a Write node from a file is faster than calculating its output by processing the node tree upstream, this can be particularly useful on large comps. When you have finished working on a branch of the node tree, you can insert a Write node after it, render the output, and use the same Write node to read the rendered image in. If you later need to edit the nodes upstream, simply make your changes and render the Write node again to update the image being read in.

To Use a Write Node to Read in the Rendered Image

1. Render an image as described in Output (Write) Nodes. We recommend rendering the image as an .exr. This way, Nuke writes the hash value of the incoming node tree into the rendered file. If the node tree changes and the rendered file gets out of date, the hashes won't match and Nuke notifies you of the problem.

2. In the Write node properties, check read file. When this is on, Nuke ignores the upstream node tree and uses the rendered image as the output of the Write node.

3. To check whether the input file is up to date with the input tree connected to the Write node, check check file matches input. This only works with .exr files written by Nuke and when the proxy mode and down-rez are disabled. If the input file cannot be checked, Nuke displays the word unchecked on the Write node in the Node Graph.

4. If there is an error when loading the rendered file, select what to do from the missing frames dropdown menu:
   - error - display an error message on any missing frames.
   - black - replace any missing frames with black.
   - checkerboard - replace any missing frames with a checkerboard image.
   - read input - display the result of the input tree rather than the rendered file on any missing frames.

Tip: You can also use the Precomp node (Other > Precomp) to reduce portions of the node tree to pre-rendered image inputs. For more information, see Using the Precomp Node.
What is the Hash Value?

The hash value is a unique number (for example, b1c9c0aff2012a8) calculated from a node and the entire tree of nodes connected to its input. The class of the node and all the current control settings contribute to the hash value.

You can display the hash value at any point in the node tree by selecting a node in the Node Graph and pressing I. The hash is different at different points in the tree.
Render Farms

Nuke is supported by virtually all third-party and proprietary render-queuing software. By integrating Nuke with such a system, the render load can be distributed across all the Nuke-licensed machines on your network, whether Windows, Mac, or Linux-based.

**Note:** Instead of setting up a render farm, you can take advantage of the internal Frame Server, which allows you to setup external slave machines to process renders faster. See [Using the Frame Server on External Machines](#) for more information.

Your installation of Nuke may be configured to send jobs to a network render farm, which is usually made available under the Render menu (i.e., Render > Render). However, because this option must be customized for each studio, you should check with your system administrator for instructions on how to send a Nuke script for network rendering.

**Tip:** If you’re attempting to force Nuke to retry an operation rather than failing, such as for license issues during rendering, you may find Nuke’s exit codes helpful:

- 0 = success
- 1 = render error
- 100 = license failure
Organizing Scripts

As scripts grow in complexity and are worked on by several people, it becomes increasingly important to organize them in a clear and meaningful way. These pages teach you how to:

• display information about your Nuke script. See Displaying Script Information.

• find and replace all or part of file names or file paths in any node with file or proxy controls. See File Name Search and Replace.

• group nodes in the Node Graph using the Backdrop node or the Group node. See Grouping Nodes in the Node Graph.

• add notes to the Node Graph. See Adding Notes to the Node Graph.

• use the Precomp node to save a subset of the node tree as a separate .nk script, render the output of this saved script, and read the rendered output back into the main comp as a single image input. See Using the Precomp Node.

• use the LiveGroup node, which combines the power of groups and precomps with the utility of gizmos. See Using the LiveGroup Node.
Displaying Script Information

To display script information, such as the node count, channel count, cache usage, and whether the script is in full-resolution or proxy mode, do the following:

1. Select **File > Comp Info** (or press **Alt+I**).
   
   The script information window opens.

![Script Information Window](image)

2. If you make changes to your script while the window is open, click **update** to update the information.

3. To close the information window, click **close**.
Using Visual Diagnostics

Nuke can display accurate script profile data onscreen or output it to .csv or .xml file to help you troubleshoot bottlenecks in slow scripts. When visual diagnostics are calculating, timing information is displayed in the Node Graph, and the nodes themselves are colored according to the proportion of the total processing time spent in each one. The data is then displayed in the Profile tab as a bar or pie chart, timeline, or as a table.

Enabling Visual Diagnostics

Nuke's visual diagnostics use Profile nodes in the node tree to gather data. You can add as many Profile nodes as required, but one strategy might be to start at the foot of the Node Graph and work your way up until CPU or memory load decreases significantly.

**Note:** Profile nodes add processing overhead, but the data gathered still provides an accurate picture of where processing time is spent in the script.

1. Navigate to **Image > Profile** to add a Profile node to the Node Graph.
2. Connect the Profile node to the point in the node tree where you want to collect profiling data.

**Note:** You cannot connect Profile nodes directly to 3D, Particle, or Deep nodes. To profile these nodes, you'll need to render their output through a render node, such as ScanlineRender, RayRender, or a third-party renderer like PRManRender.

3. Click **open profile panel** in the node properties or click the content menu and select **Windows > Profile**.
   The Profile tab opens.
4. Click the **profile node** dropdown and then select the a Profile node in the Node Graph by name, OR
   Select the a Profile node in the Node Graph and then choose **selected**.
5. Set the **frame range** to collect data from and the required **data type** using the checkboxes.

**Tip:** The initial frame range is determined by the **Project Settings > frame range** control and **all** data types are collected by default. The **frame range** control accepts complex ranges, such as **1-10, 15-25 and 1-200x10**. See [Defining Frame Ranges](#) for more information.

6. Click **profile** to start data collection.

**Note:** If you've imported existing profiling data from a **.csv** or **.xml** file, you can't profile the current script. See [Exporting and Importing Profile Data](#) for more information.

As the specified frame range is calculated, timing information is displayed and the nodes in the Node Graph are colored according to the proportion of the total processing time spent in each one.
The profiling data is stored in the **Profile** tab's **DATA** panel when complete. See [Filtering Profile Data](#) for information on how to control what profiling data is displayed.
Using Visual Diagnostics | Enabling Visual Diagnostics

**Memory**
- CameraTracker: 14.03%
- Defocus: 0.00%
- MatchGrade: 0.22%
- Merge2: 21.28%
- Read: 40.88%
- RotoPaint: 4.40%
- Shuffle: 19.19%
Filtering Profile Data

The Profile tab's FILTER dropdown allows you to output only the profiling data you need, but the full data type set selected in the PROFILE dropdown remains available if you change your mind.

The following controls determine what output is available on-screen and for export to .csv or .xml file:

- **minimum threshold** - allows you to hide display data that doesn't reach a certain minimum value. For example, setting the threshold to 15% can hide any nodes that don't use at least 15% of the available memory.

![Graphs showing data with 0% and 15% minimum threshold](image)

0% minimum threshold. 15% minimum threshold.

**Tip:** You can zoom in and out of the DATA pane using the + - buttons.

- **display data** - determines what data is displayed from CPU, wall, ops, and memory. The default, all, displays all the available profiling data:
  - **CPU** - the time that the CPU spent executing the processing code, in microseconds, aggregated over all CPU threads. For example, with multi-threaded processing this is typically much larger than the wall time. If the average CPU time per thread (CPU divided by the number of threads used) is much shorter than the wall time, this suggests the CPU threads have spent a lot of time not executing code and perhaps waiting on locks, which could indicate a performance problem.

**Note:** On Mac and Windows, the CPU time is not currently accurate. On Mac, the CPU value is always similar to the wall time.
• **wall** - the time taken as it would be measured by a clock on the wall - the actual time you have to wait for the processing to complete. The **wall** time is also measured in microseconds.

• **ops** - the number of operators called in the node. Operators are Nuke's building blocks that perform certain tasks. Nodes can contain one or more ops. For example, when a node needs to resize something it would use a Transform op rather than an implementation of its own to do the same thing. See [https://learn.foundry.com/nuke/developers/110/ndkdevguide/intro/terminology.html#op](https://learn.foundry.com/nuke/developers/110/ndkdevguide/intro/terminology.html#op) for more information.

• **memory** - the total amount of system memory used by the node.

**Tip:** Bear in mind that some node classes also contain numerical values to differentiate between current and deprecated classes, such as Text and Text2.

• **display nodes** - for large scripts, it is impractical to display all the nodes in the Node Graph in the **Profile** tab. In **bar**, **pie** and **timeline** mode, only the top 15 results by resource usage are listed. In **table** mode, all results are listed.

The **display nodes** control allows you to be more selective with your output:

• **all** - displays data for all the nodes in the script.

• **list** - enter the explicit node names to display in the **DATA** panel. You can type the names manually or select nodes in the Node Graph and then click **add selected**.

You can also find nodes listed in the **FILTER** panel by clicking **select** to highlight them in the node tree.

You can display data **by node class**, such as CameraTracker, or explicitly **by node name**, such as Read2.
Tip: You can zoom in and out of the DATA pane using the + - buttons.

- **display frames** - allows you to view a sub-set of the profiled frames:
  - **average** - the mean average of all the frames specified by the display frames control.
  - **per frame** - allows you to step through all the frames in the display frames range using the buttons.

- **chart type** - sets the form of the output produced from the profiling data:
  - **bar** or **pie** chart, and **timeline** - displays the top 15 results by resource usage.
  - **table** - displays all results in table form, a visual representation of the .csv or .xml file output.
Exporting and Importing Profile Data

You can write the onscreen profile data to .csv or .xml file for analysis or comparison using the export data profile control on the Profile tab. You can also import existing profile data from .csv or .xml files for analysis using the import data profile control on the Profile tab.

Exporting Profile Data

To export profile data to a file:
1. Click open profile panel in the Profile node's properties or click the content menu and select Windows > Profile.

2. Enter a valid file path in the export profile data control and then click export.

Tip: You can also browse to the export location using the button.

The profile data is saved to the specified location as a .csv or .xml file.
Importing Profile Data

To import profile data from a file:

1. Click **open profile panel** in the Profile node's properties or click the content menu and select **Windows > Profile**.

2. Enter a valid file path in the **import profile data** control and then click **reload**.

   **Tip:** You can also browse to the location of the file using the button.

The profile data is imported from the specified location and displayed in the DATA panel.
Using Performance Timing

Nuke can display accurate performance timing data onscreen or output it to XML file to help you troubleshoot bottlenecks in slow scripts. When performance timing is enabled, timing information is displayed in the Node Graph, and the nodes themselves are colored according to the proportion of the total processing time spent in each one, from green (fast nodes) through to red (slow nodes).

**Note:** You can also access timing information for individual nodes using Python. See https://learn.foundry.com/nuke/developers/113/pythondevguide/performance.html for more information.

Enabling Performance Timings

You can enable profiling from the Script Editor or from the command line using the `-P` argument. You can also output the timing data to an XML file by adding the `-Pf` argument and a file name, though the output is only produced at render time.

**Note:** Enabling performance timings interactively or from the command line adds an extra processing overhead, but the data still provides an accurate picture of where processing time is spent in the script.

Outputting Performance Timings Onscreen

Enabling performance timings onscreen allows you to view cumulative data, as you work, on a per node basis.

From the Script Editor, enter:

```python
nuke.startPerformanceTimers()
```

From the command line, enter:

```bash
./Nuke<version>/Nuke<version>.exe -P
```
Note: You can reset the timing data using `nuke.resetPerformanceTimers()` from the Script Editor.

When the performance timers are active, the following information is available onscreen:

- **cpu** - the time that the CPU spent executing the processing code, in microseconds, aggregated over all CPU threads. For example, with multi-threaded processing this is typically much larger than the **wall** time. If the average CPU time per thread (cpu divided by the number of threads used) is much shorter than the **wall** time, this suggests the CPU threads have spent a lot of time not executing code and perhaps waiting on locks, which could indicate a performance problem.

Note: On Mac and Windows, the CPU time is not currently accurate. On Mac, the **cpu** value is always similar to the **wall** time.

- **wall** - the time taken as it would be measured by a clock on the wall - the actual time you have to wait for the processing to complete. The **wall** time is also measured in microseconds.
- **ops** - the number of operators called in the node. Operators are Nuke's building blocks that perform certain tasks. Nodes can contain one or more ops. For example, when a node needs to resize something it would use a Transform op rather than an implementation of its own to do the same thing.

  See [https://learn.foundry.com/nuke/developers/113/ndkdevguide/intro/terminology.html#op](https://learn.foundry.com/nuke/developers/113/ndkdevguide/intro/terminology.html#op) for more information.
- **memory** - the total amount of system memory used by the node.

In addition to the timing information, nodes are color-coded according to their profiling, green through red, where red is a slow node. You can see from the example script that Defocus is a slow node, whereas Merge is doing no work at all.
Note: You can stop displaying timing data using `nuke.stopPerformanceTimers()` in the Script Editor.

Writing Performance Timings to File

You can output the timing data to an XML file using the `-Pf` argument and a file name. The output is only produced at render time, either from a Write node or from a command line render.

On Windows, for example, the following command writes profile data to file when you render from a Write node in the Node Graph:

```
Nuke<version>\Nuke<version>.exe -Pf <file path and name>.xml
```

Rendering from the command line, the following command renders 10 frames from a script called `profileTest.nk` and writes the profile data to an XML file:

```
Nuke<version>\Nuke<version>.exe -x -Pf C:\temp\profileTest.xml C:\temp\profileTest.nk 1-10
```
File Name Search and Replace

With the Search and Replace function, you can quickly replace all or part of file names or file paths in any node with file or proxy controls (for example in Read and Write nodes).

To Search for a File Name or File Path and Replace It

1. Select the node(s) where you want to replace all or part of a file name or file path.
2. Select Edit > Node > Filename > Search and Replace.
   OR
   Press Ctrl+Shift+/ (Mac users press Cmd+Shift+/).
3. In the dialog that opens, enter the string you want to search for and the string you want to replace it with. Click OK.

Nuke searches for the string in the selected nodes and replaces it with the new string.

Note: You can also enter expressions into the Search and Replace dialog. Just remember that the search field in the dialog only takes regular expressions. Any characters that have specific meanings in regular expressions, such as [ and ], need to be preceded by the \ character. For example, [getenv HOME] would need to be entered as \[getenv HOME\]. You can also pass flags alongside the expression itself to control how the expression behaves. For example, to perform case-insensitive searches, you can enter (?i) in the beginning of the expression or after one or more whitespace characters.
Grouping Nodes in the Node Graph

You can group nodes in the Node Graph using the Backdrop node or the Group node. The Backdrop node adds a background box behind the nodes, separating the nodes visually from the rest of the node tree. A Group node, instead, combines a set of nodes into a single node, acting as a nesting container for those nodes.

Grouping Nodes with the Backdrop Node

You can use the Backdrop node to visually group nodes in the Node Graph. Inserting a Backdrop node creates a box behind the nodes. When you move the box, all the nodes that overlap the box are moved, too. By inserting several backdrop nodes, you can group the nodes in your node tree onto boxes of different colors and titles. This makes it easier to find a particular node in a large node tree, for example.

You can also use the Z Order control in the Properties panel to layer-up Backdrop nodes. Backdrops with lower Z Order values appear underneath those with a higher value.

To Group Nodes with a Backdrop Node

1. Select Other > Backdrop. A Backdrop node box appears in the Node Graph.
2. Drag the triangle in the lower right corner of the box to resize the box as necessary.
3. Click on the box title bar and drag it to move the box behind the nodes you want to group together. If there are any nodes on the box, they move together with the box.

4. To change the color of the box, open the Backdrop node's controls by double-clicking on the title bar, then click the left color button and pick a new color with the color picker that appears.

5. To change the title of the box, enter a new name for the Backdrop node in the node's controls.

6. To layer-up Backdrop nodes, enter a value in the Z Order control in the Properties panel. Backdrops with lower Z Order values appear underneath those with a higher value.
7. If you later want to remove both the box and the nodes inside it, click on the title bar and press **Delete**. **Ctrl/Cmd**+clicking on the title bar and pressing **Delete** only removes the box and leaves the nodes untouched. To remove the nodes and leave the box untouched, click on the triangle in the lower right corner of the box and press **Delete**.

**Tip:** You may want to use Backdrop nodes as jump-to points throughout a project. For more information, see [Navigating Inside the Node Graph](#).

---

### Grouping Nodes with the Group Node

You can use the Group node to nest multiple nodes inside a single node.

#### To Group Nodes with a Group Node

1. Select all the nodes you want to nest inside the Group node.
2. If you want to replace the original nodes with the Group node, right-click and select **Edit > Node > Group > Collapse To Group** (or press **Ctrl/Cmd**+G on the Node Graph).
   
   If you want to keep the original nodes in the layout in addition to the Group node, right-click and select **Edit > Node > Group > Copy Nodes To Group** (or press **Ctrl/Cmd**+Alt+Shift+G on the Node Graph).
The selected nodes are nested into a group. The internal structure of the Group node is shown on a separate tab that opens.

**Tip:** As an alternative to Edit > Node > Group > Collapse to Group, you can also select Other > Group from the Toolbar or the Node Graph right-click menu.

### To View the Nodes Nested Inside a Group Node

In the Group node's controls, click the S button (short for Show) in the top right corner.

You can also select the node and then press **Ctrl/Cmd+Enter** to open the Group.

A new tab that contains the nested nodes opens.

### To Ungroup Nodes

1. Select the Group node in the Node Graph.
2. Select Edit > Node > Group > Expand Group (or press **Ctrl/Cmd+Alt+G**).
   
   The Group node is replaced with the nodes that were nested inside it.

OR

1. In the Group node's controls, click the S button in the top right corner.
A new tab that contains the nested nodes opens.

2. Copy the nodes from the new tab into your script. If you want to lock the connections between the grouped nodes so that they cannot be accidentally disconnected during the copy-paste operation, check **lock all connections** in the Group node’s controls.

3. Delete the unnecessary Group node from your script.
Adding Notes to the Node Graph

Using the StickyNote node, you can add notes to the Node Graph. The notes can be any text or HTML mark-up. Usually, they are made as annotations to the elements in the node tree.

To Add a Note to the Node Graph

1. Click on the part of the Node Graph where you want to add a note.
2. Select Other > StickyNote. A note box appears in the Node Graph.

3. In the StickyNote controls, enter your note in the label field. If you like, you can use HTML mark-up. For example,
   • to have a note appear in bold, you can use <b>my note</b>. This would appear as my note.
   • to have a note appear in italics, you can use <i>my note</i>. This would appear as my note.
   • to add an icon to your note, you can use <img src="Colorwheel.png"/>. This adds the Nuke color wheel icon. You can also use your own icons in the same way as long as you save them in your plug-in path directory. Most common image formats work, but we recommend using .png files.
Using the Precomp Node

The Precomp node is like a Group node, but its content is stored in an independent .nk file. This allows you to save a subset of the node tree as a separate .nk script, render the output of this saved script, and read the rendered output back into the main comp as a single image input.

**Note:** Precomp nodes are unsupported in Nuke Assist. You cannot create a Precomp in Nuke Assist; however, if one already exists in the script, you can view, but not modify its output. To indicate this, Precomp nodes are outlined in red in the Node Graph, and their controls are grayed out. See Nuke Products for more information.

Precomp nodes can be useful in at least two ways. Firstly, they can be used to reduce portions of the node tree to pre-rendered image inputs. This speeds up render time, as Nuke only has to process the single image input instead of all the nodes that were used to create it. Because the original nodes are saved in a separate .nk script, you also maintain access to them and can adjust them later if necessary.

Secondly, Precomp nodes enable a collaborative workflow. While one artist works on the main comp, others can work on the sections that have been exported using the Precomp node. These sections can be edited, versioned, and managed independent of the main comp. For example, say you have a comp that involves a complex, multi-layered CG render. A 3D artist can produce this as a separate script that the compositor finishing the shot then reads in using a Precomp node. This way, the 3D artist can modify and re-render the CG element portion of the comp without having to share the main comp with the compositor.
Creating Precomp Nodes

Whether you want to use the Precomp node to speed up rendering or to enable a collaborative workflow, the process of creating the Precomp is the same.

To Create a Precomp Node

1. Select the nodes you want to include in the separate precomp script. If you select a Group node, the nodes nested inside it are copied into the precomp script.
2. Select Other > Precomp (or press Ctrl/Cmd+Shift+P on the Node Graph). The Precomp Nodes dialog opens.

Note: If you don't select any nodes when you create a Precomp node, the Precomp Nodes dialog is not displayed and the node is left blank. Using the file parameter in the Precomp controls, you can then browse to an existing script to load into the Precomp node. If you have several Write nodes in the existing script and want to control which of them is used for the output of the Precomp node, you can select Other > Output to insert an Output node after the Write node you want to use.

If the Precomp node cannot find an Output node in the script, it looks for a Write node and sets Output node in the Precomp controls to the name of that node. The Output node field can also be used later to override what is set as the Output node in the precomped script. To do so, make sure you check enable. This check box allows you to toggle between the output node chosen by default and the output node you specified in the Output node field.

3. Click the file browser icon next to Precomp script path, and browse to the directory where you want to save the precomp .nk script. After the directory path, enter a name for the precomp script, for example Precomp1_v01.nk. By default, the precomp script is saved next to the main script. If the main script has not been saved, the precomp script is saved in the current directory.
4. Click the file browser icon next to Precomp render path, and browse to the directory where you want to save the rendered output of the precomped nodes. After the directory path, enter a
name for the rendered image, for example `Precomp1_####.exr`. If you like, you can also use version numbers in the name, for example `Precomp1_v01_####.exr`.

**Warning:** We recommend rendering the image as an `.exr` because that way Nuke can write the hash value of the incoming node tree into the rendered file. If the precomp script changes so that the hashes won't match, Nuke can then notify you, and you can update the resulting image. If you use a file format other than `.exr`, you do not get this notification and the rendered file is likely to become out of date. For more information on hash values, see About the Hash Value.

5. From the **Channels** dropdown menu, select the channels you want to include in the rendered output of the precomped nodes.
   If you later need to adjust this selection, you can do so by setting the channels on the appropriate Write node (by default, Write1) in the precomp `.nk` script.

6. From the **Original nodes** dropdown menu, select:
   • **add backdrop** to create a backdrop behind the precomped nodes in the Node Graph.
   • **delete** to delete the precomped nodes.
   • **no change** to do nothing to the precomped nodes.

7. Click **OK**.
   Nuke saves the selected nodes in the `.nk` script specified. This script also includes input and output nodes, a Write node, and the current project settings.
   In the Node Graph, the selected nodes are replaced with the Precomp node. Nuke opens the properties of the Precomp node.

**About the Hash Value**

The hash value is a unique number (for example, b1c9c0aff2012a8) calculated from a node and the entire tree of nodes connected to its input. The class of the node and all the current control settings contribute to the hash value.

You can display the hash value at any point in the node tree by selecting a node in the Node Graph and pressing I. The hash can be different at different points in the tree.

**Precomp Nodes and Project Settings**

When you create a Precomp node, the precomp `.nk` script gets its project settings from the main script. The main script's project settings are also used whenever the precomp script is loaded into the main script. Therefore, if you open the precomp script in a separate instance of Nuke and change its
project settings so that they no longer match the main script's settings, your changes do NOT have an effect when the precomp script is loaded into the main script. If you want to change the project settings, you should always do so in the main script rather than the precomp script.
Using a Precomp Node to Speed-up Rendering

If you want to use a Precomp node to speed up rendering, you need to have the Precomp node read in the output of the precomp script.

To Render a Precomp Node

1. Create a Precomp node. For more information on how to do this, see Creating Precomp Nodes.
2. In the Precomp node controls, click Render. Enter a frame range (for example, 1-100) in the dialog that opens and click OK.

   Nuke renders the output of the precomp script and saves the resulting image in the Precomp render path directory you specified when you created the Precomp node.

   If you have activated the proxy mode and have specified a proxy file name in the output node, the rendered image is a proxy image.

3. If the render finishes successfully, read file for output is automatically turned on in the Precomp properties. When this is checked, the Precomp node turns green and reads in the rendered precomp image rather than calculating the output of the precomp script. The word (Read) and the name of the image read in are also displayed on the Precomp node in the Node Graph to indicate this.

4. In case there is an error opening the rendered output of the precomped nodes, select what to do from the missing frames dropdown menu in the Precomp properties:
   • error - display an error message on any missing frames.
   • black - replace any missing frames with black.
   • checkerboard - replace any missing frames with a checkerboard image.
   • read input - display the result of the input tree rather than the rendered file on any missing frames.

When several Precomp nodes are used like this to replace sections in a large, complex node tree, the render times may become significantly faster.

Note: When rendering the output of the precomp script, Nuke automatically selects the Write node to use by first looking for an Output node. If it can't find any Output nodes, it tries to find a Write node, and sets output node in the Precomp node controls to the name of the Write node. If it can't find any Output or Write nodes, it produces an error.
At any point, the Precomp node properties can be used to override what is set as the Output node in the precomped script. To do so, open the Precomp properties and the advanced controls. In the output node field, enter the name of the Write node whose output you’d like to use. Make sure you also check enable. This check box allows you to toggle between the output node chosen by default and the output node you specified in the output node field.

**Tip:** You can also use Write nodes in a similar manner and have their output read in from the rendered file rather than calculated using the nodes upstream. For more information, see Using a Write Node to Read in the Rendered Image.
Precomp Revisions

The following describes how to open, edit, and reload a precomp script.

To View and Edit a Precomp Script

1. In the Precomp node properties, click **Open**.
   This starts a new Nuke session and loads the precomp script.
2. Edit the precomp script as necessary.
3. If you are using version numbers in the Write node file name, select the Write node in the Node Graph and choose **Edit > Node > Filename > VersionUp** from the menu bar. This changes the version number in the file that’s rendered, for example, from Precomp1_v01_#####.exr to Precomp1_v02_#####.exr.
4. Render the Write node. Note that if the proxy mode is on and you have entered a proxy file name for the output node, Nuke renders a proxy image.
5. Select **File > Save New Comp Version** (or **File > Save Comp As**) to save the script with a new version number, for example, Precomp1_v02.nk instead of Precomp1_v01.nk.

To Reload a Revised Precomp Script

1. Load the main comp.
2. Make sure the filename in the Precomp node’s **file** field matches the current name of the precomp script.
   If the precomp script name has been versioned up, you can simply select the Precomp node and choose **Edit > Node > Filename > VersionUp** from the menu bar. This changes the name of the script that is read in, for example, from Precomp1_v01.nk to Precomp1_v02.nk.
   If the precomp script has been saved with a different name or you want to use a different script as the precomp script, edit the filename in the **file** field.
3. In the Precomp properties, click **Reload**.
Collaborative Workflow Example

In this example, a compositor is responsible for the main comp that is delivered to the client. A lighting TD is responsible for providing a multi-layered CG render for use in the main comp. The following describes how the compositor and lighting TD could use the Precomp node to enable a collaborative workflow.

To Enable a Collaborative Workflow

1. The compositor starts building the main comp.
2. The lighting TD does a render of the CG element and a comp of the layers, saving the script as cg_v01.nk.

⚠️ **Warning:** Note that we recommend rendering precomp images as `.exr` files. This way, Nuke writes the hash value of the incoming node tree into the rendered file. If the precomp script changes so that the hashes won't match, Nuke can notify the user who can then update the resulting image. If you use a file format other than `.exr`, you don't get this notification and the rendered file is likely to become out of date. For more information on hash values, see [Creating Precomp Nodes](#).

3. The compositor creates a Precomp node reading from the file cg_v01.nk, and continues working on the main comp.
4. The lighting TD continues to revise the CG render and the comp, versioning up the Write node and the precomp script .nk name. (See [Precomp Revisions](#).)
   - When better results are achieved, the lighting TD notifies the compositor of the new and improved version of the precomp script.
5. The compositor versions up the Precomp node to the new, improved version and reloads the precomp script. (See [Precomp Revisions](#).)
Using the LiveGroup Node

LiveGroups are a type of container node that can be used in conjunction with LiveInput nodes so that multiple artists can work on different parts of the same shot as separate scripts, without the need for rendering. LiveGroups also offer all the functionality of Precomps, Groups, and Gizmos combining all the functionality that they lack individually.

<table>
<thead>
<tr>
<th>Function</th>
<th>Group/Gizmo</th>
<th>Precomp</th>
<th>LiveGroup</th>
</tr>
</thead>
<tbody>
<tr>
<td>Open source file</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Version up and down without recreating</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Expose controls from within the node</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>View internal nodes in the same Nuke session</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Accepts inputs in the same Nuke session</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Just like Precomps, LiveGroups can store independent, open source .nk files, allowing you to save a subset of the node tree as a separate .nk script, render the output of this saved script, and read the rendered output back into the master comp as a single image input.

You can also use LiveGroups like Group nodes to nest multiple nodes inside a single node. The original nodes are replaced with the LiveGroup node. When you create a LiveGroup node, its internal structure is shown in a separate Node Graph tab.
LiveGroups can also be used as shared toolsets, similar to Gizmos, but without having to export them. You can import a LiveGroup script into any other Nuke script, as long as Project Settings > is live group is enabled and the script contains an Output node.
Collaborative Scripts

The LiveGroup node is different to Precomps and Groups in that it allows multiple artists to work on the same script at the same time, drawing work from disparate scripts into the master script. A simple example is shown below, where the **Keying**, **CameraTrack**, and **VFX** LiveGroups contain the node trees for those operations in separate scripts.

![LiveGroup Node Diagram](image)

In this example, the artists can continually improve the expression linked Camera and green-screen key while the 3D artist works on the shot. See [Collaborative Workflow Examples](#) for more information.
Collaborative Workflow Examples

LiveGroups allow several artists to work on the same master script at the same time, without having to render the separate components like Precomps. You can set up a master script before beginning or start work on individual sessions and then import them into another script, depending on how you like to work.

Creating a Master Script

As a Supervisor, you might want to set up a master script to hold the work of various artists working on a shot. In this example, we'll assume that there are Keying, Tracking, and 3D artists working simultaneously.

To create a master script:
1. Create a Nuke script with the required Project Settings and read in the required assets. See Setting Up Your Script for more information.
2. Add the basic nodes required for the Keying artist and place them in a LiveGroup. See Creating LiveGroups for more information on creating LiveGroups.

Tip: You can give the LiveGroup node a more descriptive name to make things easier to read.

3. Publish the LiveGroup to a network location accessible by the Keying artist. See Editing and Publishing LiveGroups for more information. Published LiveGroups are locked and cannot be edited unless you click Make Editable in the script.
4. Repeat steps 2 and 3 for the Tracking and 3D artists. An example is shown in the image.
5. As the sub-script work is completed, you can then add the required nodes to complete the master script.

For example, you can expression link the Camera data from the CameraTracking LiveGroup to a Camera in the master script. See Managing LiveGroup Controls for more information.

6. As the sub-script artists version up their scripts, you can version up your LiveGroups in the master script to read in the latest work without the need to re-render in the master script. See Versioning LiveGroup Scripts for more information.
Combining Existing Scripts

As an artist, you might be working on a Nuke script in isolation that needs to be incorporated into a master script at a later date. LiveGroups allow you to point to a sub-script in the same way as a Precomp, but with the added functionality of LiveGroups.

To import a sub-script using a LiveGroup:
1. Open up the master script and add a LiveGroup node. The VFX LiveGroup referencing `particles_v02.nk` in the example.
2. Double-click the LiveGroup to open its Properties panel.
3. Enter the location of the sub-script in the file control.
4. Click Reload.

This example differs from creating a master script first because the VFX LiveGroup is self-contained, meaning that the assets are inside the LiveGroup rather than in the master script.

You can also work the other way, with LiveInputs referencing LiveGroups, but there are a few caveats to working this way. See Referencing a Master Script Using LiveInputs for more information.
Creating LiveGroups

You can create empty LiveGroups to read in existing .nk scripts or select a set of nodes in the Node Graph and create a LiveGroup to contain those nodes, with the option to write them out to a new .nk script.

**Note:** If you create and publish a LiveGroup Pythonically, you need to add xpos and ypos values to the nodes in the Node Graph to position them correctly before you publish the LiveGroup. See Nuke's Python Developer's Guide for more information.

Importing an Existing Script into a LiveGroup

1. In Nuke's Toolbar, select Other > LiveGroup.
   A new Node Graph tab opens showing the empty LiveGroup.
2. Enter the location of the .nk script in the file control or browse to its location and click Open.
   A warning displays, informing you that there are differences between the local script and the saved script.
3. Click Yes to overwrite the local script with the saved script.
   The LiveGroup icon turns gray and the LiveGroup tab updates to show the contents of the imported .nk script. The LiveGroup is locked as indicated by the padlock on the LiveGroup's tab and it behaves like regular Nuke Precomp node.
Adding Nodes to a LiveGroup

1. Select all the nodes you want to nest inside the LiveGroup.
2. Do one of the following:
   • Press Tab in the Node Graph, type LiveGroup and press enter,
   • Press Ctrl/Cmd+L, or
   • Right-click and select Edit > Node > Group > Collapse To LiveGroup.

The LiveGroup icon turns yellow and a LiveGroup tab shows the contents of the node. The LiveGroup is not locked and it behaves like regular Nuke Group node.
Viewing Nodes Inside a LiveGroup

In the LiveGroup's **Properties** panel, click the **S** button in the top-right corner.

A new Node Graph tab opens containing the nested nodes. LiveGroups contain a LiveInput and Output node by default, which points to the master script so that artists can collaborate without depending on Read nodes in the LiveGroup.

See [Referencing a Master Script Using LiveInputs](#) below for more information.

Referencing a Master Script Using LiveInputs

The LiveInput node can reference any LiveGroup within a script, eliminating the need to include the required assets in the sub-script. The only requirements are that the sub-script and master script have been saved and the LiveGroups you intend to connect to are pointing at the new sub-script.

1. Create a LiveInput node in the sub-script and then save the script.
2. In the master script, create a LiveGroup node and enter the sub-script name in the **file** control.
3. Save the master script.
4. In the sub-script, enter the master script name in the file control and click **Reload**.
5. Select the LiveGroup you want to reference from the **liveGroup** dropdown.
Managing LiveGroup Controls

You can expose controls from within your LiveGroup by picking existing knobs from nested nodes or by adding a new control, and even expression link these controls so that updates in the LiveGroup are passed to the master script.

Note: You cannot add or remove controls when a LiveGroup is locked. To unlock a LiveGroup, open the Properties panel LiveGroup tab and click Make Editable.

Picking and Adding Controls

LiveGroups support the same method as gizmos for exposing controls in the Properties panel. Any control from a nested node within the LiveGroup can be exposed or you can add your own.

In Nuke's Properties panel, you can add controls using the Edit button or by right-clicking in the panel and selecting Adding Knobs Manually. Drag-and-drop knobs significantly reduce the time spent managing user knobs in a node's Properties panel when compared to the legacy Manage User Knobs method within Nuke.

To add knobs to a LiveGroup:
1. Double-click the LiveGroup to open its Properties panel.
2. Double-click the node containing the knobs you want to expose to open its Properties panel.
3. Click the edit button to enable drag-and-drop functionality.
4. Drag the target knob or knobs from the source node's Properties and drop them onto the LiveGroup's Properties.
5. Click the edit button to disable drag-and-drop functionality when you've added the required knobs.

You can also expression link User knobs just like any other knob in Nuke, but with LiveGroups, you can expression link knobs from sub-scripts as well. See Expression Linking Controls for more information.

Expression Linking Controls

LiveGroups support expression links between the master and sub-scripts, allowing you to easily update exposed control values that you know are likely to change, such as Camera transforms.

To expression link a sub-script control:
1. Double-click the LiveGroup containing the controls you want to link and navigate to the User tab.
2. Double-click the node you want to link the controls to, in this case a Camera.
3. Hold **Ctrl/Cmd** and drag the animation button from the LiveGroup to the node's control properties.

4. Repeat the process for all the controls you want to link from the sub-script. The Node Graph updates to show a green arrow representing the expression link and the direction of information flow.

Now, whenever the **CameraTrack** LiveGroup is reloaded, any updates to the camera translation and rotation are automatically applied in the master script.
Editing and Publishing LiveGroups

The state of a LiveGroup in the Node Graph controls how the master script and sub-scripts behave. LiveGroups have five states:

<table>
<thead>
<tr>
<th>State</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Editable</strong></td>
<td>The LiveGroup is unlocked, but no edits have been made to the LiveGroup.</td>
</tr>
<tr>
<td><strong>Edited</strong></td>
<td>The LiveGroup has been edited in the current Nuke session.</td>
</tr>
<tr>
<td><strong>Edited with Overrides</strong></td>
<td>The LiveGroup has been edited in the current Nuke session and that the edits are overriding controls in the saved version of the LiveGroup.</td>
</tr>
<tr>
<td><strong>Published</strong></td>
<td>The LiveGroup has been saved to the location specified in the file control. LiveGroups in this state cannot be edited, but you can still adjust any controls exposed in the Properties.</td>
</tr>
<tr>
<td><strong>Published with Overrides</strong></td>
<td>The LiveGroup has been saved to the location specified in the file control and that one or more controls exposed in the Properties are overriding controls in the saved version of the LiveGroup.</td>
</tr>
</tbody>
</table>

See Overriding LiveGroup Controls for more information.

Editing LiveGroups

The **Editable** and **Edited** states apply to new LiveGroups and LiveGroups that you have edited in the current Nuke session.

**Note:** You cannot add or remove controls when a LiveGroup is locked. To unlock a LiveGroup, open the Properties panel LiveGroup tab and click Make Editable.
A LiveGroup in the **Edited** state always writes its contents into the current script on save. If its **file** control points to a sub-script, no changes are made in the sub-script until the LiveGroup is **Published** to that location.

To change the state of a published LiveGroup to **Editable** without editing it:
1. Double-click the LiveGroup node to open its **Properties**.
2. Click **Make Editable**.

The LiveGroup's state changes to **Editable** in the Node Graph and the contents of the group are added into the current script.

---

**Versioning LiveGroup Scripts**

Versions allow you to record the different stages of your workflow as you progress, quickly swapping the contents of LiveGroups in and out without overwriting existing work. You can publish different versions of LiveGroups in the same way as Nuke scripts.

To load a new version of a LiveGroup:
1. Select the required LiveGroup in the Node Graph, 
   OR
   Double-click the LiveGroup to open its **Properties**.
2. Press **Alt+up arrow** or **Alt+down arrow** to load the next version.
   If no version of the current LiveGroup exists, Nuke displays a warning.

To save a new version of a LiveGroup:
1. Double-click the LiveGroup to open its **Properties**.
2. Enter the new version number in the **file** control,
OR
Press Alt+up arrow to increment the existing version number.

Note: Nuke displays a warning that the new version doesn't exist.

3. Click Publish to save the LiveGroup to the new version.

Publishing LiveGroups

The Published state only applies to LiveGroups that you have published to the location specified in the file control.

LiveGroups in the Published state never write their contents into the current script on save. To make changes to a Published LiveGroup, click Make Editable. It becomes Editable until you publish it again by clicking the Publish button.

Note: If you create and publish a LiveGroup Pythonically, you need to add xpos and ypos values to the nodes in the Node Graph to position them correctly before you publish the LiveGroup. See Nuke’s Python Developer’s Guide for more information.

To Publish a LiveGroup:
1. Double-click the LiveGroup node to open its Properties.
2. Check that the file control points to the required location and version up the file path if required.
3. Click Publish.

   The LiveGroup’s state changes to Published in the Node Graph and the sub-script specified in the file control is updated.
**Note:** When you **Publish** a LiveGroup, the resulting `.nk` script retains the project settings from the master script. The master script's **Project Settings** are also used whenever the sub-script is loaded into the master script.

If you open the LiveGroup script in a separate instance of Nuke and change its **Project Settings** so that they no longer match the master script's settings, your changes **DO NOT** apply when the LiveGroup script is loaded into the master script. If you want to change the **Project Settings**, you should always do so in the master script rather than the sub-script.
Overriding LiveGroup Controls

Published LiveGroups cannot be edited, unless you click Make Editable in the LiveGroups Properties, but the exposed controls can be overridden in scripts that include the LiveGroup. Override controls are marked with a yellow chip in the Properties panel, so you can quickly see which controls are affecting the output.

**Note:** Override controls are also indicated on the LiveGroup node in the Node Graph.

In the example, the current color and channels controls are overrides. The double yellow chip indicates that one of the controls on the top line is an override, in this case the current color control.

Click the yellow chip to list the overridden controls on that line.
Overrides can be reverted or applied to the LiveGroup controls, if the LiveGroup is in the editable state.

To revert overrides to the values stored in the LiveGroup:
1. Open the LiveGroup Properties panel by double-clicking the node.
2. Click the custom tab on which the override controls are stored. In the example, the User tab.
3. Right-click the panel, making sure you’re not hovering over a control, and select **Revert overrides to LiveGroup values.**

The controls revert to the values stored in the LiveGroup and the overrides are removed.
To apply the overrides to the LiveGroup controls:
1. Open the LiveGroup **Properties** panel by double-clicking the node.
2. Click **Make Editable** to unlock the LiveGroup.
3. Click the custom tab on which the override controls are stored. In the example, the **User** tab.
4. Right-click the panel, making sure you’re not hovering over a control, and select **Apply override values to LiveGroup**.

![LiveGroup panel with custom tab selected](image)

The override values are applied to the LiveGroup’s controls and the overrides are removed.
5. Click **Publish** to overwrite the LiveGroup on disk or save a new version of the LiveGroup.
Configuring Nuke

These pages show visual effects supervisors how to configure Nuke for multiple artists, prior to the start of a project. These are the common application settings discussed:

• Command line operations
• Environment variables

• Gizmo, NDK plug-in, and Tcl script directories
• Python script directories
• OFX plug-in directories
• Favorite directories
• Cross-platform file paths
• Menu and Toolbar options
• Image formats
• Gizmos (Nuke group nodes or subscripts that allow only select modifications)
• Custom plug-ins (binary plug-ins made via the Nuke software developers kit)
• Generic Tcl ("Tickle") scripts
• Template scripts
• Common preferences
• Script's lookup tables (LUTs)
• Custom Viewer Processes

Note: If you copy and paste Python example scripts from this user guide into a text editor, line indentations may not be preserved. If this is the case, correct the indentations manually.
What Is a Terminal and How Do I Use One?

Many tasks in this chapter tell you to enter commands from a terminal or shell. This refers to a window where you can enter commands directly rather than making selections through a user interface.

The following describes how to open such a window for your operating system.

- **Linux**: Click the right mouse button over the desktop and select New Terminal (or Open Terminal) from the right-click menu.
- **Windows**: From the Start menu, select All Programs > Accessories > Command Prompt.
- **Mac**: Click on the Terminal dock icon.

Or

Browse to the Applications > Utilities folder on your system hard drive, and double-click the Terminal icon.

Inside the terminal or shell, you'll see a command prompt, which looks similar to this:

![Terminal prompt on Linux and Mac](image1.png) ![Command prompt for Windows](image2.png)

Once you see the command prompt, you can enter commands to perform various tasks like listing the files in a directory or running programs. Here are some specific examples:

- On Linux or Mac, type `pwd` and press Enter to view the path of the current directory. On Windows, the equivalent command would be `cd`.
- On Linux or Mac, type `ls` and press Enter to view a list of files in the current directory. On Windows, the equivalent command would be `dir`.
- On Linux, Mac, and Windows, type `cd` followed by a full path name and press Enter to change directories.
Command Line Operations

Command line flags activate various options when you launch Nuke from a command line or Terminal, and provide additional functionality to Nuke. First let's discuss how to launch Nuke from a shell.

On Windows

- **Note:** If you're using a terminal emulator, follow the steps under Mac or Linux, substituting the file paths as required.

Open a command line prompt and change directory as follows:

```
cd C:\Program Files\Nuke11.3v6\n```

To launch Nuke, type this command:

```
Nuke11.3.exe
```

Alternatively, you can set a **doskey** to point to Nuke and then you can launch Nuke from any directory:

```
doskey nuke="C:\Program Files\Nuke11.3v6\Nuke11.3.exe"
```

If you want to doskey NukeX, enter:

```
doskey nukex="C:\Program Files\Nuke11.3v6\Nuke11.3.exe" --nukex
```

If you want to doskey Nuke Studio, enter:

```
doskey nukes="C:\Program Files\Nuke11.3v6\Nuke11.3.exe" --studio
```

On Mac

Open a command line prompt and change directory as follows:

```
cd /Applications/Nuke11.3v6/Nuke11.3v6.app/Contents/MacOS/
```

To launch Nuke, type this command:

```
./Nuke11.3v6
```

Alternatively, you can set an alias to point to Nuke and then you can launch Nuke from any directory. The procedure for this depends on what your default shell is. To get the name of the shell you are using, launch Terminal and enter `echo $SHELL`. 
If you are using a bash shell, enter:

Alternatively, if you are using a tcsh shell, enter:

If you want to alias NukeX, enter:

If you want to alias Nuke Studio, enter:

**Tip:** You can add aliases to a `.cshrc` or `.bashrc` file in your home directory so that they are activated each time you open a shell. See your Systems Administrator for help setting this up.

---

**On Linux**

Open a command line prompt and change directory as follows:
cd /usr/local/Nuke11.3v6/

To launch Nuke, type this command:
./Nuke11.3

Alternatively, you can set an alias to point to Nuke and then you can launch Nuke from any directory. The procedure for this depends on what your default shell is. To get the name of the shell you are using, launch Terminal and enter `echo $SHELL`.

If you are using a bash shell, enter:
alias nuke='/usr/local/Nuke11.3v6/Nuke11.3'

Alternatively, if you are using a tcsh shell, enter:
alias nuke=/usr/local/Nuke11.3v6/Nuke11.3

If you want to alias NukeX, enter:
alias nukex=/usr/local/Nuke11.3v6/Nuke11.3 --nukex

If you want to alias Nuke Studio, enter:
alias nukes=/usr/local/Nuke11.3v6/Nuke11.3 --studio
Tip: You can add aliases to a .cshrc or .bashrc file in your home directory so that they are activated each time you open a shell. See your Systems Administrator for help setting this up.

Using command line Flags

Now you can start experimenting with command line flags on launching Nuke. Here's one that displays the version number and build date.

```bash
nuke -version
```

If you have an .nk script, you can render it on the command line without opening the GUI version. Here's an example that renders a hundred frames of a Nuke script:

```bash
nuke -F 1-100 -x myscript.nk
```

Note how you can use the -F switch on the command line to indicate a frame range, separating the starting and ending frames with a dash.

Note: We recommend that you use the -F switch whenever defining a frame range on the command line, which must precede the script name argument. However, for backwards compatibility, you can also use the old syntax. To do so, place the frame range at the end of the command and use a comma to separate the starting and ending frames. For example:

```bash
nuke -x myscript.nk 1,100
```

To display a list of command line flags (switches) available to you, use the following command:

```bash
nuke -help
```

Here's that list of command line flags in a table:

<table>
<thead>
<tr>
<th>Switch/Flag</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>-a</td>
<td>Formats default to anamorphic.</td>
</tr>
<tr>
<td>-c size (k, M, or G)</td>
<td>Limit the cache memory usage, where size equals a number in bytes. You can specify a different unit by appending k (kilobytes), M (megabytes), or G (gigabytes) after size.</td>
</tr>
<tr>
<td>--cont</td>
<td>Nuke attempts to render subsequent frames in the specified range after</td>
</tr>
<tr>
<td>Switch/Flag</td>
<td>Action</td>
</tr>
<tr>
<td>------------------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>-d &lt;x server name&gt;</td>
<td>This allows Nuke to be viewed on one machine while run on another. (Linux only and requires some setting up to allow remote access to the X Server on the target machine).</td>
</tr>
<tr>
<td>-f</td>
<td>Open Nuke script at full resolution. Scripts that have been saved displaying proxy images can be opened to show the full resolution image using this flag. See also -p.</td>
</tr>
<tr>
<td>-F</td>
<td>Frame numbers to execute the script for. All -F arguments must precede the script name argument. Here are some examples:</td>
</tr>
<tr>
<td></td>
<td>-F 3 indicates frame 3.</td>
</tr>
<tr>
<td></td>
<td>-F 1-10 indicates frames 1, 2, 3, 4, 5, 6, 7, 8, 9, and 10.</td>
</tr>
<tr>
<td></td>
<td>-F 1-10x2 indicates frames 1, 3, 5, 7, and 9.</td>
</tr>
<tr>
<td></td>
<td>You can also use multiple frame ranges:</td>
</tr>
<tr>
<td></td>
<td>nuke -F 1-5 -F 10 -F 30-50x2 -x myscript.nk</td>
</tr>
<tr>
<td>--gpu ARG</td>
<td>When set, enables GPU usage in terminal mode with an optional GPU index argument, which defaults to 0. Use --gpulist to display the selectable GPUs.</td>
</tr>
</tbody>
</table>

Tip: You can also use the NUKE_CRASH_HANDLING environment variable to control crash handling. See Environment Variables for more information.
<table>
<thead>
<tr>
<th>Switch/Flag</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>--gulist</td>
<td>Prints the selectable GPUs and their corresponding index for use with the --gpu ARG option.</td>
</tr>
<tr>
<td>-h</td>
<td>Display command line help.</td>
</tr>
<tr>
<td>-help</td>
<td>Display command line help.</td>
</tr>
<tr>
<td>-i</td>
<td>Use an interactive (nuke_i) RLM license key. This flag is used in conjunction with background rendering scripts using -x. By default -x uses a nuke_r license key, but -xi background renders using a nuke_i license key.</td>
</tr>
<tr>
<td>-l</td>
<td>New read or write nodes have the colorspace set to linear rather than default.</td>
</tr>
<tr>
<td>-m #</td>
<td>Set the number of threads to the value specified by #.</td>
</tr>
<tr>
<td>--multigpu</td>
<td>If have multiple GPUs of the same type installed, you can enable this preference to share work between the available GPUs for extra processing speed. This is a global preference and is applied to all GPU enabled nodes. See Windows, Mac OS X and macOS, or Linux for more information on the GPUs Nuke supports.</td>
</tr>
<tr>
<td>-n</td>
<td>Open script without postage stamps on nodes.</td>
</tr>
<tr>
<td>--nocrashprompt</td>
<td>When crash handling is enabled in GUI mode, submit crash reports automatically without displaying a crash reporter dialog.</td>
</tr>
</tbody>
</table>

**Note:** Overrides the GPU set in Preferences > Performance > Hardware when run in interactive mode.

**Tip:** You can also use the NUKE_NO_CRASH_PROMPT environment variable to control the crash prompt. See sales@foundry.com to obtain a replacement license.
<table>
<thead>
<tr>
<th>Switch/Flag</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>--nukeassist</td>
<td>Launch Nuke Assist, which is licensed as part of a NukeX Maintenance package and is intended for use as a workstation for artists performing painting, rotoscoping, and tracking. Two complimentary licenses are included with every NukeX license.</td>
</tr>
<tr>
<td>-p</td>
<td>Open Nuke script at proxy resolution. Scripts that have been saved displaying full resolution images can be opened to show the proxy resolution image using this flag. See also -f.</td>
</tr>
<tr>
<td>-P</td>
<td>Measure your nodes’ performance metrics and show them in the Node Graph. See Using Performance Timing for more information.</td>
</tr>
<tr>
<td>--pause</td>
<td>Initial Viewers in the script specified on the command line should be paused.</td>
</tr>
<tr>
<td>-Pf &lt;filename&gt;</td>
<td>Measure your nodes’ performance metrics and write them to an XML file at render time. See Using Performance Timing for more information.</td>
</tr>
<tr>
<td>--priority p</td>
<td>Runs Nuke with a different priority, you can choose from:</td>
</tr>
<tr>
<td></td>
<td>• high (only available to the super user on Linux/OS X)</td>
</tr>
<tr>
<td></td>
<td>• medium</td>
</tr>
<tr>
<td></td>
<td>• low</td>
</tr>
<tr>
<td>--python-no-root-knobdefaults</td>
<td>Prevents the application of knob defaults to the root node when executing a Python script.</td>
</tr>
<tr>
<td>-q</td>
<td>Quiet mode. This stops all printing to the shell.</td>
</tr>
<tr>
<td>--remap</td>
<td>Allows you to remap file paths in order to easily share Nuke projects across different operating systems. This is the command line equivalent of setting the Path Remaps control in the Preferences dialog.</td>
</tr>
<tr>
<td></td>
<td>The --remap flag takes a comma-separated list of paths as an argument. The paths are arranged in pairs where the first path of each pair maps to</td>
</tr>
</tbody>
</table>
the second path of each pair. For example, if you use:

nuke -t --remap "X:/path,Y:,A:,B:/anotherpath"

• Any paths starting with X:/path are converted to start with Y:
• Any paths starting with A: are converted to start with B:/anotherpath

The --remap flag throws an error if:
• it is defined when starting GUI mode, that is, without -x or -t
• the paths do not pair up. For example, if you use:

nuke -t --remap "X:/path,Y:,A:"

A: does not map to anything, and an error is produced.

The --remap flag gives a warning (but does not error) if you give it no paths. For example:

nuke -t --remap ""

Note: Note that the mappings are only applied to the Nuke session that is being started. They do not affect the Preferences.nk file used by the GUI.

-s #
Sets the stack size per thread, in bytes. The default value is 16777216 (16 MB). The smallest allowed value is 1048576 (1 MB).

None of Nuke's default nodes require more than the default memory stack value, but if you have written a custom node that requests a large stack from the memory buffer, increasing the stack size can prevent stack overflow errors.

--safe
Running Nuke in safe mode stops the following loading at startup:
• Any scripts or plug-ins in ~/.nuke
• Any scripts or plug-ins in $NUKE_PATH or %NUKE_PATH%
• Any OFX plug-in (including FurnaceCore)

--sro
Forces Nuke to obey the render order of Write nodes so that Read nodes can use files created by earlier Write nodes.
<table>
<thead>
<tr>
<th>Switch/Flag</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>-t</td>
<td>Terminal mode (without GUI). This allows you to enter Python commands without launching the GUI. A &gt;&gt;&gt; command prompt is displayed during this mode. Enter quit() to exit this mode and return to the shell prompt. This mode uses a nuke_r license key by default, but you can get it to use a nuke_i key by using the -ti flag combo.</td>
</tr>
<tr>
<td>--tg</td>
<td>Terminal Mode. This also starts a QApplication so that Pyside/PyQt can be used. This mode uses an interactive license, and on Linux requires an X Windows display session.</td>
</tr>
</tbody>
</table>
| -V level    | Verbose mode. In the terminal, you'll see explicit commands as each action is performed in Nuke. Specify the level to print more in the Terminal, select from:  
  • 0 (not verbose)  
  • 1 (outputs Nuke script load and save)  
  • 2 (outputs loading plug-ins, Python, Tcl, Nuke scripts, progress and buffer reports) |
| -v          | This command displays an image file inside a Nuke Viewer. Here's an example:  
  nuke -v image.tif |
| --view v    | Only execute the specified views. For multiple views, use a comma separated list:  
  left,right |
| --version   | Display the version information in the shell.                                                                                             |
| -x          | eXecute mode. Takes a Nuke script and renders all active Write nodes.  
  
  **Note:** Nuke Non-commercial is restricted to encrypted .nknc scripts with -x from the command line, that is, using the syntax:  
  nuke -x myscript.nknc  
  
  This mode uses a nuke_r license key. To use a nuke_i license key, use -xi. This is the syntax:  
  nuke -xi myscript.nk |
Switch/Flag | Action
---|---
| On Windows, you can press **Ctrl+Break** to cancel a render without exiting if a render is active, or exit if not. **Ctrl/Cmd+C** exits immediately. On Mac and Linux, **Ctrl/Cmd+C** always exits.

---

**Note:** If you still use FLEXlm licenses and you're interested in making a move to RLM licensing, please contact sales@foundry.com to obtain a replacement license.

<table>
<thead>
<tr>
<th>-X node</th>
<th>Render only the Write node specified by node.</th>
</tr>
</thead>
<tbody>
<tr>
<td>--</td>
<td>End switches, allowing script to start with a dash or be just - to read from stdin</td>
</tr>
</tbody>
</table>

**General syntax**

This is the general syntax for using these options when launching Nuke at the command prompt:

```plaintext
nuke <switches> <script> <argv> <ranges>
```

- `<switches>` - modifies the behavior of Nuke when run from the command line. A list of switches is given in the table above. These are sometimes called flags.

- `<script>` - the name of the Nuke script.

- `<argv>` - an optional argument that can be used in Nuke. See the example below.

- `<ranges>` - this is the frame range you want rendering.

**Examples**

Let's consider some practical examples.

To launch Nuke and open a script.

```plaintext
nuke myscript.nk
```

Crazy I know, but I've called my script, -myscript.nk, and the hyphen at the start of the filename has confused Nuke. To get round this if you don't want to rename your file use the double hyphen syntax:

```plaintext
nuke -- -myscript.nk
```
To display an image:
nuke -v polarbear.tif

To display several images:
nuke -v polarbear.tif whiteflower.psd mountains.cin

To display an image sequence (taxi.0001.tif, taxi.0002.tif,...,taxi.0050.tif):
nuke -v taxi.####.tif 1-50

To render frame 5 of a Nuke script:
nuke -F 5 -x myscript.nk

To render frames 30 to 50 of a Nuke script:
nuke -F 30-50 -x myscript.nk

To render two frame ranges, 10-20 and 34-60, of a Nuke script:
nuke -F 10-20 -F 34-60 -x myscript.nk

To render every tenth frame of a 50 frame sequence of a Nuke script:
nuke -F 1-50x10 -x myscript.nk

This renders frames 1, 11, 21, 31, 41.

In a script with two write nodes called WriteBlur and WriteInvert this command just renders frames 1 to 20 from the WriteBlur node:
nuke -X WriteBlur myscript.nk 1-20

**Using [argv 0]**

Let's use [argv] to vary the output file name. Launch the GUI version of Nuke and create a node tree that puts a checker into a Write node. Open the write node property panel by double clicking on it and in the file text field enter this filename:

[argv 0].####.tif

Save the script and quit Nuke. On the command line type:
nuke -x myscript.nk mychecker 1-5

This renders 5 frames (mychecker.0001.tif, mychecker.0002.tif, etc.).

You can add another variable to control the output image file type. The file text field needs this:

[argv 0].####.[argv 1]

and then render the script using this command:
nuke -x myscript.nk mychecker cin 1-5

to get mychecker.0001.cin, mychecker.0002.cin, etc.

The <argv> string can be any [argv n] expression to provide variable arguments to the script. These must be placed between the <script> and the <ranges> on the command line. You can include multiple expressions, but each must begin with a non-numeric character to avoid confusion with the frame range control. For more information on expressions, see Expressions.

Using Python to convert TIFFs to JPEGs

This command line method converts 5 TIFF frames to JPEG.

```python
nuke -t
>>> r = nuke.nodes.Read(file = "myimage.####.tif")
>>> w = nuke.nodes.Write(file = "myimage.####.jpg")
>>> w.setInput(0, r)
>>> nuke.execute("Write1", 1,5)
>>> quit()
```

It's a bit tedious typing these commands in line by line. So let's put them in a text file called imageconvert.py and get Nuke to execute the Python script.

```python
cat imageconvert.py
r = nuke.nodes.Read(file = "myimage.####.tif")
w = nuke.nodes.Write(file = "myimage.####.jpg")
w.setInput(0, r)
nuke.execute("Write1", 1,5)
nuke -t < imageconvert.py
```

You can also pass in the Python script as a command line parameter. Doing this allows you to enter additional parameters after the script name to pass into your script. When you do so, note that sys.argv[0] is the name of the Python script being executed, and argv[1:] are the other parameters you passed in. One example of this is below. See the standard Python module optparse for other ways to parse parameters.

```python
cat imageconvertwithargs.py
import sys
r = nuke.nodes.Read(file = sys.argv[1])
w = nuke.nodes.Write(file = sys.argv[2])
w.setInput(0, r)
nuke.execute("Write1", 1, 5)
nuke -t imageconvertwithargs.py myimage.####.tif myimage.####.jpg
```
Environment Variables

Environment variables are named variables used to store a value, such as a specific file path. They can be used to influence Nuke’s behavior. For example, Nuke uses the information stored in them to define where to place certain files.

Setting Environment Variables

This section teaches you how to set environment variables, check if a particular environment variable exists, and display a list of set environment variables.

On Windows

1. Right-click on My Computer and select Properties.
2. Go to the Advanced tab.
3. Click the Environment Variables button. The Environment Variables dialog opens.
4. Click the New button under either User variables or System variables, depending on whether you want to set the variable for the current user or all users. To set environment variables for all users, you need to have administrator privileges.
5. In the Variable name field, enter the name of the environment variable you want to set. For a list of the environment variables that Nuke understands, see Nuke Environment Variables.
6. In the Variable value field, enter the value for the variable. The value can be a directory path, for example.
7. Click OK.

Note: When editing existing system variables, or adding or deleting either user or system variables, you may need to log off and on again before your changes to environment variables take effect.

On Mac

On Mac, you can use the launchd.conf file to set environment variables. You may need to create the launchd.conf file if it doesn't already exist in the /etc/ directory.
Environment variables set using the `launchd.conf` file are read both when Nuke is launched from the Nuke icon and when it's launched from the Terminal.

1. Open a Terminal window.
2. Create the `/etc/launchd.conf` file, if it doesn't already exist, and then add the environment variable(s) and value(s) to the file using the following format:
   ```
   setenv <VARIABLE> <VALUE>
   ```
   For example, to set two environment variables, NUKE_PATH and OFX_PLUGIN_PATH, to point to alternate locations:
   ```
   setenv NUKE_PATH /SharedDisk/Nuke/
   setenv OFX_PLUGIN_PATH /SharedDisk/OFX
   ```
   For a list of the environment variables that Nuke understands, see Nuke Environment Variables.

3. To force the OS to read the `launchd.conf` file at startup, enter:
   ```
   launchctl < /etc/launchd.conf; sudo launchctl < /etc/launchd.conf
   ```
4. Restart your Mac to apply the changes.

**On Linux**

1. The procedure for setting an environment variable depends on what your default shell is. To get the name of the shell you are using, launch a shell and enter `echo $SHELL`.
2. Depending on the output of the previous step, do one of the following:
   - If your shell is a csh or tcsh shell, add the following command to the `.cshrc` or `.tcshrc` file in your home directory: `setenv VARIABLE value`. Replace `VARIABLE` with the name of the environment variable and `value` with the value you want to give it, for example `setenv NUKE_PATH /SharedDisk/Nuke`.
   - If your shell is a bash or ksh shell, add the following command to the `.bashrc` or `.kshrc` file in your home directory: `export VARIABLE=value`. Replace `VARIABLE` with the name of the environment variable and `value` with the value you want to give it, for example `export NUKE_PATH=/SharedDisk/Nuke`.

**Note:** If you only need to set an environment variable for a single session, or you don't want to use the Mac `launchd.conf` file, you can also set variables using the method described in the On Linux section.

**Tip:** A handy command line tool for creating and editing files in the Terminal is `nano`. To start `nano` with the correct permissions, enter:
```
sudo nano /etc/launchd.conf
```
To Check if an Environment Variable Exists

From Inside Nuke

• Press X in the Node Graph, check that TCL is enabled, and enter:

  getenv <VARIABLE>

OR

• Open the Script Editor and enter:

  import os
  print os.environ["VARIABLE"]

In both cases, VARIABLE should be replaced by the environment variable you’re interested in. For example, NUKE_TEMP_DIR on Windows returns:
C:/Users/<current_user>/AppData/Local/Temp/nuke

In the Windows Environment

1. Select Start > All Programs > Accessories > Command Prompt.
2. In the command window that opens, enter echo %VARIABLE%. Replace VARIABLE with the name of the environment variable. For example, to check if NUKE_DISK_CACHE is set, enter echo %NUKE_DISK_CACHE%.

If the variable is set, its value is displayed in the command window.

In the Mac or Linux Environment

1. Launch Terminal or a shell.
2. Enter echo $VARIABLE. Replace VARIABLE with the name of the environment variable. For example, to check if NUKE_DISK_CACHE is set, enter echo $NUKE_DISK_CACHE.

If the variable is set, its value is displayed in the Terminal or shell window.

To Display a List of Set Environment Variables

On Windows

1. Select Start > All Programs > Accessories > Command Prompt.

For a list of the environment variables that Nuke understands, see Nuke Environment Variables.
2. In the command window that opens, enter `set`.

A list of all the environment variables that are set is displayed in the command window.

**On Mac or Linux**

1. Launch Terminal or a shell.
2. Enter `printenv`.

A list of all the environment variables that are set is displayed in the Terminal or shell window.
Nuke Environment Variables

The following table lists the environment variables Nuke recognizes.

<table>
<thead>
<tr>
<th>Environment Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FN_CRASH_DUMP_PATH</td>
<td>Allows you to specify where Issue Reporter dumps are saved by default.</td>
</tr>
<tr>
<td>FN_DISABLE_LICENSE_DIALOOG</td>
<td>By default, if you have installed a temporary license, Nuke displays a dialog at start-up alerting you to the number of days remaining. If you want to disable this behavior, you can set either of these environment variables to 1 to suppress the warning message about imminent license expiration.</td>
</tr>
<tr>
<td>FN_DISABLE_LICENSE_DIALOG or FN_Nuke_DISABLE_TMPPLIC_NOTIFY_DIALOG</td>
<td>By default, if you have installed a temporary license, Nuke displays a dialog at start-up alerting you to the number of days remaining. If you want to disable this behavior until a set number of days before expiry, you can set this environment variables to the required number of days.</td>
</tr>
<tr>
<td>FN_LICENSE_DIALOG_DAYS_LEFT_BEFORE_PROMPT</td>
<td>By default, if you have installed a temporary license, Nuke displays a dialog at start-up alerting you to the number of days remaining. If you want to disable this behavior until a set number of days before expiry, you can set this environment variables to the required number of days.</td>
</tr>
<tr>
<td>FN_Nuke_DISABLE_GPU_ACCELERATION</td>
<td>This variable disables Nuke's CUDA and OpenCL capabilities. When enabled, any GPUs installed locally are disabled and cannot be selected from Preferences &gt; Performance &gt; Hardware &gt; default blink device dropdown. Any GPU accelerated nodes, such as Kronos and Denoise, default to processing on the CPU.</td>
</tr>
<tr>
<td>FN_SUBSCRIPTION_LICENSE_DIR</td>
<td>On Windows, user names containing non-ASCII characters can cause subscription licensing to fail. If a licensing error similar to the following displays: &quot;Unable to create subscription license directory: C:\Users\Zoë&quot;</td>
</tr>
<tr>
<td>Environment Variable</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Hernández\FoundryLicensing \</td>
<td>Try changing the license directory to an alternate location using this environment variable.</td>
</tr>
<tr>
<td>foundry_LICENSE</td>
<td>The location of the Nuke RLM license file, if the following recommended location is not used:</td>
</tr>
<tr>
<td></td>
<td><strong>On Mac and Linux:</strong></td>
</tr>
<tr>
<td></td>
<td>/usr/local/foundry/RLM</td>
</tr>
<tr>
<td></td>
<td><strong>On Windows:</strong></td>
</tr>
<tr>
<td></td>
<td>drive letter:\Program Files\The Foundry\RLM</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> If you still use FLEXlm licenses and you're interested in making a move to RLM licensing, please contact <a href="mailto:sales@foundry.com">sales@foundry.com</a> to obtain a replacement license.</td>
</tr>
<tr>
<td>FOUNDRY_LICENSE_DEBUG</td>
<td>This variable prints additional licensing information to the command line or Terminal.</td>
</tr>
<tr>
<td>FOUNDRY_LICENSE_FILE</td>
<td>The location of the Nuke FLEXlm license file, if the following recommended location is not used:</td>
</tr>
<tr>
<td></td>
<td><strong>On Mac and Linux:</strong></td>
</tr>
<tr>
<td></td>
<td>/usr/local/foundry/FLEXlm</td>
</tr>
<tr>
<td></td>
<td><strong>On Windows:</strong></td>
</tr>
<tr>
<td></td>
<td>drive letter:\Program Files\The Foundry\FLEXlm</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> If you still use FLEXlm licenses and you're interested in making a move to RLM licensing, please contact <a href="mailto:sales@foundry.com">sales@foundry.com</a> to obtain a replacement license.</td>
</tr>
<tr>
<td>FOUNDRY_LOG_FILE</td>
<td>This variable specifies the location of Nuke Studio's logfile. If you don't specify a logfile, all output is to screen.</td>
</tr>
<tr>
<td>FOUNDRY_LOG_LEVEL</td>
<td>This variable sets the level of logging Nuke Studio produces during</td>
</tr>
</tbody>
</table>
### Nuke Environment Variables

<table>
<thead>
<tr>
<th>Environment Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>operation. There are four levels of detail, on a sliding scale from minimal to verbose:</td>
</tr>
<tr>
<td></td>
<td>• error</td>
</tr>
<tr>
<td></td>
<td>• warning</td>
</tr>
<tr>
<td></td>
<td>• message</td>
</tr>
<tr>
<td></td>
<td>• verbose</td>
</tr>
</tbody>
</table>

**Note:** Setting the logging level to verbose can produce large log files when FOUNDRY_LOG_FILE is specified.

<table>
<thead>
<tr>
<th>FRAMESERVER_LOG_DIR</th>
<th>This variable is used to specify a different location for the Frame Server to write log files to, if you'd like to keep them separate from the default NUKE_TEMP_DIR.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>See <a href="#">Using the Frame Server on External Machines</a> for more information.</td>
</tr>
</tbody>
</table>

| HIERO_DISABLE_THUMBNAI L S | Set this variable to stop Nuke Studio loading thumbnails. |

| HIERO_DISABLE_THUMBNAI L S_CACHE | Set this variable to stop Nuke Studio caching thumbnails for improved access once loaded. |

**Note:** This variable does not clear the cache, you must remove cached files manually.
<table>
<thead>
<tr>
<th>Environment Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NUKE_AJA_CHANNEL</td>
<td>AJA cards take 3G level signal (mostly for 12-bit 444 RGB) and combine it into a single 3G-B (B denotes B level, hence the 3G-B) stream through SDI1 by default. Use these environment variables to customize this output behavior:</td>
</tr>
<tr>
<td></td>
<td>• NUKE_AJA_CHANNEL - set this variable to 2, 3, or 4 to output a single stream through SDI2, SDI3, or SDI4.</td>
</tr>
<tr>
<td></td>
<td>• NUKE_AJA_DUALOUTPUT - set this environment variable to 1 to force the card to separate the single 3G stream into two 1.5G streams through SDI1 and SDI2.</td>
</tr>
<tr>
<td></td>
<td>Combining these two environment variables can force the stream to split and output through alternate SDI outputs. For example:</td>
</tr>
<tr>
<td></td>
<td>• DUALOUTPUT + CHANNEL=1 OR CHANNEL=2 results in two 1.5G streams coming from SDI1 and SDI2.</td>
</tr>
<tr>
<td></td>
<td>• DUALOUTPUT + CHANNEL=3 OR CHANNEL=4 results in two 1.5G streams coming from SDI3 and SDI4.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> Certain modes, such as 12-bit 444, require a 3G stream. Otherwise, the card uses the single stream on the channel number specified.</td>
</tr>
<tr>
<td>NUKE_AJA_DUALOUTPUT</td>
<td></td>
</tr>
<tr>
<td>NUKE_ALLOW_GIZMO_SAVING</td>
<td>Nuke does not allow you to <strong>Overwrite</strong> and <strong>Save as</strong> gizmos by default, without copying the gizmo to a Group. Setting this environment variable to 1 enables this behavior, so artists don't need to copy a gizmo before editing it.</td>
</tr>
<tr>
<td>NUKE_CRASHHANDLING</td>
<td>Breakpad crash reporting allows you to submit crash dumps to Foundry in the unlikely event of a crash. By default, crash reporting is enabled in GUI mode and disabled in terminal mode.</td>
</tr>
<tr>
<td></td>
<td>When NUKE_CRASH_HANDLING is set to 1, crash reporting is enabled in both GUI and terminal mode.</td>
</tr>
<tr>
<td></td>
<td>When NUKE_CRASH_HANDLING is set to 0, crash reporting is disabled in both GUI and terminal mode.</td>
</tr>
</tbody>
</table>
| NUKE_DEBUG_                        | When enabled, Comp Viewer image cache data is printed to the}
<table>
<thead>
<tr>
<th>Environment Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IMAGECACHE</td>
<td>command line or Terminal. Information on disk space used, the number of files cached, and the cache location is displayed.</td>
</tr>
<tr>
<td>NUKE_DEBUG_MEMORY</td>
<td>When working on large images, Nuke may need to free up memory during rendering. When this happens and NUKE_DEBUG_MEMORY is set to 1, Nuke prints the following information to the console:</td>
</tr>
<tr>
<td></td>
<td>Memory: over maximum usage, trying to reduce usage from 1 GB to 924 MB.</td>
</tr>
<tr>
<td></td>
<td>If this variable is not set, you cannot see the debug memory messages.</td>
</tr>
<tr>
<td></td>
<td>Note that here, KB, MB, GB, and TB mean units of 1000. For example, 1MB means 1,000,000 bytes.</td>
</tr>
<tr>
<td>NUKE_DISK_CACHE</td>
<td>The location where Nuke saves all recent images displayed in the Viewer. Ideally, this should be a local disk with the fastest access time available.</td>
</tr>
<tr>
<td>NUKE_DISK_CACHE_GB</td>
<td>The maximum size the disk cache can reach (in gigabytes).</td>
</tr>
<tr>
<td>NUKE_EXR_TEMP_DIR</td>
<td>On Linux, this is the location Nuke uses for temporary files while reading PIZ-compressed .exr files. This environment variable is only relevant on Linux.</td>
</tr>
<tr>
<td></td>
<td>If this variable is not set, the location is determined by NUKE_TEMP_DIR.</td>
</tr>
<tr>
<td>NUKE_EXR_TEMP_NAME</td>
<td>Changes the naming convention of .exr temporary files during rendering.</td>
</tr>
<tr>
<td></td>
<td>Setting the variable to 1 writes temporary .exr files as &lt;filename&gt;.exr.tmp, rather than &lt;filehash&gt;.tmp as in previous releases.</td>
</tr>
<tr>
<td>NUKE_FONT_PATH</td>
<td>The location that Nuke checks for available font files when the Text node properties panel is opened.</td>
</tr>
<tr>
<td>NUKE_IGNORE_ROTO_INCOMPATIBILITY</td>
<td>This variable disables the warning dialog displayed when you open scripts containing pre-Nuke 8 RotoPaint nodes.</td>
</tr>
</tbody>
</table>
| NUKE_INTERACTIVE             | The import nuke function checks-out a nuke_r render license by
<table>
<thead>
<tr>
<th>Environment Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NUKE_ENVIRONMENT_VARIABLE</td>
<td>Default. If you want to use Nuke interactively, and you have an interactive license, set this environment variable to 1.</td>
</tr>
<tr>
<td>See Nuke as a Python Module for more information.</td>
<td></td>
</tr>
<tr>
<td>NUKE_LEGACY_CHANNEL_SORTING</td>
<td>This variable disables the new channel sorting behavior, where the RGBA layer is sorted first. Enabling this variable causes Nuke to sort channels alphabetically.</td>
</tr>
<tr>
<td>NUKE_LOCALIZATION_NUMWATCHERS</td>
<td>Controls the number of threads available for localization tasks. Increasing the number of threads can improve localization performance.</td>
</tr>
<tr>
<td>NUKE_MOV64READER_ENABLE</td>
<td>Set this variable to 0 to disable Nuke's 64-bit mov decoding and fall back to 32-bit decoding.</td>
</tr>
<tr>
<td>NUKE_NO_CRASH_PROMPT</td>
<td>When crash handling is enabled in GUI mode, this allows you to control whether reports are automatically submitted or not:</td>
</tr>
<tr>
<td>When NUKE_NO_CRASH_PROMPT is set to 1, crash reports are submitted automatically without displaying a crash reporter dialog.</td>
<td></td>
</tr>
<tr>
<td>When NUKE_NO_CRASH_PROMPT is set to 0, Nuke always displays a crash reporter dialog before submitting a crash report.</td>
<td></td>
</tr>
<tr>
<td>NUKE_NO_VIEWER_GPU</td>
<td>Disables Nuke's Comp Viewer OpenGL hardware acceleration.</td>
</tr>
<tr>
<td>NUKE_PATH</td>
<td>The location where files related to Nuke customizations are stored. For more information, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.</td>
</tr>
<tr>
<td>NUKE_TEMP_DIR</td>
<td>The location where Nuke saves any temporary files that do not have a particular place defined for them.</td>
</tr>
<tr>
<td>You can find the current location of Nuke's temporary directory from within Nuke by pressing X on your keyboard, when the focus is on the Node Graph, and then running the following TCL command: getenv NUKE_TEMP_DIR</td>
<td></td>
</tr>
<tr>
<td>NUKE_WINDOWMANAGER_</td>
<td>When enabled, data from Nuke's window manager is printed to the command line or Terminal.</td>
</tr>
</tbody>
</table>
## Nuke Environment Variables

<table>
<thead>
<tr>
<th>Environment Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEBUG</td>
<td>Set this variable to the location of your OCIO configuration file for color conversion.</td>
</tr>
<tr>
<td>OCIO</td>
<td>Set this variable to the location of your OCIO configuration file for color conversion. <strong>Note:</strong> If you plan to use the OCIO config file specified in the Preferences, ensure that the Preferences &gt; Project Defaults &gt; Color Management &gt; Export &gt; use OCIO nodes when exporting to a Comp checkbox is enabled.</td>
</tr>
<tr>
<td>OFX_PLUGIN_PATH</td>
<td>The location where Nuke looks for OFX plug-ins. For more information, see Loading OFX Plug-ins.</td>
</tr>
<tr>
<td>QT_COMPRESS_TABLET_ EVENTS</td>
<td>Due to recent updates to Qt, running Nuke on CentOS 7 Linux distributions with a tablet can cause lag when moving Roto shapes around the Viewer. Setting this environment variable compresses tablet events, eliminating the lag.</td>
</tr>
<tr>
<td>QT_PLUGIN_PATH</td>
<td>The location where Nuke looks for custom Qt libraries if you don't want to use those shipped with Nuke. Setting this environment variable adds the custom path to Nuke's Qt library paths.</td>
</tr>
<tr>
<td>TIMELINE_DISABLE_PBO_UPLOADES</td>
<td>When enabled, the performance benefit from using pixel buffer objects (PBO) for texture uploads from RAM to the GPU is disabled. You can try disabling PBOs if you notice playback degradation.</td>
</tr>
</tbody>
</table>
Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts

On start-up, Nuke scans various directories for files that customize the behavior of Nuke. It looks for information on favorite directories, menu options, image formats, gizmos, NDK plug-ins, Python scripts, generic Tcl scripts, and preferences.

⚠️ **Warning:** It's worth saying that you should edit Python files with care as mistakes could stop Nuke from running.

For more information on Python in Nuke, see [The Script Editor and Python](Help > Documentation) or the Python Developer's Guide.

To make your customizations available to all versions of a particular release, place them in the following directories:

- **Linux:** `/usr/local/Nuke/<version>/plugins/
- **Mac:** `/Library/Application Support/Nuke/<version>/plugins/
- **Windows:** `C:\Program Files\Common Files\Nuke\<version>\plugins\`

If you want Nuke to look for plug-ins somewhere else rather than in these default locations, you can also define a common plug-in path yourself. Thus, by defining the Nuke plug-in path, you can assign yourself a common shared directory from which to control Nuke for multiple artists. See [Defining the Nuke Plug-in Path](Defining the Nuke Plug-in Path) for more information.

Nuke also looks in specific sub-directories of your home directory and the Nuke application directory in the order shown by platform:

- **Linux:**
  
  `/usr/local/Nuke11.3v6/plugins`
  `/usr/local/Nuke11.3v6/plugins/nukescripts`
  `/home/login name/.nuke`

- **Mac:**

---

### Note: If you place customizations in the application directories, they'll only be available for that release.
/Applications/Nuke11.3v6/Nuke11.3v6.app/Contents/MacOS/plugins

/Applications/Nuke11.3v6/plugins/nukescripts

/Users/login name/.nuke

- Windows:

  drive letter:\Program Files\Nuke11.3v6\plugins\nukescripts or
  drive letter:\Program Files\Nuke11.3v6\plugins

  ~\n.nuke

**Note:** On Windows, the `.nuke` folder can be found under the directory pointed to by the HOME environment variable. If this variable is not set (which is common), the .nuke directory will be under the folder specified by the USERPROFILE environment variable - which is generally of the form `drive letter:\Documents and Settings\login name\` or `drive letter:\Users\login name\`

To find out if the HOME and USERPROFILE environment variables are set and where they are pointing at, enter `%HOME%` or `%USERPROFILE%` into the address bar in Windows Explorer. If the environment variable is set, the folder it’s pointing at is opened. If it’s not set, you get an error.

In addition to setting up your Python/plugin environment using the `init.py` and `menu.py` files, Nuke Studio also allows you to run Python code automatically on start-up. The default location for this Python code is:

`<STARTUP_PYTHON_PATH>/Python/Startup`

Where by default, `<STARTUP_PYTHON_PATH>` is `$HOME/.nuke` as described in the OS breakdown above.

Any Python `.py` modules or packages containing `__init__.py` found within the `/Python/Startup` directory is imported when the application launches, and added to the `hiero.plugins` namespace.
Defining the Nuke Plug-in Path

To define the plug-in path:
1. On each artist's machine, create an environment variable called **NUKE_PATH**.
2. Assign the **NUKE_PATH** environment variable to the path name of the directory where files related to Nuke customization will reside.
   
   For example, on Mac using a csh or tcsh shell:
   ```
   setenv NUKE_PATH /SharedDisk/Nuke
   ```
   Or, if you're using a bash or ksh shell:
   ```
   export NUKE_PATH=/SharedDisk/Nuke
   ```

   Loading a plug-in (plugin_find()) searches the NUKE_PATH until the first plug-in is located, ensuring that only the most local plug-in is loaded. For example if the NUKE_PATH variable contains:

   ```
   project_dir:studio_dir:company_dir
   ```

   The path is searched in the following order until the first plug-in is located:
   ```
   • ~/.nuke
   • project_dir
   • studio_dir
   • company_dir
   • nuke_dir
   ```

   However, the NUKE_PATH environment variable is parsed in reverse order when loading **init.py** and **menu.py** and all discovered copies are used. This allows localized settings to override more global ones. So, with the directory hierarchy above, **init.py** scripts would execute as follows:

   ```
   • nuke_dir/init.py
   • company_dir/init.py
   • studio_dir/init.py
   • project_dir/init.py
   • ~/.nuke/init.py
   ```
Loading OFX Plug-ins

On start-up, Nuke scans various directories for OFX plug-ins that bring additional functionality to Nuke. Paths to these directories vary between operating systems, but here are examples of where you may find them:

- **Linux:**
  
  /usr/OFX/Nuke
  
  /usr/OFX/Plugins

- **Windows:**
  
  C:\Program Files\Common Files\OFX\Nuke (or, when using 64-bit Nuke on 64-bit Windows, \Program Files (x86)\Common Files\OFX\Nuke)
  
  C:\Program Files\Common Files\OFX\Plugins (or, when using 64-bit Nuke on 64-bit Windows, \Program Files (x86)\Common Files\OFX\Plugins)

- **Mac:**
  
  /Library/OFX/Nuke
  
  /Library/OFX/Plugins

If you want Nuke to look for OFX plug-ins somewhere else you can. Just define the environment variable `OFX_PLUGIN_PATH` to point to the new directory.

For example, on Mac using a csh or tcsh shell:

```
setenv OFX_PLUGIN_PATH /SharedDisk/OFX
```

Or, if you're using a bash or ksh shell:

```
export OFX_PLUGIN_PATH=/SharedDisk/OFX
```
Defining Common Favorite Directories

Favorite directories can be accessed by artists with a single click from any Nuke file browser. Typically you would create these favorites for common directories on a project.

To Define a Common Set of Favorite Directories

To define an common set of favorite directories:

1. Create a file called `menu.py` in your plug-in path directory.
   
   For more information on plug-in path directories, see `Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts`.

2. Add an entry in the following format:

   ```python
   nuke.addFavoriteDir('DisplayName', 'Pathname')
   ```

   • Replace `DisplayName` with the string you want as the display name for the directory link, for example ‘Home’ or ‘Desktop’.

   • Replace `Pathname` with the path name to the directory to which the favorite should point.

3. In the above entry, you can also add the following optional arguments after `Pathname`:

   • `type`. This is an integer and a bit-wise operator, OR a combination of `nuke.IMAGE`, `nuke.SCRIPT` or `nuke.FONT`.

   • `nuke.IMAGE` restricts the favorite directory to appear only in the image file browser, which Nuke opens for file Read/Write operations.

   • `nuke.SCRIPT` restricts the favorite directory to appear in the script file browser, which appears when you select `File > Open Comp` or a similar menu selection to open or import script files.

   • `nuke.FONT` restricts the favorite directory to appear in the font browser.

   • `icon='Name'`. Replace `Name` with the name and file extension of the `.png` (or `.xpm`) image you wish to use as the icon for the favorite directory. This image must be stored in your Nuke plug-in path directory. It should be 24 x 24 pixels in size.

   • `tooltip='My tooltip'`. Replace `My tooltip` with the string you wish to display as pop-up help.

**Example 1**

The following entry would create a favorite called `DisasterFlickStore` which appears on all File Browsers invoked from Read nodes and which points to the `/job/DisasterFlick/img` directory.
nuke.addFavoriteDir ('DisasterFlickStore', '/job/DisasterFlick/img', nuke.IMAGE)

The result of example 1.

Example 2

The following entry would create a favorite called Test. It appears on all File Browsers invoked from Read nodes or by selecting File > Open Comp and points to the /job/Test directory. The entry also defines Test Images and Scripts as the tool tip for the favorite directory.

nuke.addFavoriteDir ('Test', '/job/Test', nuke.IMAGE | nuke.SCRIPT, tooltip='Test images and Scripts')
The result of example 2.
Handling File Paths Cross Platform

If your facility uses Nuke on several operating systems, you may want to configure Nuke to replace the beginnings of file paths so that scripts created on one platform also work on another.

For example, to ensure file paths created on Windows also work on Linux and vice versa, you can do the following:

1. Create a file called init.py in your plug-in path directory if one doesn't already exist.
   For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.
2. Open the init.py file in a text editor and add an entry in the following format:

   ```python
   import platform
   
   def filenameFix(filename):
       if platform.system() in ("Windows", "Microsoft"):
           return filename.replace( "/SharedDisk/", "p:\" )
       return filename.replace( "p:\", "/SharedDisk/" )
   ```

   This way, the Windows file paths (beginning with p:\ in the above example) are replaced with the Linux file paths (beginning with /SharedDisk/) under the hood whenever a Nuke script is used on Linux. Otherwise, the Windows file paths are used.

   Note that the file paths displayed in the graphical user interface (GUI) do not change. If you are using p:\ in a node control, this is still displayed as p:\. However, on Linux, Nuke interprets p:\ as /SharedDisk/.

   ```
Setting Default Values for Controls

You can set default values for node controls by adding a simple line of Python to your `init.py` file. Once a default value is set, all controls with matching names default to this value. For example, you can set default values for file format specific controls in the Read, Write or other file format-dependent nodes. To set a default value, use the following statement:

```python
nuke.knobDefault()```

To specify file format specific defaults, use the class name, followed by the file format extension and the control name, all separated by periods. For example:

```python
nuke.knobDefault("Read.exr.compression", "2")```

Maybe you want the last frame value of the frame range controls in the Project Settings to default to frame 200. To do this, use the following statement:

```python
nuke.knobDefault("Root.last_frame", "200")```

Or you might want to set a different default font style in the Text node:

```python
nuke.knobDefault("Text2.font", "{ Arial : Regular : arial.ttf : 0 }")"
Defining Custom Menus and Toolbars

You can freely add custom menus and menu options as well as toolbars and toolbar options to the Nuke interface. Artists can then use these options to trigger gizmos and plug-ins stored in the plug-in path directory.

For example, to add a new menu in the default Toolbar with an option to trigger a gizmo called MyGizmo, you can do the following:

1. In your home directory, create a directory called `.nuke` (if it doesn't exist already). For more information on this directory, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.
2. In the `.nuke` directory, create a file called `menu.py` if one does not already exist.
3. In a text editor, modify the file `menu.py`, adding the lines:
   ```python
   toolbar = nuke.toolbar("Nodes")
   toolbar.addCommand( "Test/MyGizmo", "nuke.createNode('MyGizmo')")
   ```
   This adds a menu labeled "Test" to the default Nodes Toolbar with an item labeled "MyGizmo" that creates an instance of the node MyGizmo.

   **Note:** Node Class() names occasionally change between major releases, such as Nuke 7 to Nuke 8. While these changes do not affect legacy scripts, you may not get the results you were expecting if a node class has been modified. The `toolbars.py` file, used to create Nuke's node toolbar, contains all the current node class names and is located in `<install_directory>/plugins/nukescripts/` for reference.

   As an example, between Nuke 7 and Nuke 8, the Text node Class() changed from Text to Text2. In the `toolbars.py` file for the two releases, the entries for the Text node appear as follows:

   ```python
   m.addCommand("Text", "nuke.createNode("Text")", icon="Text.png")
   m.addCommand("Text", "nuke.createNode("Text2")", icon="Text.png")
   ```

   It's also possible to add items to other menus in Nuke and even create your own toolbars. The following sections cover these possibilities in detail.
To Add a Toolbar

To add a toolbar:

1. Create a file called `menu.py` in your plug-in path directory if one doesn't already exist. For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.

2. Open the `menu.py` file in a text editor and add an entry in the following format:
   ```python
t=nuke.toolbar("ToolbarName")
t.addCommand("NewMenu", "PythonCode", "Shortcut", icon="IconName")
```
   - Replace `ToolbarName` with the name you want to give to the toolbar. This name appears in the content menus under Windows > Custom and above the toolbar on the title tab.
   - Replace `NewMenu` with the name of the menu you want to add to the toolbar.
   - Replace `PythonCode` with relevant Python code (usually `nuke.createNode`), and, if necessary, use the name of the gizmo, generic Python script, or plug-in file you want the menu option to invoke. For ease of use, place all such referenced files inside the plug-in path directory.
     
     If you like, you can also replace `PythonCode` by a Python callable.
   - Replace `Shortcut` with a keyboard shortcut, for example `Alt+A`, `Ctrl/Cmd+A`, or `Shift+A`. The letter `a` alone represents lower-case `a`. `F1` represents function key 1. You can combine the `Shift`, `Ctrl/Cmd`, and `Alt` keys as necessary. If you like, you can also use `#A` to represent `Alt+A`, ^A to represent `Ctrl/Cmd+A`, and +A to represent `Shift+A`.
   - Replace `IconName` with the name of the `.png` (or `.xpm`) image you wish to use as the menu icon. This image must be stored in your Nuke plug-in path directory. It should be 24 x 24 pixels in size.

3. In the above entry, you can also add the following optional arguments in the parenthesis after "ToolbarName":
   - `True`. This is the default. When True, `nuke.toolbar()` calls the toolbar with the given name or creates it if it does not exist. For example, `t=nuke.toolbar("Extras", True)` would either call an existing toolbar called Extras or create one if it did not already exist.
   - `False`. When False, the toolbar is not created if it does not already exist and `nuke.toolbar()` returns None. You can use this to find out if a toolbar with a given name already exists. For example, `t=nuke.toolbar("Extras", False)` would either call an existing toolbar called Extras or return None if such a toolbar did not exist.

The new toolbar does not appear by default, but is listed under Custom in the content menus. From there, you can insert it in any pane. Once you are happy with the new toolbar and its position, save the layout (select Workspace > Save Workspace). Thereafter, the toolbar appears whenever Nuke is launched in the saved workspace.
You can build several toolbars for different tasks and save layouts that have one or another present for easy context switching.

**Example 1**

The following entry creates a new toolbar called *Extras*. The toolbar includes an option called *Create VectorBlur* that creates a VectorBlur node. The entry also defines `v` as the keyboard shortcut for the VectorBlur node.

```python
t=nuke.toolbar("Extras")
t.addCommand("Create VectorBlur", "nuke.createNode ('VectorBlur')", "v")
```

**Example 2**

In this example, we add an option called Autoplace to the toolbar created in example 1. This option places the selected nodes neatly one after another, as illustrated in the following images:

![Before using Autoplace](image1.png) ![After using Autoplace](image2.png)

The following entry adds the Autoplace option. It also defines `Alt+A` as the keyboard shortcut for this option.

```python
def _autoplace():
n = nuke.selectedNodes()
for i in n:
nuke.autoplace(i)
t=nuke.toolbar("Extras")
t.addCommand("Autoplace", "_autoplace()", "Alt+a")
```

**To Define a Menu or Toolbar Option**

To define a menu or toolbar option:
1. If you haven’t already done so, create a file called `menu.py` in your plug-in path directory. For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.

2. Open the `menu.py` file in a text editor and add an entry in the following format:

   ```python
   menubar = nuke.menu("MenuType")
   m = menubar.addMenu("&NewMenu")
   m.addCommand("&NewItem", "PythonCode", "Shortcut", icon="IconName", index=#)
   ```

   - Replace `MenuType` with the type of menu or toolbar you want to add an item to:
     - **Nuke** adds an item to the application main menu bar.
     - ![Image](image1.png)

     **Animation** adds an item to the menu on the Animation button of all panels, and to the right-click menu of the Curve editor.

     **Properties** adds an item to the right-click menus of properties panels.

     **Node Graph** adds an item to the right-click menu of the Node Graph.

     **Nodes** adds an item to the default Toolbar.

     **Viewer** adds an item to the right-click menu of the Viewer.

     **Pane** adds an item to the content menus where it appears under **Custom**.

   - Replace `NewMenu` with the menu name. Using an existing menu name appends any new options to the existing menu. You can also add options to the default Menu bar and Toolbar.

   - Replace `NewItem` with the underlying item you want to add to the menu. You may precede any character with an & in order to flag it as keyboard shortcut trigger.

   - Replace `PythonCode` with relevant Python code (usually `nuke.createNode`) and, if necessary, use the name of the gizmo, generic Python script, or plug-in file you want the menu option to invoke. For ease of use, place all such referenced files inside the plug-in path directory.

   For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.

   If you like, you can also replace `PythonCode` by a Python callable. This has the advantage that you get informed about errors in your script on start-up instead of when the menu item is invoked. For an example of using a lambda function, see Example 3 on page 1168.

   - Replace `Shortcut` with a keyboard shortcut, for example `Alt+A`, `Ctrl/Cmd+A`, or `Shift+A`. The letter a alone represents lower-case `a`. `F1` represents function key 1. You can combine the `Shift`, `Ctrl/Cmd`, and `Alt` keys as necessary. If you like, you can also use `#A` to represent `Alt+A`, `^A` to represent `Ctrl/Cmd+A`, and `+A` to represent `Shift+A`.  

   ![Image](image2.png)
Note: By assigning a keyboard shortcut, you can overwrite existing shortcuts. For example, if you assign the shortcut Ctrl/Cmd+O to a new menu item, it is no longer used for its default purpose, which is opening a file. However, shortcuts are only overwritten in the main menu bar, the Toolbar, any user-created toolbars, and the menu you are adding the new menu item to. This means you can add a shortcut into the Node Graph, for example, without resetting the same shortcut in the Viewer. However, you cannot add a shortcut into the Node Graph without resetting the same shortcut in the main menu bar or the Toolbar.

- Replace **IconName** with the name of the .png (or .xpm) image you wish to use as the menu icon. This image must be stored in your Nuke plug-in path directory. It should be 24 x 24 pixels in size.
- Replace **#** with a number that represents the position of the item in the menu or toolbar. If you do not use an index keyword, the item is added in the end of the menu or toolbar.

**Tip:** You can also put the menu name in the addCommand call, like this:

```python
nuke.menu("MenuType").addCommand("NewMenu/NewItem", "PythonCode("name")")
```

### Example 1

The following entry creates a new menu and option called **Custom > Cue Render** in the menu bar. It inserts a gizmo called “cue_render.” The entry also defines **Ctrl+R** as the keyboard shortcut for the gizmo.

```python
menubar=nuke.menu("Nuke")
m=menubar.addMenu("&Custom")
m.addCommand("&Cue Render", "nuke.createNode('cue_render')", "Ctrl+R")
```

![Custom Menu Example](image)

### Example 2

For information on how to create a new menu in the default Toolbar with a menu item that triggers a gizmo, see the example under **Defining Custom Menus and Toolbars**.
Example 3

The following entry creates a menu and options called Custom > Filters > Blur in the menu bar. Selecting Blur inserts the Blur node.

```python
menubar=nuke.menu("Nuke")
m=menubar.addMenu("&Custom")
m.addCommand("Filters/Blur", "nuke.createNode("Blur")")
```

![Custom Menu Example](image)

You can also do the same with a lambda function:

```python
menubar=nuke.menu("Nuke")
m=menubar.addMenu("&Custom")
m.addCommand("Filters/Blur", lambda: nuke.createNode("Blur"))
```

This way, you don't have to use the backslashes.
Defining Common Image Formats

You may wish to define image formats (image resolutions and corresponding pixel aspect ratios) for a given project. These appear as dropdown menus on Read and Reformat nodes.

Defining an Image Format

To define an image format:

1. If you haven't already done so, create a file called `menu.py` in your plug-in path directory.
   For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.
2. Add an entry in the following format:
   ```python
   nuke.addFormat(" ImageWidth ImageHeight LowerLeftCorner LowerRightCorner UpperRightCorner UpperLeftCorner PixelAspectRatio DisplayName ")
   • Replace `ImageWidth` with the width (in pixels) of the image format.
   • Replace `ImageHeight` with the height (in pixels) of the image format.
   • If you wish to define an image area that is smaller than the format resolution, replace `LowerLeftCorner`, `LowerRightCorner`, `UpperRightCorner`, `UpperLeftCorner` with the proper x and y coordinates (in pixels) for the lower left, lower right, upper right, and upper left corners of this smaller image area (optional).
   • Replace `DisplayName` with display name of the format. This appears on all relevant dropdown menus.

Example

The following entry would create a new format called `full_aperture_anamorphic` with a resolution of 2048 by 1556 and a pixel aspect ratio of 2.0. (As the image corners are not explicitly defined, the image covers the entirety of the format area.)
```python
nuke.addFormat(" 2048 1556 2.0 full_aperture_anamorphic ")
```
Creating and Accessing Gizmos

Nuke enables artists and technical directors to create gizmos, which are groups of Nuke nodes that may be reused by other artists. Studios commonly use gizmos to consistently apply certain color grading techniques, process incoming footage according to a particular conversion formula, and process outgoing footage in preparation for film printing.

A gizmo is a Group Node that you create and save in a separate .gizmo file in your Nuke plug-in folder. Nuke scripts can use this gizmo just like any other node type. Saved scripts only contain the name and control settings for the gizmo; the definition is in the gizmo file and it is read at the same time the script is loaded into Nuke. Thus, you can alter the implementation of the gizmo and change all the scripts that are using it.

**Note:** Unlike other nodes, gizmos cannot be cloned. For more information, see Working with Nodes.

**Tip:** You can register gizmos as soft effects in Nuke's Timeline environment. See Soft Effects for more information.

Creating Gizmos

A gizmo is a Group Node that you create and save in a separate .gizmo file in your Nuke plug-in folder. Nuke scripts can use this gizmo just like any other node type.

Using Nuke's Export gizmo command, you can export nodes inside a Group and explicitly control which controls may be edited by the artists to ensure the processes within the gizmo are consistently applied.

To create a gizmo:

1. Select the nodes you want to include in the gizmo.
2. Select **Other > Group** from the Toolbar (or press Ctrl/Cmd+G) to group the nodes.
3. You may want to rename the group by entering a new name in the Group properties panel title field. This step is optional, and has no effect on the saved gizmo. However, it is often a good idea to give the Group the same name you intend to use for the gizmo.
4. To expose which controls the artists can adjust, see Managing Gizmo Controls.
5. Click the **export as gizmo** button.
6. In the file browser that appears, click **Home**. Type `.nuke/` after the path displayed at the bottom of the file browser.
7. Enter a name after the path, and append a `.gizmo` extension after the name. This is the name of the command that is written to any saved script that's using the gizmo. It's a good idea, and common practice, to begin the name with a capital letter, because Nuke uses this as an indication that the command is a node or a gizmo.
8. Click **Save**.

    See Accessing Gizmos in Nuke for information on how to access your gizmo in Nuke.

**Note:** Nuke does not allow you to Overwrite and Save as gizmos by default, without copying the gizmo to a Group. If you want to allow this behavior, so artists don't need to copy a gizmo before editing it, set the NUKE_ALLOW_GIZMO_SAVING environment variable to 1. See Environment Variables for more information.
Managing Gizmo Controls

You can add controls to your gizmo by picking and editing from the default controls that exist for the nodes inside your Group node or by adding a control you have created yourself to your gizmo Properties panel.

In Nuke's Properties panel, you can add controls using the Adding Knobs Using Drag-and-Drop edit button or by right-clicking in the panel and selecting Adding Knobs Manually. Drag-and-drop knobs significantly reduce the time spent managing user knobs in a node's Properties panel when compared to the legacy Manage User Knobs method within Nuke.

Adding Knobs Using Drag-and-Drop

You can drag-and-drop knobs between open node Properties panels or add your own using the knob icons listed at the top of the panel. You can also order, hide, customize, and delete knobs within the panel.

Note: The drag-and-drop interface is also useful for managing LiveGroup knobs. See Using the LiveGroup Node for more information.

Tip: You can float the User Knob Editor by clicking the content menu at the top of the pane and selecting Windows > User Knob Editor.

To add knobs to a Group:
1. Click the edit button at the top of the Group Properties panel.
The **User Knob Editor** is displayed and the node's properties are now editable.

2. To add a knob, either:

   Open the **Properties** panel of the node containing the knob(s) you want to expose and drag-and-drop the knob(s) onto the Group node's properties,
OR

Drag-and-drop a new knob from the User Knob Editor onto the Group node's properties.
Knobs added from nodes in the Group are marked with a gray dot and new knobs are marked with an orange dot.

See [Organizing Knobs](#) for information on how to rearrange knobs in the panel.

3. Click the edit button ![edit](#) at the top of the Group **Properties** panel to exit edit mode.

See [Accessing Gizmos in Nuke](#) for information on how to access your gizmo in Nuke.

### Editing Knobs

Knobs added to the **Properties** panel are edited by clicking the gray or orange dot next to the knob label.
The components you can edit depend on the knob type. For example, a *translate* knob from a node within the Group allows you to edit the following:

- **Name** - the scripting name used to call the knob from scripts and expressions. This field is alphanumeric and cannot contain spaces.
- **Label** - the display name for the knob in the Nuke interface.
- **Tooltip** - the information displayed when you hover the pointer over the knob.

**Tip:** The Nuke Reference Guide lists the labels and knob names for the majority of Nuke nodes.

A more complex example is the *Python Script Button* knob from the User Knob Editor. Adding a Python knob to the Properties panel allows you to edit an additional component, *Script*. The *Script* field adds Python to the knob, which is executed when you click the button. For example:
Click the edit button at the top of the Group **Properties** panel to exit edit mode. See [Accessing Gizmos in Nuke](#) for information on how to access your gizmo in Nuke.

**Organizing Knobs**

You can re-order, hide, and remove knobs quickly using the drag-and-drop interface. When you're dragging knobs around the **Properties** panel, an orange guide line indicates where the knob is placed when dropped.

You can drag-and-drop multiple knob selections in the same way. Hold **Ctrl/Cmd** and click to select multiple knobs in the **Properties** panel.
You can add dividing lines between controls to delineate between groups of knobs, such as transform and color knobs. To add a divider, drag-and-drop a Divider Line knob between the controls you want to separate.

Click the edit button at the top of the Group Properties panel to exit edit mode. See Accessing Gizmos in Nuke for information on how to access your gizmo in Nuke.

Adding Tabs

You can add tabs in the Properties panel to organize knobs into groups, just like Nuke's existing nodes. To add a tab:

1. Drag a Tab knob from the User Knob Editor and drop it into the Properties.

Dropping a Tab knob above other knobs moves those knobs to the new tab.
2. Drag-and-drop knobs between tabs to organize them as required.

Tip: If you want to create an empty tab, drop the Tab knob at the bottom of the knob list.

Click the edit button at the top of the Group Properties panel to exit edit mode. See Accessing Gizmos in Nuke for information on how to access your gizmo in Nuke.

Hiding and Removing Knobs

You can hide knobs by clicking the icon next to the knob. Hiding a knob doesn't hide it in edit mode, only when you've finished exposing knobs and clicked the edit button.

To remove a knob you can:
• Select the knob and press backspace or delete,
• Click the button next to the knob, or
• Drag the knob to the bottom-left of the Properties panel and drop it into the trash icon.
Click the edit button at the top of the Group Properties panel to exit edit mode. See Accessing Gizmos in Nuke for information on how to access your gizmo in Nuke.
Adding Knobs Manually

You can manually add controls to your Group node before exporting it as a gizmo in two different ways:
• by picking and editing a control from the default controls that exist for the nodes inside your Group node. For example, if the Group node contains a Grade node, you can add any of the Grade node controls to your gizmo Properties panel.
• by adding a control you have created yourself to your gizmo Properties panel.

To Pick Existing Controls

1. Right-click on the dark gray background of the Group properties panel and select Manage User Knobs. The following dialog opens.

2. To pick a control you want to give the users control of, click on the Pick button. This opens a dialog that lists all the nodes that the group contains.
Expand the list items as necessary to see the controls you can include in the gizmo controls. Select a control and click OK. You can also select multiple controls by Ctrl/Cmd+clicking on them, or pick a range of controls by Shift+clicking.

At this point, a new tab called User appears in the Group properties panel. The control you selected has been added to this tab. In our example, we selected the size control of the Text node.

The control has also been added to the Manage User Knobs dialog, ready for you to edit.

3. To edit the control you added, open the Manage User Knobs dialog and select the control from the list. Click Edit.

In most cases, you can edit the following:

- **Name** - Give the new control a unique name here. You need to use this name whenever you want to reference the control from scripts or via expressions. The name can only contain letters and digits. Spaces or punctuation are not allowed. This field cannot be left empty.

- **Label** - Whatever you enter here appears to the left of the control in the gizmo properties panel (or, in the case of buttons, on the button). If you leave this empty, whatever is in the Name field is also used as the label.

  In the Label fields of check boxes, Tcl script buttons, and Python script buttons, you can also use HTML. For example, to have your text appear in bold, you can enter `<b>text</b>`.
To add an icon to your check box or Tcl/Python button using HTML, you can enter `<img src="Colorwheel.png"/>` in the Label field. This adds the Nuke color wheel icon. You can also use your own icons in the same way as long as you save them in your plug-in path directory. Most common image formats work, but we recommend using .png files.

Note that the HTML has been changed to a slightly non-standard form where newlines are significant. If there is a newline character in your data, a new line is displayed in the label.

- **Hide** - Check this to hide the control from the users. This can be useful if you want to make a new control to contain a complex expression that you can then refer to repeatedly by other controls.
- **Start new line** - Uncheck this if you want the control to appear on the same line as the previous control in the gizmo properties panel.
- **Tooltip** - Enter a short help text here. It appears, along with the Name, in a pop-up tool tip when the user points the mouse at the control. If you do not provide a tool tip, whatever is in the Name field is used as the tool tip.

4. If necessary, repeat the previous three steps to add more controls to your Group node (future gizmo).

5. In the Group node properties panel, the controls are listed in the same order as they are in the Manage User Knobs dialog. To move a control up or down in the properties panel, select it in the dialog and use the Up and Down buttons.

6. Click the export as gizmo button.

7. In the file browser that appears, click Home. Type `.nuke/` after the path displayed at the bottom of the file browser.

8. Enter a name after the path, and append a `.gizmo` extension after the name. This is the name of the command that is written to any saved script that’s using the gizmo. It’s a good idea, and common practice, to begin the name with a capital letter, because Nuke uses this as an indication that the command is a node or a gizmo.

9. Click Save.

To Create New Controls:

1. Right-click on the dark gray background of the Group properties panel and select Manage User Knobs. The following dialog opens.
2. To add a new control, tab, static text, or divider line to the Group (gizmo) controls, click **Add** on the **Manage User Knobs** dialog and select the option you want to add. This opens a dialog where you can edit the control, tab or static text you added. In most cases, you can edit the following:

• **Name** - Give the new control a unique name here. You need to use this name whenever you want to reference the control from scripts or via expressions. The name can only contain letters and digits. Spaces or punctuation are not allowed. This field cannot be left empty.

• **Label** - Whatever you enter here appears to the left of the control in the gizmo properties panel. If you leave this empty, whatever is in the **Name** field is also used as the label.

In the **Label** fields of check boxes, Tcl script buttons, and Python script buttons, you can also use HTML. For example, to have your text appear in bold, you can enter `<b>text</b>.

To add an icon to your check box or Tcl/Python button using HTML, you can enter `<img src="Colorwheel.png"/>` in the **Label** field. This adds the Nuke color wheel icon. You can also use your own icons in the same way as long as you save them in your plug-in path directory. Most common image formats can work, but we recommend using .png files.

Note that the HTML has been changed to a slightly non-standard form where newlines are significant. If there is a newline character in your data, a new line is displayed in the label.

• **Hide** - Check this to hide the control from the users. This can be useful if you want to make a new control to contain a complex expression that you can then refer to repeatedly by other controls.

• **Start new line** - Uncheck this if you want the control to appear on the same line as the previous control in the Group properties panel.

• **Tooltip** - Enter a short help text here. It appears, along with the Name, in a pop-up tool tip when the user points the mouse at the control. If you do not provide a tool tip, whatever is in the **Name** field is used as the tool tip.

3. Use an expression to link the control you just created to a node and its control inside the Group node. This is important, because for the new control to do anything, you need to refer to it using...
an expression in some other control on a node inside the Group. For more information, see the examples below and refer to Expressions.

4. If necessary, repeat the previous four steps to add more controls to your Group node (future gizmo).

5. In the Group node properties panel, the controls are listed in the same order as they are in the **Manage User Knobs** dialog. To move a control up or down in the properties panel, select it in the dialog and use the **Up** and **Down** buttons.

6. Click the **export as gizmo** button.

7. In the file browser that appears, click **Home**. Type `.nuke/` after the path displayed at the bottom of the file browser.

8. Enter a name after the path, and append a `.gizmo` extension after the name. This is the name of the command that is written to any saved script that's using the gizmo. It's a good idea, and common practice, to begin the name with a capital letter, because Nuke uses this as an indication that the command is a node or a gizmo.

9. Click **Save**.

**To Delete Controls:**

1. Right-click on the dark gray background of the Group properties panel and select **Manage User Knobs**.

2. In the dialog that opens, select the controls that you want to delete from the list and click **Delete**.

3. To delete an entire tab, select all controls on the tab as well as the tab name and click **Delete**.

**Examples**

Below are some examples on how to create new controls for gizmos. To try them out, do the following preparations:

1. Select **Draw > Text** and **Draw > Rectangle**. Create the following setup:

2. Double-click on the Rectangle1 node.
3. In the Viewer, resize and reposition the rectangle until it looks like the following:

![Rectangle1](image1)

4. In the Rectangle1 properties panel, go to the **Color** tab. Click on the 4 button to display multiple values rather than the slider. Enter 1 as the value for r, and 0 as the value for b, g and a. This changes the color of the rectangle from white to red.

5. Copy the Rectangle1 node and paste it into the same script. Create the following connections:

![Connections](image2)

6. Double-click on the Rectangle2 node and change the color of the rectangle from red to green (r 0, g 1, b 0, a 0).

7. Select **Merge > Switch** to add a Switch node. Create the following connections:

![Connections](image3)
8. Select the Text1, Rectangle1, Rectangle2 and Switch1 nodes and press Ctrl/Cmd+G to group them. This group is the gizmo we add controls to in the following examples.

9. Delete the original four nodes from the Node Graph tab.

10. Select the Group node and append a Viewer to it.

**Example 1**

In this example, we add a control called **Version** to the Group node controls. This control is an input field. Whatever is entered in the field is called by the Text1 node and displayed in the Viewer when you view the output of the group.

1. Open the Group properties panel and right-click on the dark gray background. Select **Manage User Knobs**.

2. In the dialog that opens, select **Add > Text input Knob** to add a text input field control to your Group properties panel.

3. Enter `version` as the **Name** for the control, **Version** as the **Label**, and **Enter the version number here** as the **Tooltip**. Click **OK** and **Done** to close the dialogs.

![Text Knob](image)

This step created a tab called **User** in the Group node controls. All the controls you add or pick are added on this tab by default.

As you can see, the **Version** control is now there.

![User Node Version](image)

4. On the **Group1 Node Graph** tab, double-click the Text1 node to open its controls. In the **message** field, enter the following expression: `[value version]`. This expression calls the control named `version` that you created in the previous step. Therefore, whatever is entered in the **Version** field of the Group node (for example, `v03`), appears as a result of the Text1 node.
Example 2

This example teaches you to create a checkbox control that the users can use to specify whether they want to display or hide the version number added in the previous example.

1. In the Group properties panel, right-click on the dark gray background and select Manage User Knobs.
2. In the dialog that opens, select Add > Check Box to add a checkbox control to your Group properties panel.
3. Enter hideversion as the Name for the control, Hide version number as the Label, and Check this to hide the version number as the Tooltip.
4. To have the new control appear next to the Version control (created in the previous example) rather than below it on its own line, uncheck Start new line. Click OK and Done to close the dialogs.

The control you created appears in the Group properties panel now.

5. In the Text1 controls, go to the Node tab. Right-click on the disable control and select Add expression.

6. In the Expression field, enter hideversion (or, if you want to make it clear that the control is in the enclosing group, you can also use parent.hideversion). This calls the control you created in steps 2 and 3. Click OK.
From now on, whenever **Hide version number** is checked in the Group controls, the Text1 node is disabled and you cannot see the version number it would otherwise create.

![Image of Text1 node](image)

**Example 3**

In this example, we add a control labeled **Status** to the Group controls. This control is a dropdown menu with two options: **Finished** and **Unfinished**. When **Finished** is selected, the green rectangle is displayed. When **Unfinished** is selected, you'll see the red rectangle instead.

1. In the Group properties panel, right-click on the dark gray background and select **Manage User Knobs**.
2. In the dialog that opens, select **Add > Pulldown Choice** to add a dropdown menu control to your Group properties panel.
3. Enter **status** as the **Name** for the control and **Status** as the **Label**. In the **Menu Items** field, list the items you want to appear in the dropdown menu - in this case, **Finished** and **Unfinished**.

![Image of Pulldown Choice control](image)
Finally, enter *Select the production status here* as the **Tooltip**. Click **OK** and **Done** to close the dialogs.

The **Status** control should now appear in the Group controls.

4. On the **Group1 Node Graph** tab, double-click the Switch1 node to open its controls. Right-click on the **which** field and select **Add expression**.

5. In the dialog that opens, enter the following expression: `status==0` (or, `parent.status==0`). This expression calls the control named `status` that you created earlier in this example. For the dropdown menus, the first item is 0, the next 1, the next 2, and so on.

   From now on, whenever **Finished** is selected under **Status**, the green rectangle is shown. When **Unfinished** is selected, the red rectangle is shown.

**Example 4**

This example teaches you how to visually group and rearrange the controls you created for the Group properties panel. You can do this by renaming the **User** tab, and using static text and divider lines to group the controls on the tab.

First, we'll rename the **User** tab in the Group properties panel:

1. In the Group properties panel, right-click on the dark gray background and select **Manage User Knobs**.

2. In the dialog that opens, select **User** and click **Edit**.

3. In the **Label** field, enter a new name for the tab, for example, **Version and status**. Click **OK** and **Done** to close the dialogs.

   If you now look at the Group controls, you'll notice that the **User** tab has been renamed to **Version and status**.

Next, we'll group the two version controls of the Group node under a title called **Version controls**:

1. In the Group properties panel, right-click on the dark gray background and select **Manage User Knobs**.

2. In the dialog that opens, select **Add > Text** to add text to your Group properties panel.

3. Enter `versioncont` as the **Name** for the control and **Version controls** as the **Label**. Click **OK** and **Done** to close the dialogs.
This adds the text *Version controls* to the Group properties panel. However, the text does not appear where we want it to appear: on top of the **Version** and **Hide version number** controls. Let’s move it up.

4. Right-click on the Group properties panel again and select **Manage User Knobs**.
5. Select [**Version controls**] from the list and click **Up** three times. Click **Done**.

The text should now appear on top of the Group properties panel, above the version controls.

Finally, we’ll add a divider line between the version controls and the **Status** control:

1. In the Group properties panel, right-click on the dark gray background and select **Manage User Knobs** again.
2. In the dialog that opens, select **Add > Divider Line** to add a line to divide the controls in your Group properties panel.
3. Select the line from the Manage User Knobs dialog, where it is shown as **unnamed**.
4. Click the **Up** button once to move the line between the **Hide version number** and **Status** controls. Click **Done**.

If you now open the Group controls, you’ll notice that a there’s a line between these controls.

5. See **Accessing Gizmos in Nuke** for information on how to access your gizmo in Nuke.
Accessing Gizmos in Nuke

There are several ways to access your gizmos once you've saved them into your .nuke folder or NUKE_PATH. See Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts for more information on where Nuke looks for your gizmos.

If you're going to use a gizmo often, across multiple sessions, you can add it to a Nuke menu to provide quick access. See Defining Custom Menus and Toolbars for more information.

Otherwise, you can quickly add it per-session by:
• pressing X on the Node Graph or Properties panel and entering the gizmo name (without the extension) as a TCL command in the dialog that opens,

![TCL Command Dialog](image1)

• opening the Script Editor and entering `nuke.load('gizmo name')` where `gizmo name` is the name of the gizmo without the extension, or

![Script Editor](image2)

• selecting Other > All plugins > Update from the node toolbar and then adding the gizmo using the Tab menu in the Node Graph.
Accessing Gizmos in Nuke | Adding Knobs Using Drag-and-Drop
Sourcing Custom Plug-ins and Generic Tcl Scripts

Nuke Custom Plug-ins

The Nuke developer’s kit (NDK) allows developers to create and compile their own binary plug-ins.

To Source a Custom Plug-in

1. Place the plug-in file in the plug-in path directory. Its name should include a .dll (on Windows), .so (on Linux) or .dylib (on Mac) extension.
   For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.
2. Create a menu option referencing the plug-in file (see Defining Custom Menus and Toolbars).
   Or instruct artists to invoke the plug-in by opening the Script Editor and entering nuke.load ("plug-in name") where plug-in name stands for the name of the plug-in without the extension.

Sourcing Tcl Procedure

A Nuke script or gizmo is in fact a Tcl procedure (script). Thus, Nuke also allows you to hand code generic Tcl procedures to automate Nuke in various ways.

To Source a Generic Tcl Procedure

1. Place the Tcl procedure file in the plug-in path directory. Its name should include a .tcl extension.
   For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.
2. Create a menu option referencing the plug-in file (see Defining Custom Menus and Toolbars).
   Or instruct artists to invoke the Tcl script by opening the Script Editor and entering nuke.load ("procedure/script file name") where procedure/script file name stands for the name of the procedure of script file without the extension.

Tip: For some code samples of useful Nuke Tcl procedures, look inside the [Nuke directory]/plugins directory.
Template Scripts

You can create a template script that is loaded instead of an empty script every time you launch Nuke or select File > New Comp or File > Close Comp. This allows you to save lookup table (LUT) setups and favorite arrangements of nodes, for example.

Creating and Using a Template Script

To create and use a template script:

1. Create the script you want to use as a template.
2. Select File > Save Comp As. Navigate to ~/.nuke. The tilde (~) stands for your home directory and the full stop (.) for a hidden folder.
3. Name your script template.nk and click Save.

The next time you launch Nuke or select File > New Comp or File > Close Comp, Nuke loads the template from ~/.nuke/template.nk.

Tip: If you’re not sure of the location of your home directory, on Linux and Mac you can open a terminal window and type echo $HOME. The terminal returns the pathname to your home directory.

On Windows, you can find the .nuke directory under the directory pointed to by the HOME environment variable. If this variable is not set (which is common), the .nuke directory will be under the folder specified by the USERPROFILE environment variable. To find out if the HOME and USERPROFILE environment variables are set and where they are pointing at, enter %HOME% or %USERPROFILE% into the address bar in Windows Explorer. If the environment variable is set, the folder it’s pointing at is opened. If it’s not set, you get an error.

Here are examples of what the path name may be on different platforms:

Linux: /home/login name
Mac: /Users/login name
Windows: drive letter:Documents and Settings\login name or drive letter:Users\login name
Defining Common Preferences

The Nuke Preferences dialog (Edit > Preferences) allows any user to make myriad behavior and display adjustments to the interface. However, you may wish to assign certain default preferences for artists.

Defining Default Preferences

To define default preferences:

1. Select Edit > Preferences to display the Preferences dialog.
2. Modify the controls within the dialog as necessary. For descriptions of what the controls do, see Appendix A: Preferences.
3. Click OK. Nuke writes the modified preferences to a file called preferences11.3.nk, which is stored inside your [home directory]/.nuke directory.

Tip: If you’re not sure of the location of your home directory, on Linux and Mac you can go to a terminal window and type `echo $HOME`. The terminal will return the path name to your home directory.

On Windows, you can find the .nuke directory under the directory pointed to by the HOME environment variable. If this variable is not set (which is common), the .nuke directory will be under the folder specified by the USERPROFILE environment variable. To find out if the HOME and USERPROFILE environment variables are set and where they are pointing at, enter `%HOME%` or `%USERPROFILE%` into the address bar in Windows Explorer. If the environment variable is set, the folder it’s pointing at is opened. If it’s not set, you get an error.

Here are examples of what the path name may be on different platforms:

- **Linux**: `/home/login name`
- **Mac**: `/Users/login name`
- **Windows**: `drive letter:\documents and settings\login name or drive letter:\users\login name`

4. Move the resulting preferences11.3.nk file into your Nuke plug-in path directory.
   For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.

Your preferences now act as the defaults for your artists. However, should they make changes using the Preferences dialog, these changes will override your defaults.
Deleting (and Resetting) the Preferences

To delete (and reset) the preferences:

1. Open a terminal (or shell) as described for your operating system at the beginning of this chapter.
2. Using the prompt, go to the .nuke directory, under your home directory.
3. Enter `pwd` to display and verify the path.
   You should see something similar to
   • /users/login name/.nuke (on Linux),
   • /Users/login name/.nuke (on Mac) or
   • drive letter:\Documents and Settings\login name\.nuke or drive letter:\Users\login name\.nuke
   This is not always the case, however, because on Windows the .nuke folder can be found under
   the directory pointed to by the HOME environment variable or (if HOME is not set) the
   USERPROFILE environment variable.
   To find out if the HOME and USERPROFILE environment variables are set and where they are
   pointing at, enter `%HOME%` or `%USERPROFILE%` into the address bar in Windows Explorer. If the
   environment variable is set, the folder it’s pointing at is opened. If it’s not set, you get an error.
4. Enter `rm preferences11.3.nk` to delete the preference file.
5. Close the terminal or shell.
   The next time you launch Nuke, it rebuilds the file with the default preferences.
Altering a Script’s Lookup Tables (LUTs)

A script's lookup tables are curves that control the conversion between file or device color spaces and Nuke’s internal color space. In the Curve Editor, the x axis represents the input pixel values and the y axis the output pixel values (normalized to the 0-1 range). When applying LUTs, Nuke looks up the input value along the x axis to determine what the y value is to output.

Nuke provides many nuke-default LUTs, including: linear, sRGB, rec709, Cineon\(^1\), Gamma1.8, Gamma2.2, Panalog\(^2\), REDLog\(^3\), ViperLog\(^4\), AlexaV3LogC\(^5\), PLogLin\(^6\), SLog\(^7\), and REDSpace\(^8\).

You can also create an unlimited number of additional LUTs and edit or remove existing LUTs in the script’s settings.

By default, Nuke uses certain LUTs for certain file types or devices. In most cases, you do not need to touch these defaults. However, there may occasionally be cases when changing the defaults is necessary: for example, if your material has been shot with a camera that records in a custom color space, such as Panalog. In those cases, you can change the defaults in the script's settings so that you don’t need to change the color space on each Read or Write node.

If you do not want to use the default LUT for reading or writing certain individual images, you can select the LUT to use in the corresponding Read or Write node's controls.

---

1. The Cineon conversion is implemented as defined in Kodak’s Cineon documentation.
2. The Panalog LUT is based on a log2lin conversion with a blackpoint of 64, whitepoint of 681, and a gamma of 0.89.
3. The REDLog LUT is based on a log2lin conversion with a blackpoint of 0, whitepoint of 1023, and a gamma of 1.022.
4. The ViperLog LUT is based on a log2lin conversion with a blackpoint of 0, whitepoint of 1023, and a gamma of 1.0.
5. The Alexa LogC LUT uses the formula provided by ARRI.
6. The PLogLin LUT uses the default values for the formula, mapping log 0.43457 (code value 445 on a 1024 scale) to linear 0.18 (mid-gray) assuming a negative gamma of 0.6 and a density per code value of 0.002. (This does factor in the fact that value ranges for a log image in Nuke are still scaled to 0-1 range.)
8. The REDSpace LUT is implemented as defined in a curve provided by RED.
Managing Nuke's Native LUTs

The **Color** tab in the Nuke **Project Settings** allows you to view, add, edit, and delete Nuke's native LUTs.

*Note:* You can’t adjust LUTs that appear under the OCIO color management scheme. See [OCIO Color Management](#) for more information.

See this [Support Knowledge Base](#) article for more detailed information.

**To Display LUT Curves**

1. Select **Edit > Project Settings** to open the settings for the script.
2. Go to the **Color** tab.
3. From the list on the left, select the LUT you want to display in the curve editor. To select several LUTs, press **Ctrl** (Mac users press **Cmd**) while selecting the LUTs. All the selected LUTs are shown in the curve editor at the same time.

![LUT Curves](image)

**To Create a New LUT**

1. Select **Edit > Project Settings** to open the settings for the script.
2. Go to the **Color** tab.
3. Click the plus button (+). A dialog opens.
4. Enter a name for the new LUT and click OK.
5. Adjust the lookup curve to suit your needs. Click on the curve to select it. Ctrl/Cmd+Alt+click to add points on the curve, and drag the points to a new position. To change the shape of the curve, adjust the tangent handles.

The new LUT is now available in the global LUT settings, and the *colorspace* dropdown menu of Read and Write nodes' properties panels, the Viewer controls.

**To Edit LUTs**

1. Select Edit > Project Settings to open the settings for the script.
2. Go to the Color tab.
3. From the list on the left, select the LUT you want to edit.
4. Adjust the lookup curve to suit your needs. Click on the curve to select it. Ctrl/Cmd+Alt+click to add points on the curve, and drag the points to a new position. To change the shape of the curve, adjust the tangent handles.

   To use the usual editing commands, such as copy and paste, right-click on the curve editor and select Edit. Then, select the editing command you want to use, just like you would on any curve editor.

   **Note:** Renaming existing LUTs is currently not possible. If you want to rename a LUT, you need to add and name a new LUT, copy the information from the old LUT into the new one, and then remove the old LUT.

**To Reset the LUT Curves Back to Their Initial Default Shapes**

1. Select Edit > Project Settings to open the settings for the script.
2. Go to the Color tab.
3. From the list on the left, select the LUT you want to reset. To select several LUTs, press Ctrl/Cmd while selecting the LUTs.
4. Click reset.

**To Remove LUTs**

1. Select Edit > Project Settings to open the settings for the script.
2. Go to the **Color** tab.

3. From the list on the left, select the LUT you want to remove. Only remove LUTs that you have, for example, created by accident and are not using in your script. To remove the LUT, click the minus button (-).

The LUT is removed from the LUT settings, and the **colorspace** dropdown menu of Read and Write nodes’ properties panels.

**Note:** If you remove a LUT that is used in a node, the node continues to refer to the LUT by name and raises an error.
Selecting the LUT to Use

To select the LUT to use when reading or writing an image:

1. Double-click to open the Read or Write node’s properties panel.
2. From the **colorspace** dropdown menu, select the LUT you want to use. To use the default LUT defined in Nuke’s settings for the image type in question, select **default**.

**Default LUT Settings**

By default, Nuke uses the following LUTs in the following cases:

<table>
<thead>
<tr>
<th>LUT</th>
<th>File Type / Device</th>
<th>Default LUT</th>
</tr>
</thead>
<tbody>
<tr>
<td>working space</td>
<td>This determines what colorspace files should be converted to when read and from when written- it’s the colorspace used by Nuke under the hood. In earlier releases of Nuke, this colorspace was hidden because <strong>linear</strong> was always chosen as the <strong>working space</strong>. You may find that some operations work better in colorspaces other than <strong>linear</strong>. For example, some transforms work better in the <strong>CLog</strong> colorspace.</td>
<td>linear</td>
</tr>
<tr>
<td>monitor</td>
<td>This is used for postage stamps, OpenGL textures, the color chooser display, and all other non-Viewer image displays.</td>
<td>sRGB</td>
</tr>
<tr>
<td>8-bit files</td>
<td>This is used when reading or writing image files that contain 8-bit data. Also used by the Merge node’s sRGB switch, and to convert Primatte inputs into sRGB and outputs from sRGB.</td>
<td>sRGB</td>
</tr>
<tr>
<td>16-bit files</td>
<td>This is used when reading or writing image files that contain 16-bit integer data (not half float).</td>
<td>sRGB</td>
</tr>
<tr>
<td>log files</td>
<td>This is used when reading or writing <strong>.cin</strong> or <strong>.dpx</strong> files.</td>
<td>Cineon</td>
</tr>
<tr>
<td>float files</td>
<td>This is used when reading or writing image files that contain floating-point data.</td>
<td>linear</td>
</tr>
</tbody>
</table>

**Note:** You can only change the **working space** if you’re using OCIO **color management**.
To Change the Default LUT Settings

1. Select Edit > Project Settings to open the settings for the script.
2. Go to the Color tab.
3. From the dropdown menus under Default LUT settings, select the LUTs you want to use by default for each file type or device.

The new defaults are now used for any LUT setting where you have not selected a specific LUT. Any controls you have set to a specific LUT (that is, not set to default) continues to use the selected LUT, and only those set to default are affected.
Example Cases

Below are some examples of situations where you might need to alter the default LUTs.

Working in Video Colorspace

Emulating compositor software that works in video color space is not recommended. However, if you do need to do so, do the following:

1. Select Edit > Project Settings and go to the Color tab.
2. Under Default LUT settings, change the monitor, 8-bit files, and 16-bit files values to linear.

This prevents Nuke from converting from sRGB into linear. Nuke’s nodes still assume linear data, but the image processing is applied to your unlinearized video color space images.

Linear Data in 16-Bit Files

Some facilities use linear data in 16-bit files. If this is the case in your facility, do the following:

1. Select Edit > Project Settings and go to the Color tab.
2. Under Default LUT settings, change the 16-bit files value to linear.

Cineon Displays

Some facilities have adjusted their monitor electronics to correctly display Cineon data. If this is the case in your facility, do the following:

1. Select Edit > Project Settings and go to the Color tab.
2. Under Default LUT settings, change the monitor value to Cineon.

Color Management

Although true color management requires using other nodes, it may be useful to approximate it with a LUT that is used for the monitor setting. This way, texture maps and postage stamps resemble the final display more accurately.

If your color management is creating a monitor-corrected image, you’ll want to set monitor to sRGB so you get reasonably monitor-correct output on non-Viewer images.
OCIO Color Management

Nuke uses OpenColorIO for color management. All of the colorspace in Nuke, whether those shipped with the application or custom colorspace, are defined in OCIO config files.

Depending on the OCIO config file that you are working with, there are a number of colorspace options and roles (aliases to colorspace) that you can set in Nuke. There are also default options, which change depending on what file type you are working with. When the default option is selected, the colorspace that Nuke has set for it is listed in brackets.

Tip: Use the options in Preferences > Project Defaults > Color Management to apply them to all new projects.

1. The colorspace dropdown determines whether Nuke uses the LUTs read from the configuration specified or the Nuke native LUTs. Selecting OCIO makes the relevant OCIO LUTs available to the Read and Write nodes in scripts on a per project basis. All OCIO configurations except nuke-default automatically switch this control to OCIO.
2. Set the OpenColorIO Config you want to use for this project.
   Nuke ships with a number of default configurations, but you can use a custom OCIO config file by selecting custom from the OpenColorIO Config dropdown and then entering the file path.
   Changing the configuration updates the Default Color Transforms accordingly. If the selected configuration is invalid for certain transforms, a warning displays.
3. The working space transform determines what colorspace files should be converted to (Read) and from (Write) - it's the colorspace used by Nuke under the hood.
Note: In earlier releases of Nuke, this colorspace was hidden because **linear** was always chosen as the **working space**. You may find that some operations work better in colorspaces other than **linear**. For example, some transforms work better in the **CLog** colorspace.

4. You can use **Default Color Transforms** dropdown menus to override how clips in the Viewer, thumbnails, and so on are converted to and from the **working space**.

When the **Nuke** is selected, Reads and Writes work the same as in previous versions of Nuke, with no integrated OCIO transforms. When **OCIO** is selected:

- Reads and Writes use OCIO transforms, with no Nuke built-in LUTs applied to the image.
- Read and Write colorspace controls are populated with the list of colorspaces defined in your currently selected OCIO config.
- The **default LUT settings** dropdowns are also populated with the list of colorspaces or display transforms defined in your OCIO config. The default value for each menu match the defaults in a Nuke Studio project with the same config. These defaults can be overridden using Python callbacks. See the following path for the default implementation that ships with Nuke:

  `<install_dir>/plugins/nuke/colospsaces.py`

- The **working space** dropdown allows you to change the colorspace that Nuke uses internally for its image processing. This automatically sets the **in** colorspace of Write nodes and Viewer Processes, and the **out** colorspace for Read nodes. This defaults to the scene **linear** role defined in your OCIO config.

- Nuke Studio-created comps no longer contain automatically injected OCIOColorspace nodes. Instead, OCIO Color Management is automatically set in the comp’s **Project Settings**, and the correct OCIO colorspace is set directly into the Read and Write nodes.
Creating Custom Viewer Processes

Using look-up tables (LUTs) in Viewer Processes, you can adjust individual Viewer displays to simulate the way the image looks on output to film or some video display device. Nuke includes some predefined Viewer Process gizmos, but you can also add your own processes by registering a node or gizmo as a Viewer Process. You can register as many custom Viewer Processes as you like. If you want to use one of the 1D LUTs listed in the Project Settings in your Viewer Process, you can use the built-in gizmo called ViewerProcess_1DLUT.

**Tip:** There are a couple of commented out examples in the installed `init.py` file demonstrating how to use a 3D LUT for a Viewer Process. You can find this file in the following location:

**On Windows:**
drive letter:\Program Files\Nuke11.3v6\plugins or
drive letter:\Program Files (x86)\Nuke11.3v6\plugins

**On Mac:**
/Applications/Nuke11.3v6/Nuke11.3v6.app/Contents/MacOS/plugins

**On Linux:**
/usr/local/Nuke11.3v6/plugins

All available Viewer Processes (both custom and predefined ones) can be applied from the Viewer Process dropdown menu in the Viewer controls.

Both predefined and custom Viewer Processes can be applied from the Viewer Process dropdown menu.
Note that Viewer Processes are part of a built-in, fixed pipeline of nodes that are applied to images before they are displayed in the Viewer. This pipeline is either:

- gain > Input Process > Viewer Process > gamma > dither > channels > cliptest (if **viewer input order** has been set to **before viewer process** in the Viewer settings)

OR

- gain > Viewer Process > Input Process > gamma > dither > channels > cliptest (if **viewer input order** has been set to **after viewer process** in the Viewer settings).

However, depending on what the Viewer Process is doing, this may not be the correct order. Therefore, if your Viewer Process (or an Input Process) has controls that also exist for the Viewer, such as controls named **gain**, **gamma**, or **cliptest**, then the Viewer drives them from the corresponding Viewer controls and does not do that image processing itself. This allows you to implement these controls in your Viewer Process using whatever nodes and order you want. If your Viewer Process does not have these controls (and they are not found on any Input Process in use either), then the Viewer applies the effects in its normal way according to the built-in pipeline.

In the built-in pipeline, dither is applied to diffuse round-off errors in conversion of floating point data to the actual display bit depth. Although the cliptest is drawn at the end, it is computed on the image as input to the Viewer.
Using a Gizmo as a Custom Viewer Process

To create a custom Viewer Process, you would typically create a gizmo that includes some color correction like a look-up table (LUT) and register it as a Viewer Process using Python. (For more information on gizmos, see Sourcing Custom Plug-ins and Generic Tcl Scripts.)

If you want to use one of the 1D LUTs listed in the Project Settings in your Viewer Process, you do not need to create a custom gizmo. Instead, you can simply register a built-in gizmo called ViewerProcess_1DLUT. This gizmo takes a parameter for which LUT to use, but does not allow it to be edited. For more information, see To Register a LUT in the Project Settings as a Viewer Process.

If you want anything more complex than a 1D LUT that can be found on the LUT tab of the Project Settings, you need to create your own gizmo and register that. For more information, see To Create a Viewer Process Gizmo and To Register a Custom Viewer Process.

To Register a LUT in the Project Settings as a Viewer Process

1. Create a file called init.py in your plug-in path directory if one doesn't already exist. For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.

2. To register one of the LUTs in the Project Settings as a Viewer Process, use, for example, the following function in your init.py:
   ```python
nuke.ViewerProcess.register("Cineon", nuke.createNode, ("ViewerProcess_1DLUT", "current Cineon"))
   ```

   This registers a built-in gizmo called ViewerProcess_1DLUT as a Viewer Process and sets it to use the Cineon LUT. The registered Viewer Process appears in the Viewer Process dropdown menu as Cineon.

   Note that you can set the built-in gizmo to use any 1D LUT in the Project Settings. For example, to set it to use the Panalog LUT, use the following function:
   ```python
   ```
To Create a Viewer Process Gizmo

1. Create the node(s) you want to use as a Viewer Process. For example, you can use a ColorLookup, Vectorfield (3D LUT), or Colorspace node.

2. Select the node(s) you want to include in the Viewer Process and select Other > Group.

3. To select which controls the users of your Viewer Process can adjust, right-click on the dark gray background of the Group properties panel and select Manage User Knobs. For more information on how to add controls to your gizmo, see Creating and Accessing Gizmos.

   If you expose controls with the same name as the controls in the Viewer (such as gain or gamma), then the controls in the Viewer are used to drive these. However, if an Input Process that exposes the same controls is also in use, the Input Process takes precedence and the Viewer controls drive it, ignoring the same-named Viewer Process control(s). For more information on Input Processes, see Using the Viewer Controls > Input Process and Viewer Controls.

4. Once you are happy with the modified Viewer Process group, export it to a gizmo by clicking export as gizmo on the Node tab of the group controls.

5. In the file browser that appears, click Home. Type .nuke/ after the path displayed at the bottom of the file browser. Enter a name after the path, and append a .gizmo extension after the name. The name should begin with a capital letter. Finally, click Save.

6. Proceed to registering the gizmo as a custom Viewer Process, described below.

   **Tip:** If you like, you can test your Viewer Process gizmo as an Input Process before registering it. Do the following:
   1. In the top right corner of the Viewer, set the Viewer Process dropdown menu to None.
   2. Select the gizmo in the Node Graph.
   3. To toggle the Input Process on or off, click the IP button in the Viewer controls. If you are happy with the result, proceed to registering the gizmo as a Viewer Process.

   For more information on Input Processes, see Using the Viewer Controls > Input Process and Viewer Controls.

   **Tip:** If you want to view or modify the internals of an existing Viewer Process, you can do the following:
   1. Select the Viewer Process that you want to modify from the Viewer Process dropdown menu.
   2. Select Edit > Node > Copy Viewer Process to Node Graph. This inserts the Viewer Process gizmo you selected in the Node Graph.
   3. Double-click on the gizmo to open its controls. Go to the Node tab and click copy to
group. This gives you an editable group version of the gizmo.

4. In the Group controls, click the S button to show the internals of the group. They are shown on a new tab in the Node Graph.

5. Make your changes and export the group to a gizmo by clicking export as gizmo on the Node tab of the group controls.

Tip: If you use the ViewerLUT node in a Viewer Process gizmo, you can toggle rbg_only in the ViewerLUT controls to define whether the LUT is applied to all channels or only the red, green, and blue channels. You can also expose this control in the Viewer Process gizmo's controls, so that users can set it themselves.

To Register a Custom Viewer Process

1. Create a file called init.py in your plug-in path directory if one doesn't already exist. For more information on plug-in path directories, see Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts.

2. To register a gizmo or a node as a Viewer Process, use the following function in your init.py:

   ```python
   nuke.ViewerProcess.register()
   ```

   For example, to register a gizmo called MyProcess.gizmo as a Viewer Process and have it appear in the Viewer Process dropdown menu as My Custom Process, you would enter the following:

   ```python
   ```

   Your Viewer Process should now appear in the Viewer controls.

   If you need to unregister a Viewer Process, you can use `nuke.ViewerProcess.unregister()`. For example:
nuke.ViewerProcess.unregister("My Custom Process").

To get help on the use of these statements, you can enter help(nuke.ViewerProcess) in the Script Editor.

**Tip:** You can also pass arguments to nuke.ViewerProcess.register(). For example, to register a Blur node with its size knob set to 10, you would enter the following:

nuke.ViewerProcess.register("Blur", nuke.createNode, ("Blur", "size 10"))

**Tip:** You can easily register any LUT defined in the project settings as a Viewer Process. For how to do this, see the installed menu.py file where the built-in Viewer Processes are registered. You can find menu.py in the following location:

**On Windows:**
drive letter:\Program Files\Nuke11.3v6\plugins or
drive letter:\Program Files (x86)\Nuke11.3v6\plugins

**On Mac:**
/Applications/Nuke11.3v6/Nuke11.3v6.app/Contents/MacOS/plugins

**On Linux:**
/usr/local/Nuke11.3v6/plugins
Applying Custom Viewer Processes to Images

In the Viewer controls, you can apply a custom Viewer process to images displayed in the Viewer and open the controls for the currently active Viewer process.

To Apply Your Custom Viewer Process to Images Displayed in a Viewer

Select the process from the Viewer Process dropdown menu in the Viewer controls.

To View the Controls of the Currently Active Viewer Process

In the Viewer controls, select show panel from the Viewer Process dropdown menu.

This opens the Viewer Process’ properties panel. Any controls with the same name as the controls in the Viewer (such as gain or gamma) can only be adjusted using the Viewer controls. If these controls are also exposed on an Input Process and the Input Process has been activated, the Viewer controls drive the Input Process controls and the Viewer Process controls are disabled.

For more information on Input Processes, see Using the Viewer Controls > Input Process and Viewer Controls.
Expressions

This chapter is intended as a primer on how to apply expressions (programmatic commands) to Nuke parameters. It explains how to perform some common tasks with expressions (for example, how to link the values of one parameter to another), and concludes with a table all the functions that you may include as part of an expression.

Quick Start

Here's a quick overview of the workflow:

1. You enter Nuke expressions in the Expression dialog, which you can open either by pressing the equals sign (=) on a parameter or by right-clicking on it and selecting Add expression.

2. In the Expression dialog, enter text that either references values from other parameters (creating a linking expression - see Linking Expressions) or applies mathematical functions of some kind to the current values (see Adding Mathematical Functions to Expressions). An example of the former would be parent.Transform1.rotate, which indicates that this control takes its values from the parent control, Transform node's rotate control.

3. If necessary, you can also convert expressions between scripting languages (that is, between Nuke expressions, Tcl, and Python). See Converting Expressions Between Scripting Languages.
## Linking Expressions

Through expressions, you can link the parameters from one node and control the values of the parameters in other nodes. When creating a linking expression, type the elements listed in the table below; remember to separate each element with a period.

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node name</td>
<td>The node with the source parameter (i.e., Transform1).</td>
</tr>
<tr>
<td>Parameter name</td>
<td>The name of the parameter with the source value (for example, translate). The name is defined internally, and may not match the parameter’s label that appear in the Nuke interface. If necessary, hover over the parameter’s field with your mouse pointer and its name appears in the pop-up tool tip.</td>
</tr>
<tr>
<td>Child parameter name</td>
<td>Some parameters include child parameters, such as the fields for x and y axes, or red, green, and blue color channels. Child parameter names do match the label that appears before the parameter’s field (for example, x). (optional)</td>
</tr>
<tr>
<td>Time (optional)</td>
<td>By default, linking expressions pull values from the current frame number, but you can read values from other frames, either statically or dynamically (that is, with a temporal offset). If you want to read in a static value for a given frame, you just type that frame number inside a set of parenthesis (for example, (10)). If you want to read in dynamic values but with an offset in time, type t, the variable for time, followed by a + (for a forward offset) or - (for a backward offset), followed by a number representing the number of frames worth of offset. For example, typing (t-2) would capture values that are two frames back from the current frame.</td>
</tr>
</tbody>
</table>

Thus, to create a linking expression that pulls the value from a Transform node’s x translation field at the tenth frame, you would type = on a parameter to open the expression dialog, and then enter `Transform1.translate.x(10)` in the dialog’s Expression field.
The steps below recap the process for creating a linking expression.

**Referencing Values from Another Parameter**

To Reference Values from Another Parameter - Method 1:
1. Click on the destination parameter (the one which receives values from another parameter).
2. To display the expression dialog, right-click on the parameter and select **Add expression**, OR type = in the parameter field.

3. In the dialog that opens, type the name of the node containing the source parameter and a period. (Each node prominently displays its name on its face.)

4. If you want to enter a multi-line expression, you can click the multi-line edit field button .
5. Follow the name of the node by the source parameter’s name and a period. (If you don’t know the parameter’s name, you can hover over its field in order to see it displayed in a tool tip.)

6. Optionally, type the child parameter's name and a period.

7. Optionally, type a frame number or offset variable in brackets (for example, (2) or (t-2)) in order to specify the frame or range of frames from which you pull values.

8. Next to the expression entry field, you can click the Py button to automatically make your expression a Python callback. You can also toggle the R button to have your expression interpreted as an expression or as a series of statements. For example, with the multi-line edit mode and the Python mode on, you could enter the following expression, and get 15 as the resulting value:

```python
def example():
    a = 5
    return a

def example2():
    b = 10
    return b

ret = example() + example2()
```

9. Click OK. This links the parameters, which turn blue. In the Node Graph, a green arrow appears between the nodes to indicate that they are linked via an expression.

10. To edit the expression later on, right-click on the parameter and select Edit expression (or press = on the parameter). You can also click the animation button and select Edit expression to edit expressions for all the parameters next to the button.
To Reference Values from Another Parameter - Method 2:

1. **Ctrl/Cmd**+drag the parameter that has the values you want to use on top of the parameter that receives these values. This links the parameters, which turn blue. In the Node Graph, a green arrow appears between the nodes to indicate that they are linked via an expression.

To view or edit the expression, right-click on the parameter and select **Edit expression**.

2. If you want to link several parameters at the same time, **Ctrl/Cmd**+drag the animation button next to the source parameters on top of the animation button next to the destination parameters. To view or edit the expressions used to link the parameters, click the animation button and select **Edit expression**.

**Linking Channels and Formats Using Expressions**

You can also create expression links to connect channel, layer, and format controls with other controls in various nodes. Since these controls aren't meant to be animated, you can't use the full range of Nuke expressions, nor can you use Python or Tcl languages. You can link controls using the **Link menu** next to the control on the properties panel:

1. Click the **Link menu** and select **Set link**. An **Expression** dialog opens.
2. Enter your expression in the Expression field and click **OK**.
3. You can edit an existing link by clicking the **Link menu** and selecting **Edit link**.
4. You can also **Ctrl/Cmd**+drag the **Link menu** to another control to create a link between the two.
5. To remove a link, click the **Link menu** and select **Remove link**.
Adding Mathematical Functions to Expressions

You can incorporate mathematical functions into parameters. For example, you might negate an expression in order to invert a tracking curve which you wish to use to stabilize an element (such an expression might resemble the following: -(Transform1.translate.x)).

You can also rely on a function to add more complex mathematical operation to your expressions. The table below list all the functions which you may incorporate into Nuke expressions.

<table>
<thead>
<tr>
<th>Function</th>
<th>Purpose</th>
<th>Operator Usage</th>
<th>Related Functions</th>
<th>DeepExpression Compatible</th>
</tr>
</thead>
<tbody>
<tr>
<td>abs (x)</td>
<td>Returns the absolute value of the floating-point number x.</td>
<td>x</td>
<td>See also: fabs.</td>
<td>✔️</td>
</tr>
<tr>
<td>acos (x)</td>
<td>Calculates the arc cosine of x; that is the value whose cosine is x.</td>
<td>If x is less than -1 or greater than 1, acos returns nan (not a number).</td>
<td>See also: cos, cosh, asin, atan.</td>
<td>✔️</td>
</tr>
<tr>
<td>asin (x)</td>
<td>Calculates the arc sine of x; that is the value whose sine is x.</td>
<td>If x is less than -1 or greater than 1, asin returns nan (not a number).</td>
<td>See also: sin, sinh, acos, atan.</td>
<td>✔️</td>
</tr>
<tr>
<td>atan (x)</td>
<td>Calculates the arc tangent of x; that is the value whose tangent is x. The return value is between -PI/2 and PI/2.</td>
<td>x</td>
<td>See also: tan, tanh, acos, asin, atan2.</td>
<td>✔️</td>
</tr>
<tr>
<td>atan2 (x, y)</td>
<td>Calculates the arc tangent of x, y</td>
<td>See also: sin, tanh, acos, asin, atan2.</td>
<td>✔️</td>
<td></td>
</tr>
<tr>
<td>Function</td>
<td>Purpose</td>
<td>Operator Usage</td>
<td>Related Functions</td>
<td>DeepExpression Compatible</td>
</tr>
<tr>
<td>----------</td>
<td>---------</td>
<td>----------------</td>
<td>-------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ceil (x)</td>
<td>Round x up to the nearest integer.</td>
<td>x</td>
<td>See also: floor, trunc, rint.</td>
<td>●</td>
</tr>
<tr>
<td>clamp (x, min, max)</td>
<td>Return x clamped to [min ... max].</td>
<td>x, min, max</td>
<td>See also: min, max.</td>
<td>●</td>
</tr>
<tr>
<td>clamp (x)</td>
<td>Return x clamped to [0.0 ... 1.0].</td>
<td>x</td>
<td>See also: min, max.</td>
<td>●</td>
</tr>
<tr>
<td>cos (x)</td>
<td>Returns the cosine of x.</td>
<td>x in radians</td>
<td>See also: acos, sin, tan, cenh.</td>
<td>●</td>
</tr>
<tr>
<td>cosh (x)</td>
<td>Returns the hyperbolic cosine of x, which is defined mathematically as ((\exp(x) + \exp(-x))/2).</td>
<td>x</td>
<td>See also: cos, acos, sinh, tanh.</td>
<td>●</td>
</tr>
<tr>
<td>curve (frame)</td>
<td>Returns the y value of the animation curve at the given frame.</td>
<td>optional: frame, defaults to current frame.</td>
<td>See also: value, y.</td>
<td>●</td>
</tr>
<tr>
<td>degrees (x)</td>
<td>Convert the angle x from radians into degrees.</td>
<td>x</td>
<td>See also: radians.</td>
<td>●</td>
</tr>
<tr>
<td>exp (x)</td>
<td>Returns the value of (e) (the base of natural logarithms) raised to the power of x.</td>
<td>x</td>
<td>See also: log, log10.</td>
<td>●</td>
</tr>
<tr>
<td>exponent (x)</td>
<td>Exponent of x.</td>
<td>x</td>
<td>See also: mantissa, ldexp.</td>
<td>●</td>
</tr>
<tr>
<td>fBm (x, y, z, octaves, lacunarity)</td>
<td>Fractional Brownian Motion. This is the sum of octaves</td>
<td>x, y, z, octaves, lacunarity, gain</td>
<td>See also: noise, random, turbulence.</td>
<td>●</td>
</tr>
<tr>
<td>Function</td>
<td>Purpose</td>
<td>Operator Usage</td>
<td>Related Functions</td>
<td>DeepExpression Compatible</td>
</tr>
<tr>
<td>--------------------------</td>
<td>-------------------------------------------------------------------------</td>
<td>----------------</td>
<td>-------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>gain)</td>
<td>calls to noise(). For each of them the input point is multiplied by pow (lacunarity,i) and the result is multiplied by pow(gain,i). For normal use, lacunarity should be greater than 1 and gain should be less than 1.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>fabs (x)</td>
<td>Returns the absolute value of the floating-point number x.</td>
<td>x</td>
<td>See also: abs.</td>
<td></td>
</tr>
<tr>
<td>false ()</td>
<td>Always returns 0</td>
<td></td>
<td>See also: true.</td>
<td></td>
</tr>
<tr>
<td>floor (x)</td>
<td>Round x down to the nearest integer.</td>
<td>x</td>
<td>See also: ceil, trunc, rint.</td>
<td></td>
</tr>
<tr>
<td>fmod (x, y)</td>
<td>Computes the remainder of dividing x by y. The return value is x - n y, where n is the quotient of x / y, rounded towards zero to an integer.</td>
<td>x, y</td>
<td>See also: ceil, floor.</td>
<td></td>
</tr>
<tr>
<td>frame ()</td>
<td>Return the current frame number.</td>
<td></td>
<td>See also: x.</td>
<td></td>
</tr>
<tr>
<td>from_byte (color component)</td>
<td>Converts an sRGB pixel value to a linear value.</td>
<td>color_component</td>
<td>See also: to_sRGB, to_rec709f, from_rec709f.</td>
<td></td>
</tr>
<tr>
<td>from_rec709f (color component)</td>
<td>Converts a rec709 byte value to a linear brightness</td>
<td>color_component</td>
<td>See also: form_sRGB, to_rec709f.</td>
<td></td>
</tr>
<tr>
<td>from_sRGB (color)</td>
<td>Converts an sRGB pixel value to a linear value.</td>
<td>color_component</td>
<td>See also: to_sRGB, to_</td>
<td></td>
</tr>
<tr>
<td>Function</td>
<td>Purpose</td>
<td>Operator Usage</td>
<td>Related Functions</td>
<td>DeepExpression Compatible</td>
</tr>
<tr>
<td>------------</td>
<td>-------------------------------------------------------------------------</td>
<td>----------------</td>
<td>-------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>component)</td>
<td></td>
<td></td>
<td>rec709f, from_rec709f</td>
<td></td>
</tr>
<tr>
<td>hypot (x, y)</td>
<td>Returns the sqrt(x<em>x + y</em>y). This is the length of the hypotenuse of a right-angle triangle with sides of length x and y.</td>
<td>x, y</td>
<td>See also: atan2.</td>
<td></td>
</tr>
<tr>
<td>int (x)</td>
<td>Round x to the nearest integer not larger in absolute value.</td>
<td>x</td>
<td>See also: ceil, floor, trunc, rint.</td>
<td></td>
</tr>
<tr>
<td>ldexp (x, exp)</td>
<td>Returns the result of multiplying the floating-point number x by 2 raised to the power exp.</td>
<td>x, exp</td>
<td>See also: exponent.</td>
<td></td>
</tr>
<tr>
<td>lerp (a, b, x)</td>
<td>Returns a point on the line f (x) where f(0)==a and f(1)==b. Matches the lerp function in other shading languages.</td>
<td>a, b, x</td>
<td>See also: step, smoothstep.</td>
<td></td>
</tr>
<tr>
<td>log (x)</td>
<td>Returns the natural logarithm of x.</td>
<td>x</td>
<td>See also: log10, exp.</td>
<td></td>
</tr>
<tr>
<td>log10 (x)</td>
<td>Returns the base-10 logarithm of x.</td>
<td>x</td>
<td>See also: log, exp.</td>
<td></td>
</tr>
<tr>
<td>logb (x)</td>
<td>Same as exponent().</td>
<td>x</td>
<td>See also: mantissa, exponent.</td>
<td></td>
</tr>
<tr>
<td>mantissa (x)</td>
<td>Returns the normalized fraction. If the argument x is not zero, the normalized fraction is x times a power of two, and is always in the range 1/2 (inclusive) to 1</td>
<td>x</td>
<td>See also: exponent.</td>
<td></td>
</tr>
<tr>
<td>Function</td>
<td>Purpose</td>
<td>Operator Usage</td>
<td>Related Functions</td>
<td>DeepExpression Compatible</td>
</tr>
<tr>
<td>----------</td>
<td>---------</td>
<td>---------------</td>
<td>-------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>(exclusive). If x is zero, then the normalized fraction is zero and exponent() Returns zero.</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>max (x, y, ... )</td>
<td>Return the greatest of all values.</td>
<td>x, y, (...)</td>
<td>See also: min, clamp.</td>
<td>☺</td>
</tr>
<tr>
<td>min (x, y, ... )</td>
<td>Return the smallest of all values.</td>
<td>x, y, (...)</td>
<td>See also: max, clamp.</td>
<td>☺</td>
</tr>
<tr>
<td>mix (a, b, x)</td>
<td>Same as lerp().</td>
<td>a, b, x</td>
<td>See also: step, smoothstep, lerp.</td>
<td>☺</td>
</tr>
<tr>
<td>noise (x, y, z)</td>
<td>Creates a 3D Perlin noise value. This produces a signed range centered on zero. The absolute maximum range is from -1.0 to 1.0. This produces zero at all integers, so you should rotate the coordinates somewhat (add a fraction of y and z to x, etc.) if you want to use this for random number generation.</td>
<td>x, optional y, optional z</td>
<td>See also: random, fBm, turbulence.</td>
<td>☺</td>
</tr>
<tr>
<td>pi ()</td>
<td>Returns the value for pi (3.141592654...).</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>pow (x, y)</td>
<td>Returns the value of x raised to the power of y.</td>
<td>x, y</td>
<td>See also: log, exp, pow.</td>
<td>☺</td>
</tr>
<tr>
<td>pow2 (x)</td>
<td>Returns the value of x raised to the power of 2.</td>
<td>x, y</td>
<td>See also: pow.</td>
<td>☺</td>
</tr>
<tr>
<td>radians (x)</td>
<td>Convert the angle x from degrees into radians.</td>
<td>x</td>
<td>See also: degrees.</td>
<td>☺</td>
</tr>
<tr>
<td>Function</td>
<td>Purpose</td>
<td>Operator Usage</td>
<td>Related Functions</td>
<td>DeepExpression Compatible</td>
</tr>
<tr>
<td>----------</td>
<td>---------</td>
<td>----------------</td>
<td>-------------------</td>
<td>--------------------------</td>
</tr>
<tr>
<td>random (x, y, z)</td>
<td>Creates a pseudo random value between 0 and 1. It always generates the same value for the same x, y and z. Calling random with no arguments creates a different value on every invocation.</td>
<td>optional x, optional y, optional z</td>
<td>See also: noise, fBm, turbulence.</td>
<td>○</td>
</tr>
<tr>
<td>rint (x)</td>
<td>Round x to the nearest integer.</td>
<td>x</td>
<td>See also: ceil, floor, int, trunc.</td>
<td>○</td>
</tr>
<tr>
<td>sin (x)</td>
<td>Returns the sine of x.</td>
<td>x in radians</td>
<td>See also: asin, cos, tan, sinh.</td>
<td>○</td>
</tr>
<tr>
<td>sinh (x)</td>
<td>Returns the hyperbolic sine of x, which is defined mathematically as (exp(x) - exp(-x)) / 2.</td>
<td>x</td>
<td>See also: sin, asin, cosh, tanh.</td>
<td>○</td>
</tr>
<tr>
<td>smoothstep (a, b, x)</td>
<td>Returns 0 if x is less than a, returns 1 if x is greater or equal to b, returns a smooth cubic interpolation otherwise. Matches the smoothstep function in other shading languages.</td>
<td>a, b, x</td>
<td>See also: step, lerp.</td>
<td>○</td>
</tr>
<tr>
<td>sqrt (x)</td>
<td>Returns the non-negative square root of x.</td>
<td>x</td>
<td>See also: pow, pow2.</td>
<td>○</td>
</tr>
<tr>
<td>step (a, x)</td>
<td>Returns 0 if x is less than a, returns 1 otherwise. Matches the step function other shading languages.</td>
<td>a, x</td>
<td>See also: smoothstep, lerp.</td>
<td>○</td>
</tr>
<tr>
<td>tan (x)</td>
<td>Returns the tangent of x.</td>
<td>x in radians</td>
<td>See also: atan, cos, sin, tanh, atan2.</td>
<td>○</td>
</tr>
<tr>
<td>Function</td>
<td>Purpose</td>
<td>Operator Usage</td>
<td>Related Functions</td>
<td>DeepExpression Compatible</td>
</tr>
<tr>
<td>------------------</td>
<td>-------------------------------------------------------------------------</td>
<td>----------------</td>
<td>-----------------------------------------------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>tanh (x)</td>
<td>Returns the hyperbolic tangent of x, which is defined mathematically as sinh(x) / cosh(x).</td>
<td>x</td>
<td>See also: tan, atan, sinh, cosh.</td>
<td></td>
</tr>
<tr>
<td>to_byte (color component)</td>
<td>Converts a floating point pixel value to an 8-bit value that represents that number in sRGB space.</td>
<td>color_component</td>
<td>See also: form_sRGB, to_rec709f, from_rec709f.</td>
<td></td>
</tr>
<tr>
<td>to_rec709f (color component)</td>
<td>Converts a floating point pixel value to an 8-bit value that represents that brightness in the rec709 standard when that standard is mapped to the 0-255 range.</td>
<td>color_component</td>
<td>See also: form_sRGB, from_rec709f.</td>
<td></td>
</tr>
<tr>
<td>to_sRGB (color component)</td>
<td>Converts a floating point pixel value to an 8-bit value that represents that number in sRGB space.</td>
<td>color_component</td>
<td>See also: form_sRGB, to_rec709f, from_rec709f.</td>
<td></td>
</tr>
<tr>
<td>true ()</td>
<td>Always Returns 1.</td>
<td></td>
<td>See also: false</td>
<td></td>
</tr>
<tr>
<td>trunc (x)</td>
<td>Round x to the nearest integer not larger in absolute value.</td>
<td>x</td>
<td>See also: ceil, floor, int, rint.</td>
<td></td>
</tr>
<tr>
<td>turbulence (x, y, z, octaves, lucanarity, gain)</td>
<td>This is the same as fBm() except the absolute value of the noise() function is used.</td>
<td>x, y, z, octaves, lucanarity, gain</td>
<td>See also: fBm, noise, random.</td>
<td></td>
</tr>
<tr>
<td>value (frame)</td>
<td>Evaluates the y value for an animation at the given frame.</td>
<td>optional: frame, defaults to current frame.</td>
<td>See also: y, curve.</td>
<td></td>
</tr>
<tr>
<td>Function</td>
<td>Purpose</td>
<td>Operator Usage</td>
<td>Related Functions</td>
<td>DeepExpression Compatible</td>
</tr>
<tr>
<td>----------</td>
<td>---------</td>
<td>----------------</td>
<td>-------------------</td>
<td>--------------------------</td>
</tr>
<tr>
<td>x ()</td>
<td>Return the current frame number.</td>
<td></td>
<td>See also: frame.</td>
<td></td>
</tr>
<tr>
<td>y (frame)</td>
<td>Evaluates the y value for an animation at the given frame.</td>
<td>optional: frame, defaults to current frame.</td>
<td>See also: value, curve.</td>
<td></td>
</tr>
</tbody>
</table>
Converting Expressions Between Scripting Languages

Depending on where you need to use an expression, you might find that you want to, for example, convert Nuke expressions to Tcl expressions or embed Python functions in a Nuke expression. The different languages are used in different parts of Nuke:

- Python can be used in the Script Editor, in the Script Command (File > Comp Script Command) and in scripts run when Nuke starts (such as init.py and menu.py). For more information, see the Nuke Python documentation (Help > Documentation).
- Tcl can be used in most string knobs (where text other than just numbers can be entered), in the Script Command dialog (File > Comp Script Command), to open some compatibility start up scripts (such as init.tcl and formats.tcl).
- Nuke expressions can be used on the Add Expression dialog with most knobs in Nuke and expression entry field in the Expression node.

You can use the following functions to use different types of expressions together:

- `nuke.expression()` to use a Nuke expression in Python code.
- `expression` to use a Nuke expression in Tcl.
- `nuke.tcl()` to run Tcl code in Python.
- `python` to run Python code in Tcl.
- `[ ]` (square brackets) to embed Tcl in a Nuke expression (or a string knob).
- `[python {...}]` to embed Python in a Nuke expression.

**Tip:** Note that putting braces ( `{} `) around Python code when embedding it in Tcl may make the process a bit easier, because this prevents Tcl from performing its own evaluation of the Python code before passing it through to the Python interpreter. For example: `[python {"hello " + "world"}]`

**Tip:** Note that the "python" Tcl command by default evaluates a single line of code and returns the result. Use the "-exec" option (for example, "python -exec") if you want to run multiple lines. Please refer to the Nuke Tcl Scripting documentation (Help > Documentation > TCL Scripting) for further information.
The Script Editor and Python

Nuke ships with a comprehensive Python application programming interface (API), enabling you to perform user interface actions using Python scripting. This chapter describes how you can use the Script Editor for executing Python scripts and directs you to sources of more information on Python.

Quick Start

Here's a quick overview of the workflow:

1. Enter Python statements in Nuke's Script Editor to perform the required actions.
2. Save your script with the extension .py in a directory that is contained in the sys.path variable.
3. Later, when you want to execute the same statement sequence, import the .py file into Nuke's Script Editor. Nuke executes the statements in the specified order.

If you need more information on using Python in Nuke, you can always turn to the Nuke Python Developer's Guide (Help > Documentation).

Tip: You can import Nuke as a module into a third-party Python 2.7.13 interpreter. See Nuke as a Python Module for more information.

Tip: You can also run an interactive Python session on the command line with nuke -t.
Using the Script Editor

If you're not using a third-party Python interpreter, you can type Python scripts into Nuke's Script Editor.

Opening the Script Editor

To open the Script Editor, click on one of the content menus and select Script Editor from the menu that opens.

Input and Output Panes

The Script Editor is divided into two parts, as shown in the figure below. You use the lower part (input pane) to type in and execute your Python statement, and when you have done so, statements and their outputs appear in the upper part of the editor (output pane). Successfully executed statements are followed by a hash mark (#).

To hide the output or input pane, click the Show input only or Show output only button on top of the Script Editor.

To show both panes again, click the Show both input and output button.

Entering a Statement

To enter a statement in the Script Editor:
1. Click on the input pane of the editor to insert the cursor there.
2. Type in your statement. To use the usual editing functions, such as copy and paste, right-click on the editor and select the desired function.

When entering the statement, you'll notice that any words that are Python's keywords (such as `print` and `import`) turn green, while strings (basically, anything in quotation marks) become either red or cyan. Comments are shown in yellow.

If you like, you can change these colors and the font on the Script Editor tab of the Preferences dialog. To open the preferences, press `Shift+S`.

![Image of Script Editor with example code]

**Tip:** You can also use auto-complete to help you with entering Python statements. Start writing a command and press the Tab key. If there's only one way to end your command, Nuke auto-completes it straight away. If there are several possible completions, Nuke gives you a pop-up menu listing them. If there's no known way to complete your command, nothing happens. Even if your command is automatically completed, it is not executed automatically, just in case you don't like surprise side effects.

3. If your statement includes several lines or you want to enter several statements at once, press Return to move to the next line.
4. To execute the statement, click the Run the current script button on the top of the Editor, or press Ctrl/Cmd+Return.

**Tip:** You can also execute statements by pressing Ctrl/Cmd+Enter on the numeric keypad.

By default, successful statements disappear from the input pane, and appear in the output pane. However, if you want all statements to stay in the input pane after they are executed, you can do the following:

1. Press `Shift+S` to open the Preferences dialog.
2. Go to the Script Editor tab.
3. Uncheck clear input window on successful script execution.
4. Click Close to save the preference for the current project only, or Save_prefs to save the preference for the current and future projects.

If you enter an invalid statement, Nuke produces an error in the output pane of the Script Editor, leaving the invalid statement in the input pane. Correct the statement and execute it again until you get it right.
Note: Sometimes you may get an error if you copy and paste statements into the Script Editor from another source, like an e-mail. This may be caused by the mark-up or encoding of the source you copied the statement from. To fix the problem, re-enter the statement manually.

If you want to have all executed Python commands appear in the output pane of the Script Editor, open the Preferences dialog (press Shift+S), go to the Script Editor tab, and check echo all commands to output window. This applies to both commands executed by yourself and by Nuke. For example, if you select a node from the Toolbar, the corresponding Python command is displayed in the output pane. This does not apply to all actions you take in the graphical user interface, however, but only those that are performed by executing Python script commands.

To only execute part of a script, enter the script in the input pane and select the part you want to execute. Press Ctrl/Cmd+Return. Nuke runs the selected part of the script, leaving the script in the input pane.

To repeat a statement, click the Previous Script button on top of the Editor to move back to the previous statement. You can do this until you reach the statement you want to repeat. To execute the statement again, press Ctrl/Cmd+Enter.

To increase the indentation in the input window, press Tab.

To decrease the indentation in the input window, press Shift+Tab.

Moving Through and Clearing the Script History

In addition to stepping backwards through the history of your script, you can also step forwards. Click the Next script button to move forward through your statements.

To clear the history, click the Clear history button.

Clearing the Output Pane

Click the Clear output window button (or press Ctrl/Cmd+Backspace).
Automating Procedures

Okay, so you know how to use the Script Editor to type in a sequence of Python statements that take care of a procedure. But so far, you've still sat by your computer typing the statements in. It's time to automate the procedure. All you need to do is save your statements, and when you want to use them again later, import them into the Script Editor.

Saving Statements in a Python Module

To save statements in a Python module:

1. On the top of the Script Editor, click the Save a script button.
2. Save the script with the extension .py (for example firstmodule.py) in a directory that is contained in the sys.path variable. (To see these directories, enter print sys.path in the Script Editor. To add a directory to the sys.path variable, enter sys.path.append ("directory") where directory represents the directory you want to add.)

You have now created your first Python module.

Opening a Python Script in the Script Editor

To open a Python script in the Script Editor:

1. Click the Load a script button on top of the Script Editor. The Script to open dialog opens.
2. Navigate to the Python module that contains the script you want to open and click Open.

Nuke opens the script in the input pane of the Script Editor, but does not execute it.

Importing and Executing a Python Script

To import and execute a Python script:

1. On top of the Script Editor, click the Source a script button. The Script to open dialog opens.
2. Navigate to the Python module that contains the script you want to import and click Open.

OR
In the input pane, enter:

```python
import module
```

Where `module` represents the name of your Python module without the file extension, for example:

```python
import firstmodule
```

Nuke imports the Python module and performs the procedure defined in the module.

---

**Note:** Importing the module is done according to Python’s default rules. During the import, the module is searched in the following locations and order:

1. In the current directory.
2. In the directories contained in the `PYTHONPATH` environment variable, if this has been defined. (To view these directories, enter `echo $PYTHONPATH` in a command shell.)
3. In an installation-dependent default directory.

During the search, the variable `sys.path` is initialized from these directories. Modules are then searched in the directories listed by the `sys.path` variable. To see these directories, execute the statement `print sys.path` in the Script Editor.
Nuke as a Python Module

You can import Nuke as a module into a third-party Python 2.7.13 interpreter, granting full access to the Nuke Python-API, but from within a native Python interpreter instead of Nuke.

**Note:** Foundry cannot provide customer support for third-party Python interpreters.

To run Nuke as a Python module:

1. Add the file path for Nuke's `site-packages` directory to the `usrlocal.pth` file in your Python 2.7.13 install.
   
   For example, if you’re running on Windows, add `C:\Program Files\Nuke 11.3v6\lib\site-packages` to the `usrlocal.pth` file.

   **Tip:** You can also use relative paths to the directory containing the `usrlocal.pth` file.

2. At the Python prompt, use the `import nuke` declaration to make Nuke’s Script Editor functions and commands (such as `nuke.nodes.Blur()` to add a Blur node) available in your chosen Python interpreter.

   The `import nuke` function checks-out a nuke_r render license by default. If you want to use Nuke interactively, and you have a nuke_i license, set the `NUKE_INTERACTIVE` environment variable to 1. See [Environment Variables](#) for more information on setting environment variables.

For more information on using Nuke as a Python module, select **Help > Documentation** from Nuke’s menu bar and navigate to **Python Developers Guide > Nuke as a Python Module**.
Getting Help

In the scope of this user guide, it’s not possible to go into detail with Python and all the scripts available. However, there are several sources of more information that you may find useful if you need help using Python.

More Documentation


You may also want to read the Nuke Python API reference documentation under Help > Documentation > PythonScriptingReference.

Viewing More Examples

We only described a few examples of Python scripts in this chapter, but there are more. You can find them in the following location:

- **On Windows:**

  drive letter:\Program Files\Nuke11.3v6\plugins\nukescripts or

  drive letter:\Program Files (x86)\Nuke11.3v6\plugins\nukescripts

- **On Mac:**

  /Applications/Nuke11.3v6/Nuke11.3v6.app/Contents/MacOS/plugins/nukescripts

- **On Linux:**

  /usr/local/Nuke11.3v6/plugins/nukescripts

To view an example, select one of the .py files and open it in any text editor.

Using the Help Statement

Possibly the quickest way of getting help on the available commands is to enter the following in the Script Editor:

help (nuke)
This statement lists many of the available Python commands in an alphabetical order together with their descriptions.

You can also get help on more specific things. For example, the following statement gives you a description of what the `setValue()` method does:

```python
help(nuke.Knob.setValue)
```

**Python on the Web**

To read more about Python, check out its documentation, or interact with other Python users. Visit the Python programming language official website at [http://www.python.org/](http://www.python.org/).
Timeline Editing in Nuke Studio

Nuke Studio’s timeline is designed to provide shot management, conform, and playback capabilities for people creating visual effects and delivers visual effects sequences without resorting to other third party applications. These are the topics covered:

- **Using Tags** tells you how to quickly sort or filter source clips and shots for better visibility, organization, and export.
- **Viewing Metadata** describes how to examine extra clip data and filter your project to find the footage you need.
- **Conforming Sequences** describes the process of matching up the footage from a shoot with the required edit decisions to create a meaningful timeline.
- **Managing Timelines** describes the timeline interface and how to add and manage footage on timelines.
- **Soft Effects** tells you how to add real-time GPU effects to timeline shots.
- **Create Comp** explains the difference between comps and regular shots and how to create and manage them.
- **Annotations** allow you to add editorial comments to your timeline output, enabling collaborative work between Nuke Studio and other Nuke workstations.
- **Timeline Editing Tools** describes how you manipulate your shots directly in the timeline using a series of modal editorial tools that complement the **Multi Tool**.
- **Versions and Snapshots** describes how to record the different states of your workflow as you progress using versions and snapshots.
- **Exporting from Nuke Studio** deals primarily with Nuke Studio's shot management and export functionality when you're farming out shots or sequences to other artists. It also deals with presets, which dictate how **Create Comp** passes data between the Timeline and Compositing environments.
Using Tags

Tags are used by Nuke Studio to quickly sort or filter clips and shots for better visibility, organization, and export. Tags are used to mark shots of a particular type or content as you organize your project. The default tags supplied include Approved, Note, Reference, and other general purpose tags. You can also create custom tags by right-clicking in the Tags tab or by pressing Ctrl/Cmd+Y in the Tags tab. You can apply tags to clips, shots and Comp shots, individual frames, sequences, and tracks.

Clip and shot tags and notes can be added to exports using the burn-in feature. See Adding Burn-in Text to Exports for more information.

Tags can also be converted to Final Cut Pro or Premiere markers during export to XML. See Exporting EDLs and XMLs for more information.

Using Quick Tags

Quick tags allow you to add tags, depending on context, by right-clicking a selection and then choosing the type of tag to apply. If you’re tagging a large amount of media, you might find it more convenient to use the drag-and-drop methods described later on.

Quick tags are accessible from bins, spreadsheets, Viewers, and timelines for single or multiple selections.

1. Select the target clips or sequences.
2. Right-click a highlighted selection, go to Tags, and choose the required action, dependent on context.
   - For example, bin items only allow you to Tag Selection, whereas shots allow you to Tag Shot Selection, Tag Tracks, or Tag this Sequence

   Once you’ve selected the tag type, the Add Tag dialog displays.

   3. Select the icon to represent the tag using the Icon dropdown.
   4. Enter a Name and Note as required.
   5. Click Add to mark your selections with the chosen tag.
See Creating Custom Tags and Removing Tags for more information.
Tagging Using the Viewer

To apply a tag using the Viewer:
1. Click the Tags tab, or navigate to Window > Tags.
   The Tags panel displays.
2. Drag-and-drop the required tag from the Tags panel to the Viewer.

Depending on whether you’re looking at a clip or sequence, drop the tag on Tag this frame, Tag whole clip, or Tag whole sequence as required.

Tags applied to frames appear above the playback tools in the Viewer frame slider.

Tip: You can use Alt+Shift+left and right arrows to skip to the previous or next tag on the current clip. You can also reposition tags by dragging them along the Viewer timeline.

Tags applied to clips are displayed above the Viewer.
Tagging Shots

To apply a tag to a shot on the timeline:

1. Click the Tags tab, or navigate to Window > Tags. The Tags panel displays.
2. Drag-and-drop the required tag from the Tags panel to the timeline. Depending on where the tag is dropped, you'll mark a shot (or items if you make multiple selections) or a track.

Tags applied to shots appear on the right of the selected item(s) on the timeline.

Tags applied to tracks appear in the track header on the left of the timeline.
Tagging Shots |
Adding Notes to Tags

In some cases, a simple tag on a frame or clip may not contain all the information that you wish to pass on to the next stage of production. Adding notes to a tag can provide that extra detail.

⚠️ **Warning:** To delete a note, don’t click the - button because this refers to the tag. Instead, simply delete the notes in the window and click outside the note dialog.

1. Add notes to tags by clicking on the required tag and entering text or editing the metadata keys and values.
   The example shows a note and metadata key “Artist” added to a clip tag, but you can add notes to frame and timeline tags in the same way.

2. Click outside the dialog to save the note.

Nuke Studio allows you to “hide” tags using the Python API. Hidden tags are not displayed in the interface, unless you enable Show Hidden in the Tags popup, but the notes and metadata are still accessible.

Navigate to Help > Documentation > Hiero Python Developer’s Guide for more information on creating hidden tags.
Removing Tags

To remove a tag from a frame or shot, click the tag and then click $\square$.

You can remove all tags from a source clip or selection of clips by right-clicking your selections in the bin and choosing Tags > Clear Tags.

To remove a tag from a track or shot, click on a tag icon and select the required tag to remove.

Click $\square$ to remove your selection.
Creating Custom Tags

You may find that you require a specific tag or suite of tags that are not provided by default. Creating custom tags allows you to really control the organization of your media, and you can even create your own tag icons.

Note: Custom tags can only be created in the Tags panel.

To create a custom tag:
1. Click the Tags tab, or navigate to Window > Tags.
2. Select your project and navigate to Project > New Tag, or press Ctrl/Cmd+Y.
   The new tag is placed in the selected project.
3. Double-click the tag to open the Edit Tag dialog box.
4. Click the Icon dropdown menu to select an icon for the custom tag.

Tip: You can import your own image for the tag by selecting Custom to open the browser.

5. Enter a description for the tag in the Name field.
6. Click OK to save your changes.
Sharing Custom Tags

Custom tags can be shared between artists by saving them in a project in a shared network location. Shared tags are loaded at startup and appear in the Tags tab, below the standard tags that ship with Nuke Studio.

1. Create the required custom tags as described in Creating Custom Tags.
2. Save the .hrox project in a folder called Templates in a shared network location. For example: /SharedDisk/NukeStudio/Templates/SharedTags.hrox

Note: If your custom tags use custom icons, save the icon files in the same directory as the .hrox project.

3. Create a Python file containing the following lines, to direct Nuke Studio to the shared location:
   ```python
   import hiero.core
   hiero.core.addPluginPath("/SharedDisk/NukeStudio")
   ```
4. Save the .py file in the ~/.nuke/Python/Startup/ directory on all the machines running Nuke Studio that require access to the tags.

Note: You may need to create the ~/.nuke/Python/Startup/ path manually.

The location of the .nuke directory differs by platform. See Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts for more information.

5. Launch Nuke Studio and navigate to Window > Tags or switch to a workspace that contains the Tags tab.
   The custom tags are listed under the project name below the standard tags that ship with Nuke Studio.

Filtering and Flagging Media Using Tags

You can search for clips containing certain tags, for example, if you wanted to find all clips that you tagged as Approved.

There are two types of tag search you can perform: Filter and Flag. Select the desired search type by clicking the magnifier icon in the Project tab.

- **Filter** - displays all objects that contain the specified tag. This is the default search method.
- **Flag** - displays all objects and marks the items that don’t match the search tag.
Drag the required tag from the **Tags** panel into the search box and select the bin or bins you want to Filter or Flag.

**Tip:** If you have more than one search criteria, click the icons in the search box to display a brief description of the icons. Filters and flags persist until you change the search criteria or click the **x** icon in the search box.

The following examples show **Filtering** a bin to display only clips with the **Notes** tag applied and **Flagging** all clips that don’t have the **Notes** tag applied.
Viewing Metadata

Metadata is information that describes media content, separate from the clip itself, in the form of a table on the Metadata tab. Types of metadata include Duration, File Size, and the Path to the location of the source media.

Source Clip and Shot Metadata

To view metadata for a source clip, right-click the clip and select Open In > Metadata View, or press Alt+D.

To view metadata for a shot, select the Metadata tab in the timeline panel and click on the item to examine.

Tip: You may have to add the Metadata tab manually by clicking the icon and selecting Window > Metadata.
Filtering and Flagging Media Using Metadata

If searching your project using tags has not filtered your media effectively, you can search for clips containing certain metadata. For example, if you wanted to find all clips that had a particular resolution or frame rate.

To filter or flag using metadata:

1. Right-click the clip that contains the required metadata key and select Open In > Metadata View, or press Alt+D.
2. Drag-and-drop the required key from the Metadata panel to the bin view search box.
3. Use the metadata key as a filter or flag as described in Filtering and Flagging Media Using Tags.
Conforming Sequences

Conforming describes the process of matching up the footage from a shoot with the required edit decisions to create a meaningful timeline. Nuke Studio accepts sequences either from EDLs (edit decision lists), AAFs (advanced authoring format), or Final Cut Pro XML files from a specified directory structure containing the source media files. Nuke Studio attempts to conform the media, warning you if there are missing media.

Nuke Studio conforms EDLs into single tracks, and AAFs and XMLs into multi-track timelines. You can either conform into a brand new timeline, or into an existing timeline by adding new tracks. For example, when conforming multiple EDLs into the same timeline, you would add new tracks for each EDL sequence conformed.

![Conforming Sequences Interface](image)
Timeline Environment Project Settings

A good place to start work is by defining default Project Settings before importing sequences, particularly in the case of EDLs as they may not contain frame rate information. Project Settings only apply to the current project and override Preferences settings.

Note: You can modify Project Settings later on, for example, when you’re ingesting media.

To define Project Settings:

1. Navigate to Project > Edit Settings.
   The Project Settings dialog displays.
2. Click the General sub-menu to set the project Name.
3. Enter a Project Directory if required. This is the location of the .hrox project file and can be used as the root of the project if you want to use relative paths to source clips. See About Clips and Shots for more information.
   If you want this setting to apply to all new projects, use the Preferences > Project Defaults > General panel settings.

   Tip: Click Hrox Directory to automatically enter an expression that evaluates to the .hrox location.

4. Set the Poster Frame used by Project bin clips or use the default First frame. See Setting Poster Frames for more information.
5. Click the Sequence sub-menu to set the default Output Resolution, Frame Rate, and Start Timecode for new timelines in the current project, and set clip formatting when new clips are added to the timeline.
6. Click the Views sub-menu to set up multi-view or stereo projects. See Stereoscopic and Multi-View Projects for more information.
7. Click the Color Management sub-menu to manage the display and file colorspace for this project.
   See Color Management Settings for more information.
Tip: Use the options in Preferences > Project Defaults > Color Management to apply them to all new projects.

8. Click the RED Settings sub-menu to define the Default Video Decode Mode for new R3D files in the current project. This setting overrides the Preferences > Behaviors > File Handling > default red clip video decode mode control for existing projects. See Appendix A: Preferences for more information.

Note: Changing this setting does not change the decode setting for R3D media that has already been imported.

The dropdown contains a sliding resolution scale from FullPremium to SixteenthGood, but bear in mind that higher resolutions are slower to decode.

9. Lastly, click the Export/Roundtrip sub-menu to select:
   - External Media Track Name - sets the default name of the track created when exported media is brought back into Nuke Studio.
   - Export Directory - sets whether the Project Directory, if specified, or a custom directory is used for exports. If no Project Directory is specified, the project root in the Export dialog is used.
   
   If you want this setting to apply to all new projects, use the Preferences > Project Defaults > General panel settings.

   - Custom Export Directory - when Export Directory is set to custom, enter the required custom export directory.
   - Shot Preset - sets the default preset to use when you select Create Comp from the timeline.
   
   See Create Comp for more information.

Color Management Settings

Nuke Studio uses OpenColorIO for color management. All of the colorspace options and roles (aliases to colorspace) that you can set in Nuke Studio. There are also default options, which change depending on what file type you are working with. When the default option is selected, the colorspace that Nuke Studio has set for it is listed in brackets.

Depending on the OCIO config file that you are working with, there are a number of colorspace options and roles (aliases to colorspace) that you can set in Nuke Studio. There are also default options, which change depending on what file type you are working with. When the default option is selected, the colorspace that Nuke Studio has set for it is listed in brackets.
Tip: Use the options in Preferences > Project Defaults > Color Management to apply them to all new projects.

1. Set the OpenColorIO Config you want to use for this project.
   Nuke Studio ships with a number of default configurations, but you can:
   • use a custom OCIO config file by selecting custom from the OpenColorIO Config dropdown and then entering the file path, or
   • add your own config to your .nuke file. See Adding OCIO Configurations for more information. Changing the configuration updates the Default Color Transforms accordingly. If the selected configuration is invalid for certain transforms, a warning displays. For example, if you choose the shipped iff configuration, the 8-bit and 16-bit transforms are not compatible.

   ![Warning message]

   In this case, the non-compatible transforms are set to the raw colorspace.

2. The Working Space transform determines what colorspace files should be converted to, on import, and from, during export - it's the colorspace used by the Timeline environment under the hood.
3. You can use Default Color Transforms dropdown menus to override how clips in the Viewer, thumbnails, and so on are converted to and from the Working Space.

4. The Nuke Script Project Settings dropdown determines whether Nuke Studio uses the LUTs read from the configuration specified or the Nuke native LUTs during export. Selecting OCIO makes the relevant OCIO LUTs available to the Read and Write nodes in scripts on a per project basis.

   All configurations except nuke-default automatically switch this control to OCIO.

When the Nuke is selected, Reads and Writes work the same as in previous versions of Nuke, with no integrated OCIO transforms. When OCIO is selected:

• Reads and Writes use OCIO transforms, with no Nuke built-in LUTs applied to the image.
• Read and Write colorspace controls are populated with the list of colorspaces defined in your currently selected OCIO config.
• The default LUT settings dropdowns are also populated with the list of colorspaces or display transforms defined in your OCIO config. The default value for each menu match the defaults in a Nuke Studio project with the same config. These defaults can be overridden using Python callbacks. See the following path for the default implementation that ships with Nuke:

   <install_dir>/plugins/nuke/colordspaces.py

• The working space dropdown allows you to change the colorspace that Nuke uses internally for its image processing. This automatically sets the in colorspace of Write nodes and Viewer Processes, and the out colorspace for Read nodes. This defaults to the scene linear role defined in your OCIO config.
• Nuke Studio-created comps no longer contain automatically injected OCIOColorspace nodes. Instead, OCIO Color Management is automatically set in the comp’s Project Settings, and the correct OCIO colorspace is set directly into the Read and Write nodes.

Adding OCIO Configurations

You can add your own OCIO configurations to Nuke Studio as they become available, such as new versions of ACES. You can also add legacy configs for backward compatibility.

1. Navigate to the location of your .nuke file as shown by platform. You may have to create a .nuke folder if it doesn’t exist.
   • Linux: /users/login name/.nuke
• Mac: /Users/login name/.nuke
• Windows: ~\nuke

**Note:** On Windows, the `.nuked` folder can be found under the directory pointed to by the HOME environment variable. If this variable is not set (which is common), the `.nuked` directory is under the folder specified by the USERPROFILE environment variable - which is generally of the form `drive letter:~\Documents and Settings\login name\` or `drive letter:\Users\login name\`

To find out if the HOME and USERPROFILE environment variables are set and where they are pointing to, enter `%HOME%` or `%USERPROFILE%` into the address bar in Windows Explorer. If the environment variable is set, the folder it’s pointing to is opened.

2. Recreate the following structure within your `.nuked` folder: `~/plugins/OCIOConfigs/configs/<config name>`
3. Copy the contents of the config into the `config name` named folder. There should be a `luts` folder and `.ocio` file at the bare minimum.
4. If Nuke Studio is already running, relaunch the application to apply the change.
5. You can now select your configuration from the Project Settings > Color Management > OpenColorIO Config dropdown.
Importing Sequences

Nuke Studio allows you to import your EDL, XML, or AAF sequences in one of two ways, depending on your preferences. Either:

• Navigate to File > Import EDL/XML/AAF, use the browser to locate the EDL, XML, or AAF, and then select the file and click Open to import the sequence,

OR

• Drag-and-drop the EDL, XML, or AAF files directly from a file browser into the interface.

If you’re importing an EDL, bear in mind that there is no guaranteed frame rate information included in the file, so an Import Options dialog displays.

1. Select the correct frame rate and use the following check boxes, if required:
   • Drop Frame - when enabled, the EDL is assumed to contain drop file information. See Timeline Playback Tools for more information.
   • Assume differences in source/destination durations indicate a retime - when enabled, any disparity between the source clip (Src) and shot (Dst) duration is treated as a retime.

2. Click OK to import.

XMLs and AAFs imported into Nuke Studio support transform, crop, and retime edit decisions implemented in third-party applications, such as Adobe Premiere, Apple Final Cut Pro, and Avid Media Composer. The information in the .xml or .aaf is interpreted using soft effects, such as Transform and Crop. Non-linear retimes are represented by TimeWarp effects. Constant linear retimes are handled in the same way as in previous versions of Nuke Studio. See Notes on AAF Sequences for more information.
**Note:** Non-linear animation curves from .xml may not appear as expected when imported. As a result, you may need to adjust the handles on curves to match footage between keyframes in the Curve Editor or Dope Sheet.

See [Animating Parameters](#) for more information.

Additionally, Premiere Pro .xml exports only support constant, linear retimes. As a result, retimed shots on the Nuke Studio timeline may not match those on the Premier Pro timeline, because certain non-linear retime data is not written into the exported .xml file.

After importing the EDL, AAF, or XML the **Conforming** workspace displays and the spreadsheet and timeline are populated with offline clips - media with an unknown location.

**Note:** The **Event** column represents the clip's position on the timeline, not its event number from the edit.

![Spreadsheet and timeline](image)

Notice that clicking entries in the spreadsheet highlights the corresponding shots on the timeline?
The spreadsheet, timeline, and Viewer are linked together when viewing sequences. If suitable screen real estate exists within the current workspace, double-clicking a sequence forces the associated panel to open automatically. If you want to close a single panel in a linked group, hold the Alt modifier while closing the linked panel, otherwise all panels in the group are closed.

**Note:** If you imported an XML sequence, you may find that Nuke Studio has automatically matched media for you.

Any transform, crop, or retime edit decisions from third-party software .xml and .aaf files are represented using soft effects. These effects are imported along with the shot to which they're associated.
Notes on AAF Sequences

Avid Media Composer supports retimes using curves that map **frame to frame** or **frame to speed**. Nuke Studio handles the import differently depending on the retime method.

- **frame to frame** - describes the retiming in relative terms, such as 'at frame 100 in the output clip, display frame 50 of the source clip'.
- **frame to speed** - describes the retiming in terms of overall output duration. For example, half speed doubles the duration of the clip.

Nuke Studio’s TimeWarp effect only supports **frame to frame** mapping, which means that **frame to speed** retimes from .aaf files requires some curve-fitting to describe the required retime. As a result, the keyframes generated in Nuke Studio don't match those in Avid, but the resulting curve should match the original very closely.

**Note:** Nuke Studio currently only supports Fixed Keyframes from Avid Media Composer.

**Tip:** If you need to adjust the handles on curves, see [Animating Parameters](#) for more information.

Nuke Studio’s TimeWarp effect supports the following Spline types when importing .aaf files:

- Shelf
- Linear
- Spline
• Bezier
Conforming Sequences

Once your EDL, AAF, or XML sequences are imported, it’s time to begin the conform process to match the offline shots in your spreadsheet with the source clips on disk. You can conform sequences by searching on disk or by pre-ingesting the required clips into Nuke Studio.

**Note:** Projects containing large amounts of movie files (for example `.r3d` and `.mov`) may exceed the number of available file handles per process, causing problems opening new files or projects and exporting. You can increase the default limit of 1024 by entering the following command from the terminal, then running the application from the same session:

```
ulimit -Sn 2048
```
Conforming Using a Browser

To conform a sequence using a browser:

1. After importing a sequence, click **Match Media** on the spreadsheet and use the browser to locate the source folder containing the correct media.

   **Note:** **Match Media** can also be used on selected events in the Spreadsheet view.

2. Click **Open** to display the **Conform Options** dialog.

Nuke Studio uses a set of conform **Rules** and file name **Patterns** to match candidate media files on disk to the events, or shots, in a sequence:

- **Rules** - sets the offline media properties to match to the corresponding spreadsheet entry during conform.
  
  Rules that rely on information that doesn't exist in the event or candidate clip are ignored, and some rules compound others to identify a better match.

<table>
<thead>
<tr>
<th>Rule</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Umid</td>
<td>Match a file's unique material ID (UMID) – that is written into the file's metadata on creation – to the candidate media's UMID. If either, or both, lack a UMID this rule is ignored.</td>
</tr>
<tr>
<td>RedTapeName</td>
<td>Match a RED-style camera reel name from the event to the candidate media</td>
</tr>
</tbody>
</table>
### Rule Description

<table>
<thead>
<tr>
<th>Rule</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>name.</td>
<td></td>
</tr>
<tr>
<td>RedName</td>
<td>Look for a RED-style camera file name in the event that matches the candidate media name.</td>
</tr>
<tr>
<td>ReelName</td>
<td>Look for the event's reel name in the candidate's media name.</td>
</tr>
<tr>
<td>FullPath</td>
<td>Match the event's entire filepath to the candidate media's entire filepath.</td>
</tr>
<tr>
<td></td>
<td>The Event is the first field in the Spreadsheet view, the order in which shots appear on the timeline. Candidate media is the media that Nuke Studio is testing the conform rules against.</td>
</tr>
<tr>
<td>FileName</td>
<td>Match only the event's file name (no path) to the candidate media's file name.</td>
</tr>
<tr>
<td>FileHead</td>
<td>Match the event's file name head (no path, file extension, or padding) to the candidate media's file name.</td>
</tr>
<tr>
<td>PartialName</td>
<td>Look for the event's name in the candidate media's name and vice versa.</td>
</tr>
<tr>
<td>FolderName</td>
<td>Look for the event's name in the filepath of the candidate media.</td>
</tr>
</tbody>
</table>

All rules are enabled by default, but you may occasionally need to disable rules if they cause incorrect matches between a particular edit and set of source clips.

**Tip:** Use the Select/Deselect All buttons to quickly enable or disable rules.

- **Patterns** - sets the inclusion and exclusion parameters during the conform, separated by spaces. For example, *mov *dpx would only include or exclude .mov and .dpx files.

You could also conform by name, such as BR_Shot*, which would only include or exclude source clip names starting with that string.

**Tip:** It's always a good idea to be as specific as possible with search locations during conforms, but if the need arises, conform Rules and Patterns can save time.

3. Enable **Accept best timecode match**... to use the nearest source timecode match to conform the event, if no rules are matched.
4. When **ignore clips with non-overlapping timecodes** is enabled, any potentially matching source clip whose timecode doesn't overlap the shot in question at all is ignored.
Disabling **Ignore clips with non-overlapping timecodes** causes Nuke Studio to fall back to the other selected conform rules, even if the timecodes don't overlap.

5. Check **Conform shots that already have media** if you want to update all timeline shots. By default, the application doesn't try to conform events that are not offline.

6. When **Split sequences** is enabled, any non-contiguous file sequences found by the conform are split into separate clips, in the same way as when the **split seq** option is enabled in the file browser.

7. Click **OK** to begin the conform process. Nuke Studio attempts to conform the edits with the selected media. A dialog box informs you of the success rate once the conform is complete.

Successfully matched media is placed in a new **Conform** bin in the project.
**Note:** You can display the conform Rules matched for each spreadsheet object by hovering the cursor over the required entry.
Conforming with Pre-ingested Media

To conform with pre-ingested media:

1. If your source media has been ingested, you can drag-and-drop media from the bin view onto the Match Media button.
   
   See Ingesting Media for information on getting media into Nuke Studio.

2. Follow the Conform Options instructions described previously to complete the conform process.
   
   If you want to conform a single entry in the spreadsheet, drag-and-drop the media from the bin view onto the required entry in the spreadsheet.
Conforming individual, pre-ingested media doesn't require **Conform Options** because Nuke Studio already knows the exact location of the media and trusts your decision to replace a shot.
About the Media Spreadsheet

All events in a sequence are displayed in an easy to read format in the spreadsheet including status, the track it resides on, length, and the source file location.

After conforming, you can use the spreadsheet to locate source clips or replace shots in the timeline, as well as massage timecodes if they are invalid.

The media spreadsheet displays each entry's current media state:

- ✓ - the media was successfully conformed and its timecode is correct.
- ▢ - the media was successfully conformed, but the timecode is currently incorrect.
- ❌ - the media could not be conformed.

**Note:** Any source or destination field highlighted in yellow indicates that the entry has been rounded down for display purposes.

See Managing Timelines for more information on importing tracks and reference media.

Sorting and Custom Columns

The spreadsheet can be organized in much the same way as accounting spreadsheets:

- Right-click the column headers to display the list of default columns available. Enable or disable each column using the checkboxes.
- Click the required column to sort the spreadsheet in ascending or descending order, as indicated by the arrow in the column header.
- Drag-and-drop column headers to reorder the spreadsheet as required.
- Add custom columns, such as Tags, using the Python API. See Help > Documentation for more information on the Python API.

**Tip:** Nuke Studio's Project panel search functionality extends to the spreadsheet, allowing you to enter strings and apply searches on all or partial matches with the option to include metadata searches. Nuke Studio searches for items that match any of the input string and displays only those items by default. See Sorting and Searching Media for more information.
Spreadsheet Controls

There are also a number of controls, accessed by clicking the cog icon, that determine the spreadsheet's appearance and behavior.

- **Select Matching Name** - when enabled, selecting an item in the spreadsheet highlights all items with the same name.
- **Select Linked** - when enabled, selecting an item in the spreadsheet highlights other items linked to it, such as audio tracks ingested with video tracks.
- **Display Speed As** - sets the unit used in the Speed column of the spreadsheet. Select either fps (frames per second) or % (the percentage of the media frame rate).
- **Retime method** - sets the type of Speed retime applied on the timeline.
- **Time Edit Behaviors** - sets how source and destination In, Out, and Duration are calculated.

See [Retiming Shots](#) for more information on retime methods and [Timeline Editing Tools](#) for source/destination calculations.

You can locate, display, reconnect, or rename shots directly from the spreadsheet.

- Hold **Alt** and click an entry to move the playhead to the shot’s In point on the timeline.
- Hold **Alt** and double-click an entry to move the playhead to the shot’s In point on the timeline and zoom to fit the timeline view.
- Right-click a spreadsheet entry and select the required option:
• **Open In** - the associated source clip opens in the selected location, such as a Viewer.
• **Project View** - the associated source clip is highlighted in the bin view.
• **Reconnect Media** - attempt to reconnect the media from a specified location on disk, such as when the source was originally on a drive that is no longer connected.
• **Replace Clip** - replaces the selected entry with a specified source clip. Nuke Studio assumes that any source clip you choose is acceptable, regardless of timecode.
• **Delete** - deletes the selected entries from the spreadsheet and timeline.
• **Tags** - allows you to add tags to your selection directly from the spreadsheet view. See Using Quick Tags for more information.
• **Localize** - allows you to control the localization of clips, tracks, and sequences from the spreadsheet. See Localizing Media for more information.
• **Effects** - provides access to Create Comp and soft effects directly from the spreadsheet. See Create Comp and Soft Effects for more information.
Adjusting Timecodes

You can easily adjust single or multiple event timecodes:

1. Select the invalid entry or entries in the spreadsheet.
2. Double-click in the Src In column.
3. Adjust the timecode as required. You can enter absolute or relative timecode values:
   - **Absolute** - absolute timecodes contain eight digits and specify the new timecode for the event, regardless of the current timecode.

<table>
<thead>
<tr>
<th>Example</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>01:05:43:21</td>
<td>Sets the timecode at 1 hour, 05 minutes, 43 seconds, and 21 frames</td>
</tr>
<tr>
<td>01054321</td>
<td></td>
</tr>
</tbody>
</table>

- **Relative** - uses + and - values to alter the timecode relative to its current value. You can also use h, m, and s to denote hours, minutes, and seconds.

<table>
<thead>
<tr>
<th>Current Position</th>
<th>Example</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>01:05:43:21</td>
<td>+1h</td>
<td>02:05:43:21</td>
</tr>
<tr>
<td></td>
<td>-110</td>
<td>01:05:42:11</td>
</tr>
<tr>
<td></td>
<td>+10000</td>
<td>01:06:43:21</td>
</tr>
<tr>
<td></td>
<td>-6m</td>
<td>00:59:43:21</td>
</tr>
</tbody>
</table>

The media changes state to 🎥.

**Note:** Timelines start at 01:00:00:00 by default, but you can change this to any value using the **Sequence** panel.
If you're not sure what the timecode should be, you can:

- Hover the mouse over the target entry in the spreadsheet to view a timecode tooltip.

OR

- Examine the source clip's metadata and calculate the correct **Src In**:
  1. Right-click the required entry and select **Project View**.
  2. Right-click the clip in the bin and select **Open In > Metadata View**, or press **Alt+D**.
     The selected clip metadata is displayed in a floating pane.
Renaming Shots on the Timeline

Once you've conformed your edit, you may want to rename shots on the timeline sequentially for clarity.

To rename shots:
1. Select the shots to rename using the timeline or spreadsheet view.
2. Right-click on the timeline and select Editorial > Rename Shots.

Tip: You can also navigate to Timeline > Rename Shots or use the Alt+Shift+/ keyboard shortcut.

The Rename Shots dialog displays.

3. Select the rename type from the dropdown:
   - Simple Rename - all shots are replaced by the New Name specified.
   - Find and Replace - a simple find and replace shot name. All selected shots containing the specified Find pattern are substituted with the Replace pattern.
   - Sequential Rename - rename shots sequentially using the Pattern, Start #, and Increment fields.
   - Match Sequence - allows you to select a sequence to copy shot names from, providing that they use the same shots. For example, renaming shots on a 30 second timeline to mirror the shot names from a 60 second timeline.

Note: You can only use sequences that reside in the same project and shots that have overlapping frame ranges.

- Clip Name - all shot names are replaced by the name of the source clip they reference. This option can be used to revert previous rename operations.
• **Change Case** - the case of all shot names is changed, as specified by the **Case** dropdown. For example, selecting **Title Case** capitalizes the first character of each word.

4. Rename operations also accept token substitutions. The following tokens are recognized:

<table>
<thead>
<tr>
<th>Token</th>
<th>Resolves to</th>
</tr>
</thead>
<tbody>
<tr>
<td>{clip}</td>
<td>The name of the source clip referenced by the target shot.</td>
</tr>
<tr>
<td>{event}</td>
<td>The EDL event number associated with the target shot.</td>
</tr>
<tr>
<td>{filename}</td>
<td>The file name of the shot's source.</td>
</tr>
<tr>
<td>{fps}</td>
<td>The frame rate of the sequence containing the rename target.</td>
</tr>
<tr>
<td>{sequence}</td>
<td>The name of the sequence containing the rename target.</td>
</tr>
<tr>
<td>{shot}</td>
<td>The name of the shot.</td>
</tr>
<tr>
<td>{track}</td>
<td>The name of the track containing the rename target.</td>
</tr>
</tbody>
</table>

5. Enable **Include Clips From Audio Tracks** to rename audio shots as well as video shots.

6. Click **Rename**.

   The selected shots are renamed as specified.
Saving and Loading Projects

You can save your work in a project using the .hrox file extension. Projects can contain .nk scripts in the form of shots added by using Create Comp. See Create Comp for more information.

If you quit the application without saving, you'll be prompted to save or discard your changes:

![Save changes to 'Untitled 2' before closing?](image)

Click the required button or press D for Don’t Save or S to Save.

**Note:** If you have a .nk script open in the same session, a second prompt is displayed so you can save your script as well.

To save a project:
1. Navigate to File > Save Project or Save Project As...
   OR
   Use the Ctrl/Cmd+S or Shift+Ctrl/Cmd+S keyboard shortcuts respectively.
   The Save Project dialog box displays.
2. Browse to the save location and enter a name for the project.
3. Click Save.
   Your project is saved to the location specified and appends the .hrox file extension automatically.

To load a project:
1. Navigate to File > Open Project.

**Tip:** If you need to open a project that you worked on recently, you can select Open Recent Project to quickly locate your work.

OR
Use the Ctrl/Cmd+O keyboard shortcut.
The Open Project dialog box displays.
2. Browse to the file location and click Open.
   Your project opens and populates the necessary panel automatically.
Autosaved Projects

The autosave function creates a temporary project save file at 5 minute intervals, but you can adjust **force project autosave after** in the **Preferences > General** dialog. See **Appendix A: Preferences** for more information.

At startup, the application scans for autosaved projects and displays a prompt if autosaves exist.

Click **Yes** to load the autosave or **No** to ignore and delete it.

Opening a project also uses the autosave functionality. If the autosave is more recent than the saved project file, a prompt displays:

Click **Yes** to load the autosave file or **No** to load the original project file.

**Note:** Clicking **No** does not delete the autosaved project in this case.
Managing Timelines

Timelines contain video and audio shots that reference the source clips in your project. Once the conform process is complete, the timeline displays your clips in context and enables you make finer edits. Timelines can contain any number of video sequences and audio tracks with each track containing shots that reference the source clips in your project - making changes to shots in the timeline does not affect the original source clip.

Nuke Studio also features real-time soft effects on the timeline and the ability to add shots containing .nk scripts. See Soft Effects and Create Comp for more information.

Note: Conformed EDLs only support one video sequence. If you've created multiple EDLs from the same edit, you can add each one into the timeline using the right-click New Track > New Track(s) from EDL/XML option or the Import Track button in the spreadsheet tab. See Adding Tracks to the Timeline.

• Video Toggles - quickly turn off and on video tracks during playback. Hold Alt and click to solo the selected track. You can also enable and disable track blending and masking. See Blending Tracks on the Timeline for more information.

• Disk Caching Controls - click and hold to display the disk caching options for the current timeline. See Caching Frames in the Disk Cache for more information.
• **Timecode** - displays the timecode or frame number depending on the **Time Display** mode selected. You can adjust the scale using the Scale Slider or by using the mouse wheel.

• **Playhead Position** - displays the playhead location synchronized with the contents of the Viewer.

• **Video Tracks** - contain all video sequences for the current timeline.

• **Audio Tracks** - contain all the audio clips for the current timeline.

• **Audio Toggles** - quickly mute audio or set the track output during playback to left, right, or mono.

• **Track Lock** - secure the selected track to disable all editing tools.

**Tip:** Selecting tracks while holding Ctrl/Cmd allows tools to affect multiple tracks at once, such as locking, disabling, and resizing tracks.

Video tracks in multi-track timelines are read from the highest number track downward, for example Video 3, Video 2, Video 1. As a result, if video is present on track 3, video on track 2 in the same time slice is obscured.

In this example, although the playhead crosses clips on two video tracks, only the clip in Video 3 is displayed in the Viewer.

Audio tracks, on the other hand, play back simultaneously - all the audio tracks crossed by the playhead in the example play back together, creating a complete audio backing for the video.
Tip: Enabling Preferences > Panels > Timeline > show frame end marker draws an extra line on the timeline to the right of the playhead, indicating the end of the current frame.
Adding Tracks to the Timeline

You can add empty tracks to existing timelines or import other EDL, AAF, or XML edits - effectively another sequence within the timeline.

To import EDL, AAF, or XML edits:

1. Select the required sequence in the project bin, right-click, and select Import > New Track(s) from EDL/XML,
   OR
2. Click Import Track in the spreadsheet tab.

3. Use the browser to locate the EDL, AAF, or XML files, select the file(s) and click Open to import the sequence.

   **Note:** If you're importing EDLs, bear in mind that there is no guaranteed frame rate information included in the file. Select the correct frame rate then click OK in the dialog supplied.

4. Conform the new track as described in Conforming Sequences.

To add new tracks:

- Drag-and-drop a clip above or below existing tracks as shown,
OR

- Right-click in the timeline and select **New Track > New Video Track** or **New Audio Track**.
**Note:** You can also collapse and expand existing tracks using the right-click **Editorial** menu, and resize the track header to accommodate longer track names.
Adding Clips to the Timeline

The timeline allows you to add clips by simple drag-and-drop from either the Viewer or bins. Using the Viewer restricts you to a single clip, the current clip, but you can drag as many clips as you like from bins.

**Tip:** You can create a new sequence by dragging a clip to an empty timeline pane.

New timelines pick up their frame rate from the *Project > Edit Settings > Sequence* sub-menu by default. Dropping a clip with a different frame rate on a new timeline displays a warning:

However, if the timeline is already populated and the clip you're adding doesn't have the same frame rate as the timeline, you're prompted to choose whether the clip's duration or frame count is retained.

Take care not to overwrite existing shots - the most recent clip overlays any existing shot. To avoid this, do one of the following:

- Move the playhead to the target area of the timeline in the record Viewer, load the required clip in a source Viewer, and then use **Insert** (N) or **Overwrite** (M) to place the clip into the timeline at the playhead position on the lowest, unlocked track available.

**Note:** You can only **Insert** or **Overwrite** using clips from the current project.
See **Insert, Overwrite, and 3-Point Editing** for more information on source/record editing.

- Use the **Multi** or **Move/Trim** tools to make space for the new clip and then drag-and-drop it in to the space (see **Using the Move/Trim Tool** for more information).

- Drag-and-drop the new clip at the end of the sequence, then using the **Multi** or **Move/Trim** tools, drag the new clip to an existing transition, hold down the **Alt** modifier, and drop the clip to **Ripple** all other shots down the timeline.
**Tip:** Enabling Preferences > Panels > Timeline > show frame end marker draws an extra line on the timeline to the right of the playhead, indicating the end of the current frame.
Audio and the Timeline

Audio tracks on the timeline are handled in much the same way as video tracks. By default, linked audio and video tracks are edited at the same time, but you can lock either track and move them independently or hold Alt to select a single track, if required.

**Note:** Although you can import and edit multi-channel audio, during playback audio is mixed to 48 KHz stereo output.

**Note:** Nuke Studio does not currently support any QuickTime audio on Linux. Support for audio on Linux is scheduled for a later release.

When both Viewer inputs contain clips, the audio output is set by the orange marker in the color picker information bar, displayed by clicking 🎤. In the following example, input A is providing the audio output:

![Audio Pickup](image)

The volume slider in the upper-right corner of the Viewer controls the output level for that Viewer only.

![Volume Control](image)

Audio output for shots can be toggled between left, right, and mono using the audio toggles in the track header. Click on the icon to cycle between outputs:
You can also control audio on a per track and per shot basis. Audio track headers and shots have independent volume controls in the timeline and Properties tab.

- **Track headers** - click and hold the mute icon on the header to display the volume slider.

- **Shots** - select an audio shot and click on the Properties tab to display the Volume control.

**Tip:** You can control the volume on multiple shots simultaneously by holding Ctrl/Cmd and selecting the required items before adjusting the volume slider.
The **Preferences > Panels > Viewer (Sequence)** sub-menu contains audio controls allowing you to control the volume level for all new Viewers. See [Appendix A: Preferences](#) for more information.

**Note:** If the frame rate drops too low, audio is automatically muted and the speaker changes to the no audio playback icon.
WAV Shots

Audio can be recorded at the same time as shooting the video or it can be unrelated to the shoot, for example sound effects or music. You can add .wav clips to the timeline in two ways:

- **Drag-and-drop** - drag your .wav clip to a timeline audio track and drop it in to place.
- Navigate to **File > Import File(s)** or **Import Folder(s)**.

**Tip:** You can also use the Ctrl/Cmd+I and Ctrl/Cmd+Shift+I keyboard shortcuts.

Use the **Timeline Editing Tools** to move the clip into place and set its output.
Displaying Audio Waveforms

Visualizing an audio waveform helps synchronization with video events, and Nuke Studio displays waveforms in the timeline by default.

Audio shots are manipulated in the same way as video shots, so using waveforms in conjunction with the Timeline Editing Tools on page 1389 enables you to quickly synchronize audio and video events. Audio shots also support **Fade In**, **Fade Out**, and **Dissolve** transitions in the same way as video. See **Adding Transitions** for more information.

You can toggle the waveform display on and off by right-clicking in the timeline and selecting **View > Audio Waveforms**. You can also control how the waveform appears, when enabled. Open the Preferences and navigate to **Panels > Timeline** to toggle between full and **half waveforms**.

![Waveform Display](image)

Displaying waveforms in audio-heavy projects can cause significant slow down on the timeline, so Nuke Studio includes a preference to limit how much system memory is available for waveform display. In the Preferences, navigate to **Performance > Caching > Audio Waveforms** and set the waveform memory control to the required amount.
Audio Scrubbing

Nuke Studio’s timeline supports audio scrubbing, allowing you synchronize audio and video more easily while scrubbing the playhead. Audio scrubbing is enabled by default, but you can disable it by right-clicking in the timeline tab and clicking Audio > Audio Scrubbing or by pressing Ctrl/Cmd+Alt+S.

Audio shots cache temporarily to increase responsiveness during scrubbing. If you need to clear the audio cache, navigate to Cache > Clear Audio Cache.

Note: Audio scrubbing is not currently available through monitor output cards. Audio scrubbing is only supported through internal audio output devices.
Synchronizing Audio and Video

Nuke Studio allows you to massage the synchronization between audio and video tracks using audio latency adjustment during playback in the Viewer, or by a default amount in the Preferences > Panels > Viewer (Sequence) sub-menu.

**Note:** Latency adjustments can take a few seconds to affect the audio track.

1. Mark a portion of the timeline containing the target audio and video shots using In and Out markers.
2. Press 🎥 or use the L keyboard shortcut to begin playback.
3. Click the Viewer settings icon and increment the latency using the controls in the popup.
4. Adjust the latency until the tracks are in sync.
PulseAudio on Linux

PulseAudio on Linux distributions has been linked with fluctuating frame rates due to the latency when retrieving audio samples. If detects that your setup is running PulseAudio alongside the application, a warning message displays.

Stopping PulseAudio

You can disable PulseAudio for the current user or all users on a machine. To stop the daemon, do the following:

1. Open the ~/.pulse/client.conf file to disable PulseAudio for the current user, OR
   Open the /etc/pulse/client.conf file to disable PulseAudio for all users.
2. Set the following attribute and ensure the line is not commented out:
   autospawn = no
3. Call pulseaudio --kill to end the PulseAudio process.
4. Call ps -e | grep pulse to check the process stopped correctly.

Note: PulseAudio restarts automatically when you restart your machine, but you can prevent this by navigating to System > Preferences > Startup Applications and disabling the PulseAudio Sound System.

Note: Ending PulseAudio while other applications are running may disable audio output. Stop and start the application to re-enable audio output. Additionally, the desktop audio slider may be removed.
Restarting PulseAudio

To start the PulseAudio daemon, do the following:
1. Open the ~/.pulse/client.conf file to enable PulseAudio for the current user,
   OR
   Open the /etc/pulse/client.conf file to enable PulseAudio for all users.
2. Set the following attribute and ensure the line is not commented out:
   
   autopawn = yes

3. Call pulseaudio --start to start the PulseAudio daemon.
4. Call ps -e | grep pulse to check the process started correctly.
Using Reference Media

Importing a reference version of your timeline enables you to compare your current timeline against the reference media to avoid issues with continuity, missed frames, and so on.

To import reference media, click Set Reference Media and use the browser to locate the required file.

The reference media is automatically imported into Reference tracks, pushing existing tracks outward, and marked with the Reference Media tag.

After importing the reference media, use the show/hide icon or A/B input tools to compare the current timeline against the reference clip. See Comparing Media for more information.
Comparing Media

The Viewer A/B tools allow you to quickly compare media using the two Viewer input buffers. Select a clip, sequence, shot or track and press 1 or 2 to place your selection in the Viewer input buffers. You can also drag-and-drop items into the input buffers using the Viewer hotspots.

**Note:** The Viewer currently treats all alpha channels as premultiplied, which can result in the Viewer background being “added” to the image. If you're working with un-premultiplied images, set the Viewer background to **Black**. See Appendix A: Preferences for more information.

When the Viewer input buffers contain sequences, the A and B dropdowns control what is displayed in the Viewer using track names and tags. Selecting a track or tag from the dropdown displays the selected media in the Viewer.

![Viewer interface with A and B dropdowns](image)

Use the **wipe**, **stack**, **horizontal**, and **vertical** modes to control how the buffers are displayed in the Viewer.

**Note:** If you're working in a multi-view project, using stereo footage for example, you can set which view is output in the A and B buffers using the Views buttons over the Viewer. See Displaying Views in the Viewer for more information.
The **wipe** and **stack** modes also allow you to blend the two buffers together, and in the case of **wipe** mode, provides a handle in the Viewer to quickly wipe between the two inputs.

![Wipe stack modes](image)

The color picker overlay displays a description of the contents of the A and B inputs, or **No Clip** when there is no clip at the playhead, for instance, when there is a gap in a timeline or if a track is disabled.

![Color picker overlay](image)

The orange triangle in the overlay denotes the clip currently supplying audio and timecode information in the Viewer.
Caching Frames in the Playback Cache

The playback cache places frames in RAM for rapid retrieval during playback, rather than creating files locally as with Caching Frames in the Disk Cache or Localizing Media.

The white bar under the Viewer represents the contents of the playback cache, a full bar indicating that the entire clip or timeline is currently accessible from RAM, optimizing playback. You can:

• Temporarily disable caching using the pause button above the Viewer, or use the P keyboard shortcut.

Clicking pause again, resumes caching from the playhead position.

• Flush the cache completely by navigating to Cache > Clear Playback Cache. Caching is automatically paused after flushing, but clicking the pause button resumes caching from the playhead position.

There are also a number of Preferences that affect how much RAM is available and when caching should occur. To set the caching behavior:

1. Navigate to Nuke > Preferences (OS X) or Edit > Preferences (Linux and Windows),
   OR
   Use the Preferences keyboard shortcut Shift+S.
2. Select Performance > Caching and set the total RAM cache available using the playback cache size field.

   **Note:** You can't set this to a value higher than 80% of the memory available (rounded down to the nearest half-GB). For example, if you have 6 GB of memory available, the maximum cache size available 4.5 GB.

3. You can enable free timeline playback RAM to discard any frames cached to RAM (the white bar in the timeline Viewer) when you switch to the Node Graph within Nuke Studio, freeing the RAM for use in Nuke.

   **Note:** When you switch back to the timeline, the cached files are re-cached, which can be time consuming.
4. Enable **pause caching when the application goes to the background** to pause playback caching when the application loses focus.
   When you click back into Nuke, caching picks up from where it stopped.

5. Enable **clear cache when the application goes to the background** to flush the playback cache when the application loses focus.
   When you click back into Nuke, caching starts again from the position of the playhead.
Caching Frames in the Disk Cache

Timeline Disk Caching provides reliable playback for more complex timelines by rendering frames to disk using the GPU. The cache provides persistent frames per edit in the timeline that only needs updating for full changes on the edit, such as adding a soft effect. For editorial changes, only the new frames need to be cached.

**Note:** Frames are always cached at the sequence resolution, regardless of source clip format and Viewer proxy settings.

You can cache whole sequences, selections of clip ranges, and frame ranges specified using In and Out points. Files in the disk cache are frames identical to what you see rendered in the timeline Viewer, written into .exr sequences, and saved in NUKE_TEMP_DIR/TimelineCache by default.

**Tip:** You can find the location of Nuke’s general cache directory from within Nuke by hitting X on your keyboard, when the focus is on the Node Graph, and then running the following Tcl command:

getenv NUKE_TEMP_DIR

Cached frames are represented in the timeline by the state of the timeline cache icon:

- None of the frames in the current timeline are cached.
- The current timeline is partially cached.
- The current timeline is fully cached.

Cached frames are represented in the Viewer with an orange bar, under the RAM cache bar, which is white by default.
See Caching Sequence Ranges, Caching Selected Shot Ranges, and Caching In/Out Ranges for more information.

You can set the cache directory location, size of the timeline cache, and type of EXR compression used by:

- Clicking and holding the timeline cache icon and selecting Cache Settings, or
- Opening the Preferences and navigating to Performance > Caching.
Caching Sequence Ranges

Caching a sequence caches exactly what you see when play through the entire timeline. As the playhead walks the timeline, all tracks are read from the highest track downwards, so what you see in the Viewer is what Hiero caches to disk.

Hiero can only cache one sequence at a time, either from the timeline itself, the Cache menu, or the Project bin:

**Tip:** You can pause caching at any time by navigating to Cache > Disk Cache > Pause in the menu bar.

- Select the required sequence in the timeline tab, click and hold the icon, and then select **Cache Sequence Range**.

- Select the required sequence in the timeline tab and then select **Cache > Disk Cache > Cache Sequence Range** from the menu bar.
• In the **Project** bin, right-click the sequence you want to cache, and then select **Cache > Cache Sequence Range**.

See [Clearing Cached Frames](#) for information on how to clear frames from the disk cache.
Caching Selected Shot Ranges

Caching shot ranges allows you to be more selective than caching an entire sequence, though it works in the same way. All tracks are read from the highest track downwards, so what you see in the Viewer for the shot selection is what Hiero caches to disk.

Tip: You can pause caching at any time by navigating to **Cache > Disk Cache > Pause** in the menu bar.

As a result of this, you may not get the frames you expect. For example, caching the shot in the image produces frames from the three shots on the highest track, including the soft effect, not the selected reference media.

You can cache clip ranges from the timeline itself or from the **Cache** menu:

- Select the required clip range in the timeline tab, right-click and then select **Disk Cache > Cache Selected Shot Ranges**.
• Select the required clip range in the timeline tab and then select **Cache > Disk Cache > Cache Selected Shot Ranges** from the menu bar.

See **Clearing Cached Frames** for information on how to clear frames from the disk cache.
Caching In/Out Ranges

Caching ranges uses In and Out points set on the timeline to determine what Hiero caches to disk. All tracks are read from the highest track downwards, so what you see in the Viewer between the In and Out markers is what Hiero caches to disk. See Using In and Out Markers for more information.

**Tip:** You can pause caching at any time by navigating to **Cache > Disk Cache > Pause** in the menu bar.

After setting In and Out points, you can cache that range from the timeline itself or from the **Cache** menu:
- Right-click in the timeline tab and then select **Disk Cache > Cache In/Out Range**.
- Select **Cache > Disk Cache > Cache In/Out Range** from the menu bar.
See Clearing Cached Frames for information on how to clear frames from the disk cache.
Clearing Cached Frames

Frames in the timeline cache can be cleared in a number of ways, including Clear All frames across all projects and the more selective Clear Range option.

Clearing All Cached Frames

The Clear All option is only available from the Preferences dialog under Performance > Caching, because it clears all frames from all projects and should be used with care. Clicking Clear All displays a warning dialog containing all the projects that are affected.

Click OK to proceed or Cancel to retain all the frames in the cache.

Clearing Sequences

Instead of using the Clear All option, you can clear certain sequences within a project. You can clear a sequence from the timeline itself, the Cache menu, or the Project bin:

• Select the required sequence in the timeline tab, click and hold the icon, and then select Clear Sequence Range.
• Select the required sequence in the timeline tab and then select **Cache > Disk Cache > Clear Sequence Range** from the menu bar.

• In the **Project** bin, right-click the sequence you want to clear, and then select **Cache > Clear Sequence Range**.
Clearing Selected Shot Ranges

You can also clear selected ranges of shots from a sequence, rather than the entire sequence, from the timeline itself or from the **Cache** menu:

- Select the required clip range in the timeline tab, right-click and then select **Disk Cache > Clear Selected Shot Ranges**.
• Select the required clip range in the timeline tab and then select **Cache > Disk Cache > Clear Selected Shot Ranges** from the menu bar.

---

**Clearing In/Out Ranges**

Clearing frames using In and Out markers is the most fine-grained method for removing frames from the timeline cache. Only the frames bracketed by the markers are removed, regardless of how the frames were cached originally. See **Using In and Out Markers** for more information.

After setting In and Out points, you can clear that range from the timeline itself or from the **Cache** menu:

• Right-click in the timeline tab and then select **Disk Cache > Clear In/Out Range**.
• Select **Cache > Disk Cache > Clear In/Out Range** from the menu bar.
Viewing Multi-Format Timelines

All sequences and shots added to them adopt the **output resolution** and **clip reformat** settings from the **Preferences** dialog under **Project Defaults > General**. You can override these settings on a per **Project** or per **Sequence** basis.

The **Preferences** dialog controls the default settings for new projects. Existing projects are unaffected by changes to the preferences.

The **Project Settings** control the default settings for new sequences and shots in the current project. Existing sequences and shots are unaffected.

**Project Settings** override **Preference** settings.

The **Sequence** settings control the current sequence.

**Sequence** settings override **Project Settings** and **Preference** settings for the current timeline.

The **Properties** settings control shots in the current sequence.

**Properties** settings override **Sequence** settings, **Project Settings**, and **Preference** settings for shots on the current timeline.

Reformatting applied to shots on a timeline carry over into any export, including **Create Comp**. The reformat options in the **Export** dialog are applied after the transforms applied here. See **Exporting from Nuke Studio** for more information.
Setting the Sequence Format

The current sequence format is controlled by the **Output Resolution** dropdown in the timeline **Sequence** tab. The selected resolution is applied to all shots on the timeline, but by default, shots retain their native resolution. For example, setting the **Output Resolution** to 2k_DCP on a timeline containing **HD_1080** shots results in the output resolution being larger than the shot.

**Tip:** Enabling the **Custom Sequence Format** guide at the top of the Viewer makes it easier to see differences between output resolution and shot format.

A 1920x1080 shot at native resolution.  
The same shot with **Output Resolution** set to 2048x1080.

Similarly, setting the **Output Resolution** to a format smaller than the shot's native resolution results in the clip resolution being larger than the sequence resolution.

A 1920x1080 shot at native resolution.  
The same shot with **Output Resolution** set to 720x486.

Shots with a higher resolution than the sequence format enable you to apply soft effects, such as Transform, more easily. See **Soft Effects** for more information.
Reformatting Shots

Shots on the timeline retain the source clip's resolution by default. You can force shots to the sequence format using the timeline's Properties tab:

1. Select the target shots on the timeline.
2. Click the Properties tab to display the shot options.

3. Click the Clip Reformat dropdown and select To Sequence Resolution.
4. The shot is reformatted to the Output Resolution specified on the Sequence tab.
5. Depending on the source clip, you may need to change the Resize Type:
   - Width - scales the original until its width matches the format's width. Height is then scaled to preserve the original aspect ratio.
   - Height - scales the original until its height matches the format's height. Width is then scaled to preserve the original aspect ratio.
   - Fit - scales the original until its smallest side matches the format's smallest side. The original's longer side is then scaled to preserve original aspect ratio.
   - Fill - scales the original until its longest side matches the format's longest side. The input's shorter side is then scaled to preserve original aspect ratio.

Note: The Volume control is only available when you have an audio shot selected.
• **Distort** - scales the original until all its sides match the lengths specified by the format. This option does not preserve the original aspect ratio, so distortions may occur.

*TIP:* When **Clip Reformat** is **To Sequence Resolution**, disabling **center** places the clip at the bottom-left of the **Output Resolution**, rather than the center.
Refreshing and Replacing Shots

During the post process, media inevitably changes location or form. Nuke Studio can reload or replace your media using the refresh, rescan, reconnect, and replace functions.

Though all four options deal with reloading shots, each has a particular use dependent on context:

- **Reconnect Media** - allows you to redirect the file path when the source file location changes.
- **Replace Clip** - replaces the selected shot with a specified source clip. Nuke Studio assumes that any source clip you choose is acceptable, regardless of timecode.
- **Refresh Clips (F8)** - allows you to reload the shot when the source file location has not changed, such as when work has been done on the clip offline. Selecting refresh only refreshes the clip's current frame range.
- **Rescan Clip Range (Alt+F5)** - similar to Refresh Clips, above, but rescan also checks for additional frames that may have been added to the source file and adds them to the shot's frame range.
- **Set Soft Trims** - sets the files handles on the selected clip(s). See Setting Soft Trims for more information.
Setting Soft Trims

Soft Trims limit the handles on shots to a pre-defined amount, simulating In and Out points on the source clips, allowing you to use other timeline tools on the shots such as Slip Clip and Slide Clip.

To set Soft Trims on a shot(s):
1. Select the shot(s) on the timeline.
2. Right-click and select Clip > Set Soft Trims.
   The Set Soft Trims dialog displays.
3. Set the number of frames to add to the head and tail of each shot:

   ![Diagram showing Soft Trims]

   - **Use full available range** - sets the handles to the full extent of the source clip frame range.
   - **Use Frames** - adds the specified number of frames to the head and tail of the shot(s).
4. Click OK to add the specified number of handles.

   If the handles requested are not within the available frame range, a warning dialog displays with a suitable correction for each selected shot.

   ![Warning dialog]

   Click **Yes** to accept, or **No** to abort the operation.
Note: With shots used in multiple sequences, click Yes to All to accept the correction in all instances.
Enabling and Disabling Shots

You can temporarily enable or disable tracks and shots on the timeline to selectively view your media without removing shot(s), for example if you wanted to view to lower level video tracks within a timeline.

To enable or disable a track or shot(s):
1. Select the item(s) you want to enable or disable.
2. Right-click on a highlighted item and select Editorial > Disable Track or Disable Items to disable the selection.

**Tip:** You can also use the D keyboard shortcut to disable or enable your selection.

Disabled items appear gray, and are effectively removed from the timeline.

3. Right-click the item and select Enable Clip, or press D again, to re-enable the clip.
Adding Transitions

Nuke Studio supports basic video and audio fade transitions as well as dissolves between shots on the same track. Transitions come in three flavors:

- **Fade in** - fades in from black on a single shot.
- **Fade out** - fades out to black on a single shot.
- **Dissolve** - fades out from one shot and into the next, by merging frames.

**Tip:** Once a transition is in place, it can be nudged in the same way as an edit using the , (comma) and . (period) keyboard shortcuts, providing the required handles exist.

To add a fade transition:

1. Right-click the target shot and select **Editorial > Add Transition > Fade In, Fade Out, Audio Fade In** or **Audio Fade Out** to add the fade icon.
2. Adjust the fade by dragging the fade icon using the **Multi Tool** or **Move/Trim** tool.

To add a dissolve transition:
Note: You can only add dissolves between shots when they're on the same track and have sufficient handles available on both sides of the transition.

1. Select the **Multi Tool** or **Roll Edit** tool and hover the mouse pointer over an edit between two shots.

   **Tip:** Clicking and holding the edit point displays available handles as a red overlay.

2. Right-click and select **Editorial > Add Transition > Dissolve** or **Audio Crossfade**, or use the **Ctrl/Cmd+T** keyboard shortcuts, to add the dissolve icon to the edit.

3. Adjust either side of the dissolve by dragging the icon, in a similar way to using the **Multi Tool** or **Move/Trim** tool.
Invalid Transitions

Transitions are controlled in a similar way to shots, in that you can drag-and-drop them, but with the following restrictions:

• A fade can not be dragged past the ends of the shot it's attached to, and if the item is deleted, the fade is deleted with it.
• Dissolve ends can not be dragged past the ends of the shots they are attached to, and if both items are deleted, then the dissolve is also deleted.

If only one of the shots linked by the dissolve is deleted, the transition remains so that another item can be placed on the other side.

Invalid transitions are colored red on the timeline. In most cases, adjusting the length of the transition should be enough to correct the error.
Retiming Shots

In addition to transitions, Nuke Studio supports constant retimes on shots. Decreasing the speed of a shot causes frames to play more than once, whereas increasing the speed skips frames.

**Note:** Audio is not currently supported for retimes and is automatically muted to avoid playback problems.

To retime shots using the **Speed** column in the spreadsheet:

1. Select the required event(s) in the spreadsheet view.
2. Click the cog icon and select the required **Retime method**:
   - **Keep source duration** - the shot length is altered on the timeline depending on the retime applied.
     - For example, retiming a shot to 50% renders frames 1, 1, 2, 2, 3, 3, 4, 4, and so on in the Viewer, and as a result, the item’s length is doubled on the timeline.
     - Retiming a shot to 200% renders frames 1, 3, 5, 7, and so on in the Viewer, but the item’s length is halved on the timeline.
   - **Keep timeline duration** - the shot length on the timeline is maintained regardless of the retime applied.
     - For example, retiming a shot to 50% renders frames 1, 1, 2, 2, 3, 3, 4, 4, and so on in the Viewer, but the item’s length on the timeline remains the same, effectively removing the second half of the item.
     - Retiming a shot to 200% renders frames 1, 3, 5, 7, and so on in the Viewer, but the item’s length on the timeline remains the same. If no extra frames are available from the source, the item is filled with black frames.
3. Double-click the **Speed** column and enter the retime value.

The following example shows a shot and the results of 50% and 200% retimes with the **Keep source duration** and **Keep timeline duration** methods selected.

Notice that the **Keep timeline duration** method doesn't change the length of the shot on the timeline and inserts blank filler frames on the 200% retime?
You can also retime shots using the **Src**, **Dst**, and **Duration** columns of the spreadsheet, though the calculation method depends on the **Time Edit Behaviors** applied.

1. Select the event(s) in the spreadsheet view.
2. Click the cog icon and select the required **Time Edit Behaviors**:

<table>
<thead>
<tr>
<th>Modify</th>
<th>Using</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Src In</td>
<td>Retime</td>
<td>Adjusts the event's <strong>Src In</strong> and retimes the remaining frames to maintain <strong>Dst Duration</strong>. Before and after a 2 second <strong>Src In</strong> increase:</td>
</tr>
</tbody>
</table>

![Original clip](image)

![50% Source retime](image)

![200% Source retime](image)

![50% Destination retime](image)

![200% Destination retime](image)
<table>
<thead>
<tr>
<th>Modify</th>
<th>Using</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Src Out</td>
<td>Retime</td>
<td>Adjusts the event’s <strong>Src Out</strong> and applies a retime to maintain <strong>Dst Duration</strong>. Before and after a 2 second <strong>Src Out</strong> increase:</td>
</tr>
<tr>
<td><img src="image1.png" alt="Before" /></td>
<td><img src="image2.png" alt="After" /></td>
<td></td>
</tr>
<tr>
<td>Src Dur</td>
<td>Retime</td>
<td>Adjusts the event’s <strong>Src Dur</strong> and <strong>Src Out</strong>, and applies a retime to maintain <strong>Dst Duration</strong>. Before and after a 50 frame <strong>Src Dur</strong> increase:</td>
</tr>
<tr>
<td><img src="image3.png" alt="Before" /></td>
<td><img src="image4.png" alt="After" /></td>
<td></td>
</tr>
<tr>
<td>Dst In</td>
<td>Retime</td>
<td>Adjusts the event’s <strong>Dst In</strong> and retimes the remaining frames to maintain the relationship between <strong>Dst In</strong> and <strong>Out</strong>. Before and after a 2 second <strong>Dst In</strong> increase:</td>
</tr>
<tr>
<td><img src="image5.png" alt="Before" /></td>
<td><img src="image6.png" alt="After" /></td>
<td></td>
</tr>
<tr>
<td>Dst Out</td>
<td>Retime</td>
<td>Adjusts the event’s <strong>Dst Out</strong> and retimes the remaining frames to maintain the relationship between <strong>Dst Out</strong> and <strong>In</strong>. Before and after a 2 second <strong>Dst Out</strong> increase:</td>
</tr>
<tr>
<td><img src="image7.png" alt="Before" /></td>
<td><img src="image8.png" alt="After" /></td>
<td></td>
</tr>
</tbody>
</table>
Modify | Using | Result
---|---|---
Dst Dur | Retime | Adjusts the event’s Dst Dur and Dst Out, and applies a retime to accommodate the new Dst Duration.

Before and after a 50 frame Dst Dur increase:

<table>
<thead>
<tr>
<th>Speed</th>
<th>Src In</th>
<th>Src Out</th>
<th>Src Dur</th>
<th>Dst In</th>
<th>Dst Out</th>
<th>Dst Dur</th>
</tr>
</thead>
<tbody>
<tr>
<td>100.0%</td>
<td>00:00:00:01</td>
<td>00:00:04:00</td>
<td>100</td>
<td>01:30:00:00</td>
<td>01:00:03:24</td>
<td>100</td>
</tr>
</tbody>
</table>

3. Adjust the values as required to retime the shot(s) by the specified amount.

**Note:** Any source or destination field highlighted in yellow indicates that the entry has been rounded down for display purposes.

To retime a shot using the **Timeline** menu:
1. Select the required shot(s) on the timeline.
2. Navigate to **Timeline > Retime**.
   * The **Clip Speed** dialog displays.
3. Enter the required retime value as a percentage.
4. Select the required retime method using the dropdown:
   • **Keep timeline duration** - the shot length on the timeline is maintained regardless of the retime applied. When increasing speed, if no extra frames are available from the source, the shot is filled with black frames.
   • **Keep source duration** - the shot length is altered on the timeline depending on the retime applied. For example, a 200% retime halves the length of the item.
   • **Anchor current frame** - the shot length on the timeline and the current frame’s position are maintained after the retime. When increasing speed, if no extra frames are available from the source, the shot is filled with black frames.
5. Click **OK** to retime the shot(s).
Using Freeze Frames

The freeze frame feature enables you to create shots of any length using a single frame. To achieve this, the application takes the first frame of the shot and applies a 0% retime, which is reversible by applying a 100% retime.

To freeze frame shots:
1. Select the target item(s) on the timeline.
2. Right-click the item and select **Editorial > Make Freeze Frame**, OR
   Navigate to **Timeline > Make Freeze Frame**.

   **Note:** Freeze Frames can also be created using the spreadsheet retime modes to modify **Src Dur** to 0, or make **Src In** and **Src Out** equal.

3. The selection is retimed to 0% and colored blue on the timeline for easy identification.

4. Use the **Multi Tool** or **Slip Clip** to set the freeze frame from the available range.
5. Drag the item's edit points, using the **Multi Tool** or **Move/Trim** as required, to set the length of the shot. There's no upper limit for the length of a freeze frame shot.
Blending Tracks on the Timeline

Nuke Studio allows you to perform merges between tracks in the timeline, for example overlaying a logo on a shot without heading into the Compositing environment. Tracks that are designated as blend tracks have a blue header in the timeline for convenience and are blended using a sub-set of Nuke’s Merge node operations.

See the Nuke Online Help for a full description of the available blend modes.

**Note:** The Viewer currently treats all alpha channels as premultiplied, which can result in the Viewer background being “added” to the image.

You can add soft effects to blended tracks as normal (see Soft Effects for more information) and blended tracks are included along with the shot in Export operations. See Exporting from Nuke Studio and Create Comp for more information.

Adding New Blend Tracks

1. Right-click in the timeline, select **New Track > New Video Track Blend**, and then choose the blend mode to apply.
A new track is added at the top of the track stack, colored blue to indicate that it's going to be blended over the track below.

2. Add the required shot to the blend track as you would any other shot. See Adding Clips to the Timeline for more information.

3. Click and hold the Blend icon to select the blend mode.

4. Click the Blend icon to toggle blending on and off.

Converting Tracks to Blend Tracks

1. Click the Blend icon to toggle blending on,

   OR

   Right-click in the header of the target track and select Editorial > Blend Mode > Enable Track Blend.
The selected track is converted into a blend track, colored blue to indicate that it's going to be blended over the track below.

2. You can add shots to the blend track as you would any other shot. See Adding Clips to the Timeline for more information.

   The Viewer displays the higher track blended with the track below.

3. Click and hold the **Blend** icon to select the blend mode.

**Masking Blended Tracks**

**Masking** limits the effect of the blend track to just those areas covered by the alpha channel in the blend image. For example, using the **Multiply** blend mode with masking disabled multiplies the background plate in non-alpha areas, which may not be the result you require.
Blending Tracks on the Timeline | Masking Blended Tracks

Background track B.  
Blend track A.

A multiplied by B with Masking enabled.  
A multiplied by B with Masking disabled.

See the Nuke Online Help for a full description of the available blend modes.

Click the Mask icon to toggle alpha masking on and off. The mask option also carries over into the Node Graph when you Create Comp for a masked blend operation:

Comp of A over B using the multiply merge operation with masking disabled.  
The same comp using the multiply merge operation, but with masking enabled.
Stereoscopic and Multi-View Projects

Nuke Studio provides multi-view support for as many views as you need. The views do not have to be stereo pairs, but since that is the most obvious application, these pages mainly deal with stereoscopic projects. See Creating Views in a Project and Importing Source Clips for more information.

Existing views inside a project are managed in the Viewer, timeline, and in the Properties panel of most soft effects. In the Viewer, all views in the current project are represented by buttons that allow you to switch between views with a single click. See Displaying Views in the Viewer for more information.

The timeline employs a views button that allows you to switch between views per track. See Displaying Views in the Timeline for more information.
The **Properties** panel includes a split button for controls that support multiple views. Split controls only affect individual views. See [Applying Changes to Selected Views](#) for more information.

You can create comps or export multi-view shots and effects in a similar way to regular shots. See [Exporting Multi-View Source Clips](#) for more information.
Creating Views in a Project

Views in Nuke Studio are managed in the **Project Settings**. Views can be processed separately or together and you can see the effect of your changes on each view. If you're working with multi-view source clips, such as `.exr` and `.sxr`, Nuke Studio offers to create views for you on import. See [Importing Multi-View Clips](#) for more information.

Creating and Managing Views

If you're working with single-view clips, it's a good idea to create the project views before you import your footage so that the views are assigned correctly. If you're working with multi-view clips, the views are set up automatically on import.

1. Select **Project > Edit Settings**.
2. Go to the **Views** sub-menu. The available views are listed in the **Views** panel.
3. If you want to remove the view called **main** and add views called **left** and **right**, click the **Set up views for stereo** button. The two views are assigned colors. To change the colors, double-click on the color field and select another color from the color picker that opens.
4. Enable **Use colors in UI** to apply the selected color for each view to the associated button above the Viewer.
You can add and remove views using the + and - buttons or move views using the up and down arrows above the views panel.

Each view has its own button above the Viewer controls.

Tip: If you decide that you only need the **main** view in the project, click **Set up Views for Mono** in the **Project Settings**.

See **Importing Multi-View Clips** for information on reading source clips in to Nuke Studio.
Importing Source Clips

Nuke Studio supports multi-view clips in two formats:

- **single-view** - formats such as `.dpx` and `.jpg`, with file names that Nuke Studio can interpret as multi-view using specific naming conventions. For example:

  ```
  myStereo_comp1_left.####.dpx and myStereo_comp1_right.####.dpx
  ```

  See Importing Single-View Clips for more information.

- **multi-view** - formats such as `.exr`, `.srx`, and `.mov`, that support multiple views per file. For example:

  ```
  myStereo_comp1.####.exr containing two layers called left and right.
  ```

**Note:** Nuke Studio can assign any shot on the timeline to any view in the project, but stereo is the most common use case. See Importing Multi-View Clips for more information.

Importing Single-View Clips

After setting up views in the **Project Settings**, Nuke Studio can interpret single-view clips as multi-view using specific naming conventions. These conventions can be applied to the file names themselves or to the directories in which they reside.

**File Name Variables**

If the images you want to read in contain a view name or the initial letter of one (for example, `left`, `right`, `l` or `r`) in their file names, replace this with the variable `%V` or `%v` in the file name. Use `%V` to replace an entire view name (for example, `left` or `right`), and `%v` to replace an initial letter (for example, `l` or `r`). When a variable is used, Nuke Studio reads in the missing inputs and combines all inputs into a single output.

**Note:** You can enable detect views in the file browser to automatically substitute the `%V` or `%v` variable where possible.

To use the `%V` and `%v` variables manually:
1. Click **File > Import File(s)** or press **Ctrl/Cmd+I** to display the file browser.
2. Locate a single-view file, for example **pubstitch.left.####.dpx**
3. Replace the view name with the **%V** variable, continuing the example **pubstitch.%V.####.dpx**

![File browser screenshot]

4. Click **Open**.

Nuke Studio reads in both **pubstitch.left.####.dpx** and **pubstitch.right.####.dpx** with the same Read node, provided that views called **left** and **right** exist in the **Project Settings**.

**Note:** Mac and Linux operating systems can be case-sensitive or case-insensitive. If your OS is case-sensitive, you'll need to make sure you use the correct case when naming your left and right views, as the **%v** variable can only retrieve the case used in the view name.

Both input images are combined into a single source clip, marked with a **✓** icon, which can display any of the combined views.
See Displaying Views in the Viewer and Displaying Views in the Timeline for more information.

Directory Name Variables

You can also use the %V and %v variables at a directory level. For example, if you have set up views called cam3, left, and right, and you have the following directories and files:

- stereo/pubstitchcam3/pubstitch_cam3.#####.dp
- stereo/pubstitchleft/pubstitch_left.#####.dp
- stereo/pubstitchright/pubstitch_right.#####.dp

If you read in pubstitch_cam3.#####.dp and change the file path to stereo/pubstitch%V/pubstitch_%V.#####.dp, all three inputs are read in with the same Read node, providing that the cam3, left, and right views exist in the Project Settings.
All three input images are combined into a single source clip, marked with a \( \checkmark \) icon, which can display any of the combined views.

See Displaying Views in the Viewer and Displaying Views in the Timeline for more information.
Importing Multi-View Clips

Nuke offers to automatically create views when you import single files containing multiple views, such as .exr and .srx, unless they already exist. For multiple files, views must be created manually. See Importing Source Clips for more information.

1. Click File > Import File(s) or press Ctrl/Cmd+I to display the file browser.
2. Locate a multi-view file, for example pubstitch_stereo.####.exr
3. Click Open.

The Create missing views? dialog displays.

4. Click Add Views, Replace Views, or No:
   • Add Views - add the views in the incoming clip to those that exist in the project.
   • Replace Views - replace all existing project views with those in the incoming clip.
   • No - import the clip and display only the first view in the file, retaining any existing views in the project.

You can now access the views in your project. See Displaying Views in the Viewer and Displaying Views in the Timeline for more information.

Multi-View QuickTime Files

Multi-view .mov files only display one view by default. To enable all views in a multi-view .mov file:
1. Double-click the .mov to display its Properties panel.
2. Disable First track only.

You’ll notice that the .mov in the bin is now marked with ✅ to denote multiple views.

You can now access the views in your project. See Displaying Views in the Viewer and Displaying Views in the Timeline for more information.
Displaying Views in the Viewer

You can only display the views that exist in your Project Settings. To see a list of these views, or to add or delete views, select Project > Edit Settings and go to the Views tab. For more information, see Creating Views in a Project.

Displaying a Particular View

1. Double-click the clip or sequence to load it in the timeline Viewer.
2. On top of the Viewer controls, click the view to display. In the example, main, left or right.

**Tip:** You can also press the ; (semicolon) and ’ (forward single quote) keys to move between different views in the Viewer.

**Note:** Nuke Studio lists the views in .exr files in the order they appear in the clip's header, so a view named 'left' may not always be the first view displayed above the Viewer. If your views do not appear in the correct order, you can rearrange them in the Project > Edit Settings > Views tab. See Creating Views in a Project for more information.

Displaying Two Views Next to Each Other

1. Right-click in the Viewer and select the Stereo Modes menu.
2. Select one of the following options:
Displaying Views in the Viewer

- **Side by Side** - displays the views side by side at the correct aspect ratio, and adds selection controls above the Viewer.
- **Squeezed Side by Side** - displays the views side by side and squeezed to fit the format horizontally, and adds selection controls above the Viewer.
- **Squeezed Above by Below** - displays the views above and below each other and squeezed to fit the format vertically, and adds selection controls above the Viewer.

Displaying a Blend Between Two Views

1. Right-click in the Viewer and select the **Stereo Modes** menu.
2. Select one of the following options:
   - **Interlace H** - displays the views interlaced horizontally, and adds selection controls above the Viewer.
   - **Interlace V** - displays the views interlaced vertically, and adds selection controls above the Viewer.
   - **Checkerboard** - displays the views using an alternating checkerboard pattern (one pixel from left and one pixel from right), and adds selection controls above the Viewer.
   - **Anaglyph** - displays the views simultaneously using a red hue for left and green hue for right, and adds selection controls above the Viewer.
   - **Flicker** - displays both views alternately, and adds selection controls above the Viewer.

Displaying OpenGL Stereo in Timeline Viewers

The Viewer **OpenGL Stereo** mode allows you to see both views at once on a 3D monitor for review purposes. OpenGL Stereo is only supported on NVIDIA Quadro series GPUs and AMD Radeon Pro series GPUs on Windows and Linux OS.

**Note:** **OpenGL Stereo** mode is not supported on Mac due to limitations in macOS and not supported with AMD GPUs on Linux.

Enabling OpenGL Stereo Output

**Windows**

To enable NVIDIA GPU stereo output:
1. Right-click on the desktop and select NVIDIA Control Panel.
2. Navigate to **3D Settings > Manage 3D Settings > Stereo - Enable.**

![3D Settings](image)

3. Proceed to **Switching to OpenGL Stereo Output.**

To enable AMD GPU stereo output:
1. Double-click the AMD taskbar icon and select **Advanced Settings.**
2. Navigate to the AMD Pro Settings and check **Enable Quad Buffer Stereo.**

![Radeon Pro and AMD FirePro Advanced Settings](image)

3. Select either **Auto-Stereo (Horizontal Interleaved)** or **Auto-Stereo (Vertical Interleaved)** and click **Apply.**
4. Proceed to **Switching to OpenGL Stereo Output.**
Linux

To enable NVIDIA GPU stereo output:
1. Open a command prompt and enter:
   ```bash
   nvidia-xconfig --stereo=3
   ```

   **Tip:** For more information on the `nvidia-xconfig` utility, please see the `man` page: `man nvidia-xconfig`

2. Proceed to **Switching to OpenGL Stereo Output**.

   **Note:** OpenGL Stereo mode is not supported with AMD GPUs on Linux.

Switching to OpenGL Stereo Output

1. Right-click in the Viewer to display the context-sensitive menu.
2. Navigate to Stereo Modes and select OpenGL Stereo.
   
   If you select OpenGL Stereo mode before enabling your GPU settings, a warning is displayed.

   ![Warning Message]
   
   The first time you select OpenGL Stereo, a warning message is displayed.

   ![OpenGL Stereo Warning]

   **Tip:** You can disable the warning by enabling **Don't show again** and clicking **OK**.
OpenGL Stereo is displayed in the Timeline Viewer.

**Note:** Switching to and from OpenGL Stereo mode causes playback to pause. Press play to resume playback.
Displaying Views in the Timeline

You can only display the views that exist in your Project Settings. To see a list of these views, or to add or delete views, select Project > Edit Settings and go to the Views tab. For more information, see Creating Views in a Project.

Adding a multi-view clip to the timeline groups all views into a single track. All the views in the clip are assigned a Viewer button. Click the assigned view icon to display the views available in the shot. All Views are visible by default.

1. Right-click a multi-view shot on the timeline and select Editorial > Split Views to Tracks.

Note: You can also import single-view files manually and then assign them views in the timeline individually, providing that the views exist in the Project Settings.

Splitting Views to Separate Tracks

Working with all views in a single track is not always convenient, such as when you need to apply different color corrections to stereo cameras. Nuke Studio includes a Split Views to Tracks option that quickly separates views into individual tracks. You can use Split Views to Tracks with grouped single- and multi-view shots.

1. Right-click a multi-view shot on the timeline and select Editorial > Split Views to Tracks.
A separate track is created for each view in the group or file.

The new track names are suffixed with the view name, for example Video1_left, and the views are assigned appropriately.

**Note:** If a view exists in the Project Settings, but there's no corresponding view in the source files, empty placeholder tracks are added.

2. You can change the view assigned to a track by clicking the icon and selecting from the list of available views.
Displaying Views in the Timeline | Splitting Views to Separate Tracks

See Applying Changes to Selected Views for information on adding soft effects to different views.

**Displaying Two Views Next to Each Other**

1. Right-click in the Viewer and select the **Stereo Modes** menu.
2. Select one of the following options:
   - **Side by Side** - displays the views side by side at the correct aspect ratio, and adds selection controls above the Viewer.
   - **Squeezed Side by Side** - displays the views side by side and squeezed to fit the format horizontally, and adds selection controls above the Viewer.
   - **Squeezed Above by Below** - displays the views above and below each other and squeezed to fit the format vertically, and adds selection controls above the Viewer.

**Displaying a Blend Between Two Views**

1. Right-click in the Viewer and select the **Stereo Modes** menu.
2. Select one of the following options:
   - **Interlace H** - displays the views interlaced horizontally, and adds selection controls above the Viewer.
   - **Interlace V** - displays the views interlaced vertically, and adds selection controls above the Viewer.
   - **Checkerboard** - displays the views using an alternating checkerboard pattern (one pixel from left and one pixel from right), and adds selection controls above the Viewer.
• **Anaglyph** - displays the views simultaneously using a red hue for left and green hue for right, and adds selection controls above the Viewer.

• **Flicker** - displays both views alternately, and adds selection controls above the Viewer.
Applying Changes to Selected Views

By default, Nuke Studio applies any changes you make to all views. To apply changes to a particular view only (for example, the left view but not the right), you can:

- Split the views into separate tracks. Separating views allows you to apply effects, such as a Transform or Timewarp, to individual views without affecting the other views. See Splitting Views to Tracks for more information.
- Split the view off in the soft effect's Properties. Splitting properties allows you to perform the same operation on multiple views, but with different values for each, such as a Grade or ColorCorrect. See Splitting Views in the Properties Panel for more information.

Splitting Views to Tracks

1. Right-click a multi-view shot on the timeline and select Editorial > Split Views to Tracks.

2. Right-click the track to which you want to add the soft effect, select Effects and then the required soft effect. For example, adding a Transform to the left view applies the effect to only the left view track. See Soft Effects for more details on adding effects to shots.
You can create comps or export multi-view shots and effects in a similar way to regular shots. See Exporting Multi-View Source Clips for more information.

Splitting Views in the Properties Panel

1. Right-click the track to which you want to add the soft effect, select **Effects** and then the required soft effect. For example, adding a ColorCorrect a multi-view shot.

   See **Soft Effects** for more details on adding effects to shots.

2. Select the view you want to make changes to using the buttons above the timeline Viewer.
3. Open the effect’s **Properties** and click the view button on the right, next to the control you want to adjust.

4. Select **Split off [view name]**. For example, to apply changes to a view called **left**, select **Split off left**. You can also split all the effect’s controls by selecting **Split all knobs** from the right-click menu.

   ![Split view button](image)

An eye appears on the split view button. Any changes you make using the control in question are only applied to the view you chose to split off. Changes to controls that have not been split off are still applied to all views.

You can create comps or export multi-view shots and effects in a similar way to regular shots. See [Exporting Multi-View Source Clips](#) for more information.

### Showing Separate Values for Each View

Once you have split off a view, you can apply changes to the existing views separately. Click on the arrow on the left side of a split control to divide the control into values for each view.

![Split control](image)

In the example, the **left** view is split for the **saturation** control and **left** and **cam3** views are split for the **contrast** control.
Unsplitting Views

1. In the effect's controls, click the view button.
2. Select Unsplit [view]. For example, to unsplit a view called left, you'd select Unsplit left. The view is unsplit, and all changes you made to individual views are lost.

Note: The * (asterisk) denotes there is more than one unsplit view remaining for the saturation control and ( ) denotes that the right view is the only unsplit view for the contrast control.
Soft Effects

You can add soft effects to your timeline in any of the workspaces. A soft effect is a real-time effect, processed on GPU instead of CPU.

You can add custom plug-in or gizmo soft effects to the Add Effect menu using Python. Valid custom soft effects must have a GPUEngine implementation using DD::Image::Iop::gpuEngine type functions. For more information see Nuke's NDK Reference Guide.

Soft Effects must also be registered after creation. An example of how to register a plug-in or gizmo as a custom soft effect is located in:

<install_directory>/pythonextensions/site-packages/hiero/examples/custom_soft_effect.py

Available Soft Effects

Below is a brief summary of the available soft effects. These are similar to the tools in Nuke's Node Graph. See the Nuke Online Help for more information about them.

<table>
<thead>
<tr>
<th>Soft Effect</th>
<th>Summary</th>
</tr>
</thead>
<tbody>
<tr>
<td>Transform</td>
<td>Allows you to translate, rotate, scale, and skew shots from a single control panel.</td>
</tr>
<tr>
<td>Mirror</td>
<td>Allows you to flip the input image around the center of the format area. A flip on the x axis mirrors the image vertically. A flop on the y axis mirrors the image horizontally.</td>
</tr>
<tr>
<td>Crop</td>
<td>Allows you to cut out the unwanted portions of the image area. You can fill the cropped portion with black or adjust the image output format to match the cropped image.</td>
</tr>
<tr>
<td>TimeWarp</td>
<td>Allows you to slow down, speed up, or even reverse selected frames in a clip without necessarily altering its overall length. Sequences imported from .xml and .aff files also support non-linear retimes.</td>
</tr>
</tbody>
</table>

Note: Create Comp and Create Comp Special are not soft effects.
<table>
<thead>
<tr>
<th>Soft Effect</th>
<th>Summary</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Warning:</strong> TimeWarp effects are only allowed on tracks with clips as they are linked to the clips.</td>
<td></td>
</tr>
<tr>
<td>Grade</td>
<td>Allows you to define white and black points by sampling pixels from the Viewer. For example, you can use this for matching foreground plates to background plates.</td>
</tr>
<tr>
<td>LUT</td>
<td>Allows you to use the OpenColorIO library to load a colorspace conversion from a file (usually a 1D or 3D LUT) and apply it. You can also load other file-based transformations, for example an ASC ColorCorrection XML.</td>
</tr>
<tr>
<td>CDL</td>
<td>Allows you to apply an ASC CDL (American Society of Cinematographers Color Decision List) grade based on the OpenColorIO Library. For more information, see <a href="http://opencolorio.org">http://opencolorio.org</a></td>
</tr>
<tr>
<td>Colorspace</td>
<td>Allows you to convert images from one colorspace to another, for example from Nuke's native colorspace to other color spaces more appropriate to a given process or intended display device. This supports RGB, HSV, YUV, CIE, and CMS formats (and various sub-formats). It can adjust for different primaries, white point, and different encodings.</td>
</tr>
<tr>
<td>ColorCorrect</td>
<td>Allows you to make quick adjustments to saturation, contrast, gamma, gain, and offset. You can apply these to a clip's master (entire tonal range), shadows, midtones, or highlights.</td>
</tr>
<tr>
<td>Text</td>
<td>Allows you to add text overlays on your images. You can simply type in the text you want to have displayed or use Tcl expressions (such as [metadata values]) or Tcl variables to create a text overlay. Text overlays can also be animated using animation layers in the Groups tab, so that their properties (such as position, size, and color) change over time.</td>
</tr>
<tr>
<td>Burn-In</td>
<td>Allows you to quickly add standard burn-in elements on the timeline. You can control the color, opacity, font, scale, and so on, as well as use the dropdowns to determine what element is added from the file or sequence metadata.</td>
</tr>
<tr>
<td>Soft Effect</td>
<td>Summary</td>
</tr>
<tr>
<td>-------------</td>
<td>---------</td>
</tr>
<tr>
<td></td>
<td>You can also reference custom metadata from shots. For example: hiero/tags/Approved</td>
</tr>
<tr>
<td></td>
<td>Extracts the <strong>Approved</strong> tag from the shot. You can also append <strong>note</strong> to include any notes associated with the tag: hiero/tags/Approved/note</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> You must precede spaces and slashes in the tag name with <code>\</code> (backslashes) to enable Nuke Studio to process the tag name correctly. For example:</td>
</tr>
<tr>
<td></td>
<td>hiero/tags/Blue\ Screen/note</td>
</tr>
<tr>
<td></td>
<td>You can also add burn-in through the <strong>Export</strong> dialog, see Adding Burn-in Text to Exports for more information.</td>
</tr>
</tbody>
</table>

**ChromaKeyer**

- Allows you to pull a quick chroma key from green or bluescreen areas of your footage.
- Use the **screen color** selector to choose a color from the **Source** input to use as the blue/green screen color. To remove blue/green spill from the foreground object, use the **despill** controls to pick skin tones from the source. Use the **matte** parameters to improve the matte.

**BlinkScript**

- Allows you to run Foundry's Blink framework on the timeline, enabling you to write code once and run it on any supported device.

**Warning:** BlinkScript is very flexible, as there are no restrictions on the code you can write within a kernel. As a result, code compiled from the Kernel Source can cause Nuke to crash, so please use caution!

The BlinkScript soft effect supports a subset of the functionality available in the full BlinkScript node:

- You can't publish your kernels to Groups or Gizmos.
- Due to the way stacks of soft effects are processed in Nuke Studio, the BlinkScript soft effect only contains one input source and produces only one
<table>
<thead>
<tr>
<th>Soft Effect</th>
<th>Summary</th>
</tr>
</thead>
<tbody>
<tr>
<td>Soft Effect</td>
<td>output.</td>
</tr>
<tr>
<td>• Currently, the BlinkScript soft effect only supports eAccessPoint data access, which means that only one point from the input can be accessed at a time, and only one point from the output can be written, for each iteration position.</td>
<td></td>
</tr>
<tr>
<td>• The following functions are not supported by the BlinkScript effect:</td>
<td></td>
</tr>
<tr>
<td>• log10</td>
<td></td>
</tr>
<tr>
<td>• round</td>
<td></td>
</tr>
<tr>
<td>• rsqrt</td>
<td></td>
</tr>
<tr>
<td>• abs for integer types</td>
<td></td>
</tr>
<tr>
<td>• modf(a, *b)</td>
<td></td>
</tr>
<tr>
<td>• sign</td>
<td></td>
</tr>
<tr>
<td>• rcp</td>
<td></td>
</tr>
<tr>
<td>• max, min, and clamp for integer types</td>
<td></td>
</tr>
<tr>
<td>• median, atomicInc and atomicAdd.</td>
<td></td>
</tr>
<tr>
<td>• The only data types supported by the BlinkScript effect are int, float, and bool.</td>
<td></td>
</tr>
</tbody>
</table>

Note: The BlinkScript soft effect supports both pixel-wise and component-wise kernels, but the former is preferred for performance reasons. For more information about this and a more detailed description of the language, see [https://learn.foundry.com/nuke/developers/113/BlinkKernels/](https://learn.foundry.com/nuke/developers/113/BlinkKernels/)

---

**Adding Sequence-Level Soft Effects**

**Warning:** Please keep in mind the following:

• Sequence-level soft effects are only permitted on the same track as clips if they're trimmed to exactly match the in and out points of individual clips. In this case, each effect is linked to a specific clip.
• Soft effects can be trimmed arbitrarily if they're on tracks with no clips.

As well as adding soft effects using the spreadsheet view (see next section), the timeline provides some additional ways to add soft effects. You can either:
• Right-click a shot on the timeline, select **Effects** and then select the soft effect you want to apply, or
• Select a shot on the timeline, click the **Add Effect** button to the left of the timeline, and then select the soft effect you want to apply.

![Soft Effects](image)

**Note:** You can add a soft effect to multiple shots by selecting the required shots first and then right-clicking on one of them and selecting the soft effect you want to add. A soft effect is added to each of the selected shots. You can also add a single soft effect for the whole track by right-clicking on the track header and then selecting the soft effect you want to add.

**Note:** TimeWarp effects are only allowed on tracks with clips (and therefore linked to clips).

**Using the Spreadsheet View**

You can add soft effects using the spreadsheet view in any workspace by doing the following:
1. To open the spreadsheet view in any workspace, select **Window > New Spreadsheet View**.
2. Right-click a sequence in the bin view.
   This opens a context menu.
3. If the spreadsheet view is not already populated in the context menu, select **Open In > Spreadsheet View**.
This loads the sequence in the spreadsheet view that you previously opened.

4. Right-click an event from the list in the spreadsheet view and select **Effects** to open a list of all available soft effects.

**Note:** **Create Comp** and **Create Comp Special** are the only items in the **Effects** list that are NOT a soft effect.

5. Select the required soft effect from the **Effects** list.

The sequence-level soft effect is then displayed above the shot and is color coordinated. For example, if the effect appears in green on the timeline, the corresponding effect properties are highlighted in the same color in the **Properties** pane.

When you insert a soft effect, its properties panel opens automatically. If you have it open, the effect properties panel displays in the **Properties** pane. If the **Properties** pane is not open, the effect’s properties panel appears as a floating dialog.
Adding Shot-Level Soft Effects

Warning: Soft effects added at shot-level, must match the length of the shot on the locked track. Any soft effect that is trimmed beyond the end of a shot, or a different length from the shot is marked with red hashing to show that it is invalid.

You can add shot-level soft effects on the timeline, by doing the following:

1. Right-click the shot that you want to add a soft effect to.
2. Select Open In > Timeline View.
3. Click the Effects menu icon and select the soft effect you want to apply. For example, you can select Grade.

The soft effect is then displayed above the shot as a colored box.

When you close the timeline view of the shot – as it is a shot-level soft effect – the soft effect is displayed as a colored line within the top of the shot. The color of the line displayed reflects the highest soft effect added to the shot.

Soft Effect Controls

Adding an effect displays the associated controls in the Properties panel, similar to Nuke nodes. Adjusting the controls affects the shots underneath the effect in real-time. For example, adding a Grade effect at sequence level displays the Grade controls in the Properties panel.
See Properties Panels for more information on node controls.

If you intend to animate soft effect controls using keyframes, you can use the Curve Editor and Dope Sheet to fine-tune the output. To add the Curve Editor or Dope Sheet to the interface, navigate to Window and select the required panel.

See Animating Parameters for more information.
Editing Sequence-Level Soft Effects

You can copy, move, and cut soft effects just like you can with shots in the timeline. You can perform these actions by either accessing them from Edit in the right-click menu, or by using the keyboard shortcuts. You can copy soft effects to different tracks, and different sequences, but not different projects.

Moving

You can move a sequence-level soft effect by simply clicking and dragging the soft effect to a different shot, or even onto a different video track.

Copying

Nuke Studio allows you to copy a sequence-level soft effect above the original to create a stack, to a different track, or to a different sequence. You can also copy a sequence-level soft effect to a shot open in the timeline view, therefore pasting it as a shot-level soft effect. You can copy a soft effect by doing the following:
1. Select the soft effect you want to copy by clicking it.
2. Select Edit > Copy (or Ctrl/Cmd+C).
3. Move the playhead to where you want to paste the copy.
4. Select Edit > Paste (or Ctrl/Cmd+V).

Cloning

Nuke Studio allows you to clone a sequence-level soft effect. This copies the soft effect and links it to the original, which means when one of these is edited, the changes are reflected in the other one. You can clone a soft effect to a different track or even a different sequence. You cannot clone a soft effect in different projects.

To clone a soft effect:
1. Select the soft effect you want to clone by clicking on it.
2. Select Edit > Copy (or press Ctrl/Cmd+C).
3. Move the playhead on the timeline to where you want to place the new clone.
4. Select Edit > Clone (or press Alt+K).

The new clone is placed at the current playhead position on the timeline. You can repeat steps 3 and 4 to create multiple clones that are all linked, at different places on the timeline.
Clones are indicated by a C highlighted in red in the left of the soft effect.

**Note:** Cloning animation in soft effects is not supported.

### Copying as Clone

You can also copy a sequence-level soft effect as a clone. This means, when you paste a new copy of the soft effect above a selected shot, it is automatically linked to the original soft effect as a clone. Therefore, any changes made to either of the cloned soft effects, are reflected in the other.

To copy a soft effect as a clone, do the following:

1. Select the soft effect you want to copy as a clone by clicking on it.
2. Select **Edit > Copy as Clones** (or press Ctrl/Cmd+K).
3. Click on the shot that you want to clone the soft effect to.
4. Select **Edit > Paste** (or press Ctrl/Cmd+V).

   The soft effect is copied as a clone on your selected shot. You can repeat steps 3 and 4 to create multiple clones that are all linked above different selected shots.

   Clones are indicated by a C highlighted in red in the left of the soft effect.

**Note:** Cloning animation in soft effects is not supported.

### Decloning

To declone a soft effect, simply click on the clone you want to declone and select **Edit > Declone** (or press Alt+Shift+K).

**Note:** For more information about copying, moving, and cutting soft effects, see [Timeline Editing Tools](#).

### Deleting

To delete a soft effect, simply right-click on it and select **Edit > Delete** (or press Backspace).
Editing Shot-Level Soft Effects

To edit a shot-level soft effect, you first need to open the shot with the applied soft effect, in the timeline view. You can do this by right-clicking the shot and selecting **Open In > Timeline View**. You can then copy, delete or move the shot-level soft effect in exactly the same way as sequence-level soft effects.

You can copy and paste the shot-level soft effect on top of the original, creating a stack in the timeline view. When you close the timeline view, stacked shot-level soft effects are displayed as a single line within the top of the shot. Nuke Studio allows you to copy a shot-level soft effect, return to the full sequence, and then paste it as a sequence-level soft effect. You can also paste a shot-level soft effect to another shot open in the timeline view.

**Note:** You cannot clone shot-level soft effects.

Enabling and Disabling Soft Effects

You can choose to disable and re-enable soft effects from the output. To enable or disable a sequence-level soft effect, select the soft effect by clicking on the colored box and then pressing **D**. To enable or disable a shot-level soft effect, you first need to open the shot – that includes the soft effect – in the timeline view. Then you can select the soft effect and press **D**.
Create Comp

In the Timeline environment, you can choose to create a Nuke Comp of a shot to be able to open it in the Compositing environment. You can then add any necessary compositing work and render out the script.

You can only create comps from the timeline when the Frame Server is running. You can check the status of the server in the status bar at the bottom-right of the interface.

Before you create a Nuke Comp, you can choose to change Export settings, and set the required Shot Preset setting in the Project Settings dialog to get the required result.

It is not necessary to change the Export and Project Settings. If you don't change these settings when creating a Nuke Comp, the default settings are used.

**Note:** You can use Nuke Studio's internal Frame Server to take advantage of external slave machines to process renders faster. See Using the Frame Server on External Machines for more information.

Edit Export Settings

If you do not change the export settings, the Using Local export preset setting control just uses the default setting of Basic Nuke Shot With Annotations.

To change this setting, do the following:
1. Select File > Export... or press Ctrl+Shift+E.
This opens the Export dialog.

2. In the Using Local export preset: section, you can select the type of shot preset to export when you use Create Comp.

![Using Local export preset:]

For more information, see Introduction to the Export Dialog.

3. Select the required setting and then close the Export dialog.

Edit Project Settings

If you do not change the Project Settings, the Shot Preset control just uses the default setting of Basic Nuke Shot With Annotations.

To change the Shot Preset setting in the Project Settings dialog, do the following:

1. Select Project > Edit Settings.
2. Open the Export / Roundtrip section by clicking on it.
3. Use the Shot Preset dropdown to select your required setting.

**Note:** You can use custom shot presets, but they must contain a Nuke Project File and Nuke Write Node Content preset to be valid for selection. See Using Local and Project Presets, Nuke Project File Settings, and Nuke Write Node Settings for more information.
4. After selecting the required setting, close the **Project Settings** dialog.

Creating and Editing a Comp

To create a Nuke Comp of a shot, do the following:

1. Right-click on the shot that you want use to create a Nuke Comp.
2. Select **Effects > Create Comp**.
   - If you have not set a project root folder up in the **Project Setting** dialog, Nuke Studio asks you to set a project root folder.
3. Click **Choose**, select to the required location, and then click **OK**.
   - A Nuke Comp is created, and placed directly above the original shot on the next available track.
The Nuke Comp is displayed in light red, signifying that the Nuke Comp has not been rendered. You can choose to render it by either, right-clicking and selecting **Render Comp**, or by selecting **Render > Render Selected Comp Containers**.

**Note:** You can render all Nuke Comps by selecting **Render > Render All Comp Containers**.

4. Open the Nuke Comp in the Compositing environment in the same session by double-clicking the Nuke Comp. You can also open the Nuke Comp in the Compositing environment in the same session by either right-clicking on the Nuke Comp above the timeline, or in the Project bin and selecting **Open In > Node Graph**.

To open the Nuke Comp in a new NukeX session you can either hold **Ctrl/Cmd** and double-click on the Nuke Comp, or you can right-click on the Nuke Comp and select **Open In > New Nuke Session**.

If you have not set a project root folder up in the **Project Setting** dialog, Nuke Studio asks you if you want to save and set a project root folder.
You can now edit the script and add VFX using any of the tools available in the Compositing environment. See *Using the Compositing Environment*.

### Nuke Comp Colors

<table>
<thead>
<tr>
<th>Color of Comp</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Light Red</td>
<td>This signifies that the comp has not been rendered.</td>
</tr>
<tr>
<td>Dark Red (with <strong>OFF</strong>)</td>
<td>This signifies that the comp is offline.</td>
</tr>
</tbody>
</table>
Creating Multiple Shot Comps

You can create Nuke Comps from multiple shots on a single track or across multiple tracks, by doing the following:

1. Select multiple shots on the timeline by using Shift+click to select a row of adjacent shots and/or Ctrl/Cmd+click to select multiple non-adjacent shots. These can be across multiple tracks.
2. Right-click on one of the selected shots and select Effects > Create Comp.
   This opens a Create Comp dialog.
3. The **Create Comp** dialog displays all the selected clips, and allows you to select a master clip from the list simply by clicking on it.

   The master clip is highlighted yellow.

4. To create one Nuke Comp for all the shots, select the **Add a single comp across all shots** radio button. Or, if you want to create a Nuke Comp for each of the selected shots, select the **Add a separate comp for each shot** radio button.

   When you create a Nuke Comp on multiple shots on a single track, the Node trees are automatically connected and are displayed in reverse chronological order, as shown below.
When you create a Nuke Comp on multiple shots across different tracks, the Node trees from the same track are connected, but the Node trees on different tracks are not connected by default (see below). You can choose to have them connected using the Create Comp Special settings. See Create Comp Special for more information.

Create Comp Special

You can also choose to create a Nuke Comp using Create Comp Special. When you use the Create Comp Special option, a dialog containing export options is automatically opened, allowing you to set the export settings for the selected Nuke Comp.
To use the *Create Comp Special* option:

1. Right-click on the shot that you want to use to create a Nuke Comp.
2. Select **Effects > Create Comp Special**.
   
   This opens a **Create Comp** dialog that contains the available export settings.

![Create Comp dialog](image)

You can use this dialog to set the export location, set a version number, define the tracks to export, filter by tag, and define the range. See **Introduction to the Export Dialog** for more information.

3. Select the Nuke script in the **Path** section of the dialog, and then go to the **Content** section.
   
   A number of additional controls are displayed. You can use these to specify what your Nuke Comp includes and how it is created. You can select the **Connect Tracks** checkbox to connect all the Node trees – across different tracks – in the Node Graph. See **Introduction to the Export Dialog** for more information about these options.

4. After specifying your required settings, click **Create Comp**.
   
   A dialog appears stating you have changed the export templates. You can choose to keep the changes by selecting **Yes** or click **No** to discard the changes.
   
   The Nuke Comp is displayed in light red, signifying that the Nuke Comp has not been rendered. You can choose to render it by either, right-clicking and selecting **Render Comp**, or by selecting **Render > Render Selected Comp Containers**.

5. Double-click on the Nuke Comp to open it in the Compositing environment.
You can now add VFX using any of the toolsets available in the Compositing environment. See Compositing with Nuke.

Enabling and Disabling a Nuke Comp

You can disable and re-enable a Nuke Comp on the timeline by selecting the Nuke Comp you want to disable and pressing D.

Saving a New Version and Updating the Comp

When you are happy with the changes you've made, you can save a new version of the script by selecting File > Save New Comp Version (or pressing Alt+Shift+S).

After saving a new version of the Comp, you can update the original Nuke Comp you created. To version-up the Nuke Comp, do the following:
1. On the timeline, right-click the original Nuke Comp.
   The Nuke Comp is versioned-up. Depending on your Preferences > Performance > Threads/Processes > Rendering > background renders setting, the comp may need rendering manually.

Stereo and Multi-View Comps

Multi-view comps are similar to regular comps, but you can create a single comp with all views or separate comps per view. The Multi-View Nuke Shot template is selected automatically when you create a comp from a multi-view project.
1. Right-click on the shot that you want use to create a multi-view Comp. The shot can be a multi-view track or single views split into separate tracks.

   For separate tracks, you can right-click a single track to export all views or select all the per-track views and right-click.
2. Select **Effects > Create Comp**.

3. If you selected multiple per-track views to create the Comp, you can create a single Comp containing all views or a separate Comp per view and select the master shot.

   ![Create Comp dialog](image)

If you're using independent files per track, that is without importing multi-view files or using %V functionality, the separate tracks in the Comp script are not connected by default. If you want the tracks to be connected in the script, click the cog icon 🔄 to display the **Create Comp** dialog and enable **Connect Tracks** in the **Nuke Project File** preset.

![Export dialog](image)
4. Click **OK**.

A Nuke Comp is created, and placed directly above the original shot on the next available track.

The Nuke Comp is displayed in light red, signifying that the Nuke Comp has not been rendered. You can choose to render it by either, right-clicking and selecting **Render Comp**, or by selecting **Render > Render Selected Comp Containers**.

**Note:** You can render all Nuke Comps by selecting **Render > Render All Comp Containers**.

5. Open the Nuke Comp in the Compositing environment in the same session by double-clicking the Nuke Comp. You can also open the Nuke Comp in the Compositing environment in the same session by either right-clicking on the Nuke Comp above the timeline, or in the Project bin and selecting **Open In > Node Graph**.
To open the Nuke Comp in a new NukeX session you can either hold Ctrl/Cmd and double-click on the Nuke Comp, or you can right-click on the Nuke Comp and select Open In > New Nuke Session.

If you have not set a project root folder up in the Project Setting dialog, Nuke Studio asks you if you want to save and set a project root folder.

You can now edit the script and add VFX using any of the tools available in the Compositing environment. See Using the Compositing Environment.
Annotations

Annotations can be used as quick instructions for a compositor to implement in Nuke Studio’s Compositing environment. You can add annotations to a clip, a section marked with in and out points, or a whole sequence.

Annotations can be exported with a Nuke Comp and can then be viewed and/or deleted in the Compositing environment. When all the suggested changes have been made to the script in the Compositing environment, this can be saved as a new comp version and then rendered back to the Timeline environment. If you want to add new annotations to the rendered Nuke Comp, you can choose to re-export annotations only.

Workflow

The following steps show an example of Nuke Studio workflow for annotations:

1. In the Viewer, open the Annotations menu. See The Annotations Menu for more information for more information about it.
2. Add an annotation to a shot(s) at sequence or shot-level, by using the Annotation menu tools. See Adding Annotations for more information.
3. You can choose to edit a sequence or shot-level annotation after it has been created. See Editing Sequence-Level Annotations or Editing Shot-Level Annotations for more information.
4. Create a Nuke Comp of the shot with the annotations, ensuring annotations are enabled in the export settings.
5. Open the Nuke Comp by double-clicking it. See Viewing Annotations in the Compositing Environment for more information.
6. After the suggested changes are made in the Compositing environment, select File > Save New Comp Version.
7. Return to the timeline, and version up the Nuke Comp by right-clicking it and selecting Versions > Version Up.

The Nuke Comp is versioned up. Depending on your Preferences > Performance > Threads/Processes > Rendering > background renders setting, the comp may need rendering manually.
8. You can add new annotations to the rendered Nuke Comp by ensuring you select the Clip radio button and then using the Annotations menu tools.
9. After adding the new annotations, right-click the rendered Nuke Comp and select **Export > Re-Export Annotations**.
10. Open the rendered Nuke Comp by double-clicking it.
11. Double-click the Precomp node to open its properties and version it up. For example, if the file path has `v01.nk` at the end, change it to `v02.nk`. See **Re-Exporting Annotations from the Timeline** for more information.
12. Display the Precomp in the Viewer.
   Your new annotations are displayed.

The Annotations Menu

You can open the Annotations menu by selecting the paint brush icon at the top-right of the Viewer.

The Annotations menu displays down the left side of the Viewer.

![Annotations Menu](image)

**Note:** Annotations on sequences and shots are only visible when you have the Annotations menu open.

Adding Annotations

To add an annotation, do the following:
1. After you have opened the Annotations menu, move the playhead on the timeline to where you want to add your annotation.
2. Click the + (addition) icon.
A dialog is displayed containing annotation options.

3. You can choose which level you want to add your annotation to, sequence or shot-level, by selecting either the **Sequence** or **Clip** radio buttons.

When you add an annotation to the clip level, a turquoise line is displayed within the top of the shot. When you apply an annotation to the sequence level, a turquoise box appears above the selected section on a separate **Annotations** track.

4. From the **Range** dropdown select one of the following:
   - **Current Item** - Applies the annotation to the shot at the current playhead position.
   - **Current Frame** - Applies the annotation to the frame at the current playhead position.
   - **In/Out Points** - Applies the annotation to in and out points that have already been marked on the timeline.
   - **All** - Applies the annotation to the whole track.
   - **Custom** - When you select this from the **Range** dropdown, the in and out point fields within the dialog, become active. You can then use these to set the section that you want your annotation to appear on.

5. When you have set where you want your annotation to appear, click **New**.
   A label is added to the location of your annotations, detailing the clip timecode and name.

You can click and drag this label to place it anywhere in the Viewer.

6. To draw in your annotation, select the paint brush tool from the Annotation menu.
   This is highlighted orange when selected.

7. Before drawing in the Viewer, you can set the brush and paint settings:
   - Select the brush and/or background color by clicking the paint colors icon.

**Note:** The timecode displayed in the In and Out fields is derived from the clip's metadata, not its position in the sequence.
This opens a color wheel that allows you to select the color and brightness, and an opacity slider underneath that you can use to set the opacity of the paint.

• Select the paint brush size icon to set the required brush size. You can either use the slider to drag to your required brush size, or enter it into the brush size field.

**Note:** You can also edit these settings after drawing by clicking the selection tool in the Annotations menu, selecting the lines that you've drawn in the Viewer, and then adjusting the paint brush settings.

8. Click and drag in the Viewer to draw with your selected brush settings.

9. To add text to your annotation, click the text icon in the Annotations menu and then click anywhere in Viewer to enter your required text.

   A text dialog appears allowing you to type your required text, align it horizontally and vertically, and adjust the text size. You can then click and drag the text box to anywhere in the Viewer.
Enabling and Disabling Annotations

You can choose to disable and re-enable annotations from the output. To enable or disable a sequence-level annotation, simply select the annotation by clicking on the turquoise box and then pressing D. To enable or disable a shot-level annotation, you first need to open the shot – that includes the annotation – in the timeline view. Then you can select the annotation and press D.

Editing Sequence-Level Annotations

You can choose to edit, copy, move, and delete any annotations that were added on the sequence-level.

To remove an annotation from the sequence-level, simply select the turquoise box representing the annotation you want to move from the Annotation track, right-click and select Edit > Delete (or press Backspace or Delete on the keyboard).

To copy an annotation that was added at sequence level:
1. Select the annotation and then click Edit > Copy (or press Ctrl/Cmd+C). You can also access these tools in the right-click menu.
2. Move the timeline playhead to the position where you want to paste the annotation.

Note: You can also copy and paste annotations, that were added at sequence-level, between different sequences.

To move an annotation that was added at sequence level, you can simply click and drag it to the required location. You can also drag annotations to different track levels.

You can trim an annotation at either end by hovering the cursor over one end of the annotation until it changes into the trim icon:

Then click and drag to where you want to trim the annotation to.

To edit the actual annotation, you can use the selection tool in the Annotations menu to select the annotation in the Viewer and move it, delete it, or replace it.
Editing Shot-Level Annotations

You can choose to move, trim, and simply delete shot-level annotations.

To edit a shot-level annotation, you first need to open the annotated shot as a timeline. To do this:

1. Right-click the shot with the annotation you want edit.
2. Select **Open In > Timeline View**.

   The shot opens in the Timeline View, and the annotation now appears as on a separate level from the video. To view the annotation in the Viewer, ensure the Annotation menu is open.

To move a shot-level annotation from the Timeline view, hover the cursor over the annotation until it changes into a move icon:

Then simply click and drag it to where you want to move it to.

You can trim an annotation at either end by hovering the cursor over one end of the annotation until it changes into the trim icon:

Then click and drag to where you want to trim the annotation to.

To delete the annotation, click the turquoise box and select **Edit > Delete** (or press **Backspace** or **Delete** on the keyboard).

To edit the actual annotation, you can use the selection tool in the Annotations menu to select the annotation in the Viewer and move it, delete it, or replace it.
Viewing Annotations in the Compositing Environment

If you want to view annotations in the Compositing environment, you need to ensure annotations are enabled when creating a Nuke Comp. You can do this by either opening the Export dialog or the Project Settings dialog and checking the option Basic Nuke Shot With Annotations is selected. See Create Comp for more information. You can also enable annotations using the Create Comp Special option, as shown below.

To create a Nuke Comp with annotations, do the following:

1. Create a Nuke Comp by right-clicking on the shot that you want use to create a Nuke Comp, and selecting Effects > Create Comp Special.
   The Create Comp Special dialog opens.

2. In the Using Local export preset setting in the middle-top of the dialog, ensure Basic Nuke Shot With Annotations is selected. Also, in the Tracks For This Export section in the bottom-left of the dialog, ensure either all tracks is selected, or that the Annotations checkbox is selected with certain tracks.

3. Click Create Comp.
   A message warns that you've changed the export templates and asks whether you want to keep them.

4. Click Yes or No.
   This creates a Nuke Comp above the selected shot(s). The Nuke Comp is displayed in light red, as it has not been rendered yet. See Create Comp for more information on Nuke Comp colors.

5. Open the Nuke Comp by double-clicking it.
   When the Nuke Comp script opens, an additional Precomp node is created, which is separate from the Nuke Comp script. The Precomp node contains a copy of the whole script and includes the annotations.

6. Connect the Viewer node to the Precomp node.
   Your annotation is visible in the Viewer over the footage.

Viewing the Annotation Node Group

If you wish, you can view the annotations by doing the following:

1. Double-click the Precomp node to open its properties panel, and the click Open.
   This opens the contents of the Precomp node as a node tree in a new instance of Nuke.

2. Ensure you have the Node Graph tab selected.
The node tree displays in two parts. The top part represents the shot-level settings and the bottom part represents the sequence-level settings. Depending on where you set your annotation to be, an Annotations node group is displayed in the node tree.

3. Double-click the Annotations node group to open its properties panel.
4. In the top-right of the node properties, click the node structure icon to display the contents of the node group.
   This displays the contents of the Annotations node group in a new Node Graph tab.

Re-Exporting Annotations from the Timeline

You may want to add new annotations after the Nuke Comp has been edited and rendered. In this case, you can re-export the annotations only.

To re-export annotations, do the following:
1. Move the playhead to the rendered Nuke Comp.
2. Press the + in the Annotations menu at the side of the Viewer, and select the Clip radio button.

   **Tip:** You cannot add sequence-level annotations to a Nuke Comp.

3. Add an annotation using the brush or text tools in the Annotation menu.
4. Right-click the Nuke Comp, and select Export > Re-Export Annotations.
5. Return to the Nuke script and double-click the Precomp node to open its properties, and version it up.
   For example, if the file path has v01.nk at the end, change it to v02.nk. You can also version up the Precomp node by doing one of the following:
   • Select the Precomp node and press Alt+Up Arrow.
   • Select the Precomp node and click Edit > Node > Filename > Version Up.

6. Ensure the Precomp node is connected to a Viewer node.
   Your new annotation is now visible in the Viewer.
Timeline Editing Tools

The timeline editing tools allow you to manipulate your shots directly in the timeline, in single- or multi-view projects, using a series of modal editorial tools that complement the Multi Tool. Select the tool you need for the job and then select a new tool and continue editing.

The timeline editing tools are grouped for convenience - each tool group contains several tools and you can cycle between them by clicking the tool or using keyboard shortcuts. The editing tools work the same way in single- and multi-view timelines.

<table>
<thead>
<tr>
<th>Icon</th>
<th>Tools</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Multi Tool Icon" /></td>
<td>Multi Tool</td>
<td>The Multi Tool's functionality is equivalent to most of the other tools combined, but doesn't require modal tool selection.</td>
</tr>
<tr>
<td><img src="image" alt="Move/Trim Icon" /></td>
<td>Move/Trim</td>
<td>The Move/Trim tool allows you to manipulate the position of a shot or its output by adding or removing handles.</td>
</tr>
<tr>
<td><img src="image" alt="Select Icon" /></td>
<td>Select</td>
<td>The marquee Select tool allows you to make multiple selections quickly by lassoing shots. Hold Shift to add to the selection and Alt to subtract from the selection.</td>
</tr>
<tr>
<td><img src="image" alt="Selection by Track Icon" /></td>
<td>Selection by Track</td>
<td>The track selection tools allow you to quickly select multiple items depending on the initial selection. For example, the Select Track to Right tool selects all shots to the right of the target shot, within a single track.</td>
</tr>
<tr>
<td><img src="image" alt="Slip Clip Icon" /></td>
<td>Slip Clip</td>
<td>The Slip Clip tool allows you to shift a shot's In and Out points by the same amount and in the same direction, retaining the original duration but altering the timeline output.</td>
</tr>
<tr>
<td><img src="image" alt="Slide Clip Icon" /></td>
<td>Slide Clip</td>
<td>The Slide Clip tool allows you to move a shot in relation to the item before and/or after the target item, without changing its length or timeline output.</td>
</tr>
<tr>
<td><img src="image" alt="Roll Edit Icon" /></td>
<td>Roll Edit</td>
<td>The Roll Edit tool enables you to roll a single edit within the available handles, shortening one shot while lengthening the other, but keeping the overall duration the same.</td>
</tr>
<tr>
<td>Icon</td>
<td>Tools</td>
<td>Description</td>
</tr>
<tr>
<td>------</td>
<td>-------</td>
<td>-------------</td>
</tr>
<tr>
<td><img src="image" alt="Ripple Edit Icon" /></td>
<td>Ripple Edit</td>
<td>The Ripple Edit tool operates similarly to the trim function of the Move/Trim tool, except that downstream shots are rippled to automatically close any resulting gaps in the timeline.</td>
</tr>
<tr>
<td><img src="image" alt="Retime Clip Icon" /></td>
<td>Retime Clip</td>
<td>The Retime Clip tool allows you to trim a shot’s In or Out point and automatically retime the clip to fill the new shot duration.</td>
</tr>
<tr>
<td><img src="image" alt="Razor Icon" /></td>
<td>Razor</td>
<td>The Razor and Razor All tools allow you to cut shots in to separate parts so you can remove sections or rearrange items on the timeline.</td>
</tr>
<tr>
<td><img src="image" alt="Join Icon" /></td>
<td>Join</td>
<td>The Join tool can only be used on edit points between two razored shots, denoted by the yellow arrows at the edit.</td>
</tr>
</tbody>
</table>

**Tip:** Enabling Preferences > Panels > Timeline > show frame end marker draws an extra line on the timeline to the right of the playhead, indicating the end of the current frame.

The modal editorial tools are mapped to the Q, W, E, R, and T keyboard shortcuts when the timeline is the active tab.

**Note:** For a full list of keyboard shortcuts, please see Appendix B: Keyboard Shortcuts.

Pressing a keyboard shortcut multiple times selects the tools within each mode. For example, pressing E twice, rapidly in succession activates Slide Clip. Pressing them slowly in succession does
not achieve the same result, but instead, remains on the first item in the menu. This allows you to activate a tool without knowing the current state of tool selection.
• mapped to Q, cycles through Multi Tool, Move/Trim, and Select.
• mapped to W, cycles through Track Selection tools.
• mapped to E, cycles through Slip Clip and Slide Clip.
• mapped to R, cycles through Roll Edit, Ripple Edit, and Retime Clip.
• mapped to T, cycles through Razor, Razor All, and Join.
Using the Multi Tool

Unlike the other editing tools available, the Multi Tool changes function depending on the position of your pointer in relation to the shot(s) selected.

The Multi Tool's functionality is equivalent to most of the other tools combined, but doesn't require modal tool selection:

- **Move** - placing the mouse in the center of a shot activates the tool. Drag the selected shot to the required position on the timeline.
- **Trim** - placing the mouse at the left or right of the shot activates the tool. Drag the edit point to the new position and release the mouse to complete the trim.

See **Using the Move/Trim Tool** for more information.

- **Select** - click-and-drag to marquee select clips. Hold Shift to add to your selection or Alt to subtract.

See **Using the Selection Tools** for more information.

- **Slip** - placing the mouse at the bottom of the shot activates the tool and displays the slip handles. Drag the shot to the new position and release the mouse to complete the slip.

See **Using the Slip Clip Tool** for more information.

- **Slide** - placing the mouse at the top of the shot activates the tool and displays the slide handles. Drag the shot to the new position and release the mouse to complete the slide.

See **Using the Slide Clip Tool** for more information.

- **Roll** - placing the mouse on the edit between shots activates the tool and displays the handles. Drag the edit to the new position and release the mouse to complete the roll.

See **Using the Roll Edit Tool** for more information.

- **Razor** - when using the Multi Tool, Razor cuts are menu driven. Navigate to Timeline > Razor Selected or Razor All to make cuts at the current playhead position.
See Using the Razor and Join Tools for more information.
Using the Move/Trim Tool

The Move/Trim tool allows you to manipulate the position of a shot or its output by adding or removing handles. Activate the Move/Trim tool by clicking the tool or pressing Q twice.

Moving Shots

Click and drag the selected shot(s) to the required position on the timeline. A time count popup, in conjunction with the snap to edit function, helps you to reposition the shot(s) accurately.

You can also move shots up and down the track hierarchy using drag-and-drop or the Alt+, (comma) and Alt+, (period) keyboard shortcuts.
The following table describes the Move/Trim tool’s modifiers and actions:

<table>
<thead>
<tr>
<th>Mode</th>
<th>Method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Overwrite</td>
<td>drag-and-drop</td>
<td>The default move mode. The dragged shot overwrites any items that are present in the move location.</td>
</tr>
<tr>
<td>Ripple</td>
<td>drag then hold Alt and drop</td>
<td>Drag-and-drop shots on top of other items without overwriting content - items are pushed down the timeline to accommodate the move.</td>
</tr>
<tr>
<td>Duplicate</td>
<td>hold Alt and drag then release Alt and drop</td>
<td>Copy the shot, then drag-and-drop on top of other items overwriting existing content - items are not pushed down the timeline to accommodate the move.</td>
</tr>
<tr>
<td>Ripple and Duplicate</td>
<td>hold Alt then drag and drop while holding Alt</td>
<td>Copy the shot, then drag-and-drop items on top of others without overwriting content - items are pushed down the timeline to accommodate the move.</td>
</tr>
</tbody>
</table>

Note: On Linux, hold Ctrl+Alt for Duplicate and Ripple Duplicate modifiers.
<table>
<thead>
<tr>
<th>Action</th>
<th>Keyboard Shortcut</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delete</td>
<td>Backspace</td>
<td>Delete the selected shot(s) or gap(s).</td>
</tr>
<tr>
<td>Ripple Delete</td>
<td>Shift + Backspace</td>
<td>Remove the selected shot(s) and ripple items down stream to close gaps in the timeline.</td>
</tr>
</tbody>
</table>

**Note:** The ripple effect may not close gaps entirely, because Nuke Studio does not allow linked tracks to become desynchronized during rippling.

If you need to nudge shots horizontally by just a frame or two, you can select the items on the timeline and press ``, (comma) to nudge it left or `. (period)` to nudge it right. Press `Shift`, (comma) or `. (period)` to nudge the shot horizontally by the FrameIncrement set under the Viewer.

**Note:** You cannot overwrite other shots on the timeline horizontally using the nudge keys. However, vertical nudging (`Alt`+, and `Alt`+) overwrites other tracks.

### To Move Shots Using the Spreadsheet View:

1. Select the required events in the spreadsheet view.
2. Click the cog icon and select the required **Time Edit Behaviors**:

<table>
<thead>
<tr>
<th>Modify</th>
<th>Using</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dst In</td>
<td>Move Destination</td>
<td>Adjusts the event's <strong>Dst In</strong> and <strong>Out</strong> by the same amount, moving the shot's position on the timeline by the specified amount, while maintaining Speed. Before and after a 2 second <strong>Dst In</strong> increase:</td>
</tr>
<tr>
<td></td>
<td></td>
<td><img src="image" alt="Before and after a 2 second Dst In increase" /></td>
</tr>
<tr>
<td>Dst Out</td>
<td>Move</td>
<td>Adjusts the event's <strong>Dst Out</strong> and <strong>In</strong> by the same amount, moving the</td>
</tr>
<tr>
<td>Modify</td>
<td>Using</td>
<td>Result</td>
</tr>
<tr>
<td>--------</td>
<td>---------------</td>
<td>------------------------------------------------------------------------</td>
</tr>
<tr>
<td></td>
<td>Destination</td>
<td>shot’s position on the timeline by the specified amount, while</td>
</tr>
<tr>
<td></td>
<td></td>
<td>maintaining Speed.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Before and after a 2 second <strong>Dst Out</strong> increase:</td>
</tr>
</tbody>
</table>

3. Adjust the **Dst In** or **DstOut** to move the shot(s) by the specified amount.

**Trimming Shots**

Click-and-drag the edit point to the new position and release the mouse to complete the trim.

**Tip:** Use the **Ripple Edit** tool, activated by pressing **R** twice, to ripple downstream shots automatically.
The Viewer displays the new In or Out point (depending on whether you're adjusting the beginning or end of the shot), allowing you to accurately gauge the new output.

**Note:** Trimming multiple shots simultaneously trims each item by the same amount and in the same direction.

Alternatively, click an edit point and nudge the edit using the Ctrl/Cmd+\(\rightarrow\) (numeric pad) keys or hold Shift to nudge by the Frame Increment set under the Viewer.

By holding Ctrl/Cmd and dragging an edit, you can add blank frames past the end of the shot's handles. Blank frames are colored red on the timeline for clarity:
To Trim Shots Using the Spreadsheet View:

1. Select the required events in the spreadsheet view.
2. Click the cog icon and select the required Time Edit Behaviors depending on whether you're using the In or Out points or duration:

<table>
<thead>
<tr>
<th>Modify</th>
<th>Using</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Src In</td>
<td>Trim In</td>
<td>Trims the event’s <strong>Src In</strong>, <strong>Dst In</strong>, and <strong>durations</strong> while maintaining speed. Before and after a 2 second <strong>Src In</strong> increase:</td>
</tr>
<tr>
<td></td>
<td></td>
<td><img src="image1" alt="Before and after Src In increase" /></td>
</tr>
<tr>
<td>Src Out</td>
<td>Trim Out</td>
<td>Trims the event’s <strong>Src Out</strong>, <strong>Dst Out</strong>, and <strong>durations</strong> while maintaining speed. Before and after a 2 second <strong>Src Out</strong> increase:</td>
</tr>
<tr>
<td></td>
<td></td>
<td><img src="image2" alt="Before and after Src Out increase" /></td>
</tr>
<tr>
<td>Modify</td>
<td>Using</td>
<td>Result</td>
</tr>
<tr>
<td>--------</td>
<td>--------</td>
<td>--------</td>
</tr>
<tr>
<td>Src Dur</td>
<td>Trim Out</td>
<td>Trims the event’s Src Dur, Dst Dur, and Src/Dst Out while maintaining speed.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Before and after a 50 frame Src Dur increase:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>![Table showing changes before and after a 50 frame Src Dur increase]</td>
</tr>
<tr>
<td>Dst In</td>
<td>Trim In</td>
<td>Trims the event’s Dst In, Src In, and durations while maintaining speed.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Before and after a 2 second Dst In increase:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>![Table showing changes before and after a 2 second Dst In increase]</td>
</tr>
<tr>
<td>Dst Out</td>
<td>Trim Out</td>
<td>Trims the event’s Dst Out, Src Out, and durations while maintaining speed.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Before and after a 2 second Dst Out increase:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>![Table showing changes before and after a 2 second Dst Out increase]</td>
</tr>
<tr>
<td>Dst Dur</td>
<td>Trim Out</td>
<td>Trims the event’s Dst Dur, Src Dur, and Dst/Src Out while maintaining speed.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Before and after a 50 frame Dst Dur increase:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>![Table showing changes before and after a 50 frame Dst Dur increase]</td>
</tr>
</tbody>
</table>
3. Adjust the values as required to trim the shot(s) by the specified amount.
Using the Selection Tools

The timeline editing tools include a marquee selection tool and several context dependent track selection tools.

The marquee Select tool, activated by clicking the tool or pressing Q three times, allows you to make multiple selections quickly by lassoing shots.

Hold Shift to add to the selection:

The track selection tools, activated by clicking the tool or by pressing W, selects multiple items depending on the initial selection:

- **Select Track to Right** or **Left** - all shots right or left of the target shot are selected, within a single track.
• **Select All in Track** - all shots on the target track are selected, regardless of the item selected.

• **Select All Tracks Right or Left** - all shots right or left of the target item are selected, regardless of which track they occupy.
Using the Slip Clip Tool

The **Slip Clip** tool allows you to shift a shot’s In and Out points by the same amount and in the same direction, retaining the original duration but altering the timeline output. Activate the **Slip Clip** tool by clicking the tool or pressing **E**.

**Note:** The target shot must have handles to use the **Slip** tool.

The **Slip Clip** tool displays different Viewer previews depending on whether the playhead is on the target shot or not, but the basic principles are the same.

Click the target clip to display the available handles and then drag the shot to the new position. Release the mouse to complete the slip.

![Slip Clip Tool](image)

**Note:** Using the **Slip Clip** tool does not move the shot on the timeline, only the output is changed.

Alternatively, nudge the slip using the `, (comma) or . (period) keys or hold **Shift** to nudge by the **Frame Increment** set under the Viewer.

**Tip:** If you’re using the **Multi Tool**, you can nudge using the “slip bar” by clicking at the bottom the shot.

The following Viewer previews are displayed, depending on the playhead position:

**Note:** The Viewer background always displays the playhead’s current position.
• When the playhead is not currently on the target shot, the Viewer displays the **In** frame (1) and **Out** frame (2), allowing you to accurately gauge the new output.

• When the playhead is on the target shot, the Viewer displays the **In** frame (1), the **current** frame (2), and **Out** frame (3), allowing you to accurately gauge the output of the shot against the current frame.

• When the playhead is on the target shot and A/B compare is active, the Viewer displays the target shot (1) and the reference shot (2), allowing you to synchronize your working track against the reference track.

Slipping Using the Spreadsheet View

You can slip shots using the **Src In** and **Src Out** columns of the spreadsheet:

1. Select the required event in the spreadsheet view.
2. Click the cog icon and select the required **Time Edit Behaviors** depending on whether you’re using the In or Out point:

<table>
<thead>
<tr>
<th>Modify</th>
<th>Using</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Src In</td>
<td>Slip Source</td>
<td>Adjusts the <strong>Src In</strong> and <strong>Src Out</strong> by the same amount, slipping the event while maintaining speed. Before and after a 2 second <strong>Src In</strong> increase:</td>
</tr>
<tr>
<td></td>
<td></td>
<td><img src="image1" alt="Graph" /></td>
</tr>
<tr>
<td>Src Out</td>
<td>Slip Source</td>
<td>Adjusts the <strong>Src Out</strong> and <strong>Src In</strong> by the same amount, slipping the event while maintaining speed. Before and after a 2 second <strong>Src Out</strong> increase:</td>
</tr>
<tr>
<td></td>
<td></td>
<td><img src="image2" alt="Graph" /></td>
</tr>
</tbody>
</table>

3. Adjust the **Src In** or **Src Out** to slip the shot(s) by the specified amount.
Using the Slide Clip Tool

The Slide Clip tool allows you to move a shot in relation to the item before and/or after the target item, without changing its length or timeline output. Activate the Slide Clip tool by clicking the tool or pressing E twice.

The shot either side of the target are shortened or lengthened within the limits of their handles to accommodate the slide.

**:Note:** The surrounding shots must have handles to use the Slide tool.

Click the target shot and then drag it to the new position and release the mouse to complete the slide.

For example, if you slide the target shot (2) five frames to the right, the preceding item (1) ends five frames later and the next item (3) starts five frames later.

The first image shows a timeline containing three shot, and the second shows the same shots with the target (2) sliding to the right.
The Viewer displays the new end point of the previous shot on the left and the new start point of the next shot on the right, allowing you to accurately gauge the slide.

The two center images (2) represent the start and end frames of the target shot, which don't change.
Using the Roll Edit Tool

The **Roll Edit** tool enables you to roll a single edit within the available handles, shortening one shot while lengthening the other, but keeping the overall duration the same. Activate the **Roll Edit** tool by clicking the tool or pressing **R**.

**Note:** At least one of the target items must have handles to use the **Roll** tool.

1. Click an edit point between two shots to display the available handles as a red overlay.
2. Drag the edit to the new position and release the mouse to complete the roll.

For example, if you roll a number of frames at the end of one shot (1), the next item (2) starts that number of frames later. The first image shows a timeline containing two shots, and the second shows the same items with the edit point “rolled” to the right.
The Viewer displays the pre-edit shot on the left and the post-edit item on the right, allowing you to accurately gauge the new position of the edit.

Alternatively, click the edit point between the shot and nudge the edit using the , (comma) or . (period) keys or hold Shift to nudge by the Frame Increment set under the Viewer.
Using the Retime Clip Tool

The Retime Clip tool allows you to trim a shot's In or Out point and automatically retime the clip to fill the new shot duration. Activate the Retime Clip tool by clicking the tool or pressing R three times.

Click-and-drag the edit point to the new position and release to complete the trim and retime. For example, trimming a 50 frame shot to 25 frames retimes the clip to 200%.

Alternatively, click an edit point and nudge the edit using the , (comma) or . (period) keys or hold Shift to nudge by the Frame Increment set under the Viewer.
Tip: By holding \texttt{Ctrl.Cmd} and dragging an edit, you can retime past the end of the shot's handles.
Using the Razor and Join Tools

The **Razor** tools allow you to cut shots in to separate parts so you can remove sections or rearrange items on the timeline. Activate **Razor** and **Razor All** by clicking the tool or pressing **R**.

Place the cursor on the target shot, and if the cut is permissible, click to razor the shot or all shots depending on which tool you have selected.

**Tip:** The Razor cursor indicates whether a cut is permissible or not, such as on existing edits.

You can also apply cuts at the playhead position from the menu bar using **Timeline > Razor Selected**, or all tracks using **Timeline > Razor All**.

**Tip:** Use the **C** (with the shot under the playhead selected) and **Shift+C** keyboard shortcut, or the right-click context menu, to perform the respective cuts.

The **Join** tool can only be used on edit points between razored shots, denoted by the yellow arrows at the edit.
Copying Cuts Between Tracks

The **Copy Cuts** function allows you to quickly apply cuts from one track to other tracks on the timeline. For example, in the timeline shown, you could copy the cuts from the second video track to the Reference audio tracks.

To copy cuts:

1. Select the shots containing the cuts to copy, or if you intend to copy all the cuts from a track you don't need to make a selection.

2. Right-click in the timeline and select **Editorial > Copy Cuts**.

The **Copy Cuts** dialog displays.
3. If you made a selection on the timeline, use the dropdown to select **Copy All Cuts** or **Copy Selected Cuts** as required.

   This dropdown is not displayed if no shots were selected.

4. Click the **From** dropdown to select the source track.

5. Check all the destination tracks in the **To** field to which you want to copy the cuts.

6. Choose whether or not the resulting shots are named identically to the source track.

   Selecting **None** retains the destination clip name.

7. Click **OK** to copy the cuts to the destination track(s).
Insert, Overwrite, and 3-Point Editing

Insert and Overwrite edits are applied at the current playhead position by default, but the use of In and Out points in the clip Viewer and/or sequence Viewer can give you greater control over the result. 3-point editing, makes use of In and Out points in the clip Viewer and an In or Out in the sequence Viewer to control where the clip is placed on the timeline.

Inserting Clips

By default, Insert places the entire contents of the clip Viewer into the timeline at the current playhead position, on the lowest available track. All shots downstream of the playhead are rippled to make room for the clip. No items are overwritten or removed.

![Pre-insert timeline](image1) ![Post-insert timeline](image2)

Note: If the playhead is not positioned at an edit point, or there are shots on other tracks, the Insert action cuts the shot(s) at the playhead and ripples the cut downstream. For example, the Post-insert image shows the audio shot being cut and rippled, even though it doesn't reside on the same track.

You can select a track before inserting if you don't want to target the lowest available track. Even if the target track is empty, shots on all other unlocked tracks are rippled by the same amount.

You can also use In and Out points to control where the clip is inserted and how many frames are included. See 3-Point Editing for more information.

To insert a clip at the playhead:
1. Navigate to Workspace > Editing to display the 2-up Viewer layout.
2. Double-click your sequence in the bin view to load it into the right-hand sequence Viewer.

3. Double-click the source clip to load it into the left-hand clip Viewer.

4. Place the playhead at the required edit point or timecode and select the target track, if necessary.

5. Navigate to Clip > Insert, or press N, to insert the clip into the timeline.
   
   All shots downstream of the clip are rippled to make room for the duration of the edit.

To insert a clip at an In or Out point:

1. Navigate to Workspace > Editing to display the 2-up Viewer layout.

2. Double-click your sequence in the bin view to load it into the right-hand sequence Viewer.

3. Double-click the source clip to load it into the left-hand clip Viewer.

4. Place an In or Out point on the timeline to determine the clip's position:
   
   - **In point** - the source clip is inserted so that the first frame is at the In point specified.
   
   - **Out point** - the source clip is inserted so that the last frame is at the Out point specified.

5. Navigate to Clip > Insert, or press N, to insert the clip into the timeline.
   
   All shots downstream of the In or Out point are rippled to make room for the duration of the edit.

### Overwrite Edits

Unlike inserting, **Overwrite** does not incorporate downstream ripple and doesn't alter the length of your sequence. Any shots you overwrite are destroyed, though they can easily be recovered from the source clips in the bin view.

![Pre-overwrite timeline](image1.png) ![Post-overwrite timeline](image2.png)

You can select a track before overwriting if you don't want to target the lowest available track.

You can also use In and Out points to control what the clip overwrites and how many frames are included. See 3-Point Editing for more information.

To overwrite at the playhead:

1. Navigate to Workspace > Editing to display the 2-up Viewer layout.

2. Double-click your sequence in the bin view to load it into the right-hand sequence Viewer.
3. Double-click the source clip to load it into the left-hand clip Viewer.
4. Place the playhead at the required edit point or timecode and select the target track, if necessary.
5. Navigate to **Clip** > **Overwrite**, or press **M**, to overwrite from the playhead for the duration of the source clip.
   All shots under the source clip are overwritten.

To overwrite from an In or Out point:
1. Navigate to **Workspace** > **Editing** to display the 2-up Viewer layout.
2. Double-click your sequence in the bin view to load it into the right-hand sequence Viewer.
3. Double-click the source clip to load it into the left-hand clip Viewer.
4. Place an In or Out point on the timeline to determine the clip's behavior:
   - **In point** - the source clip begins overwriting from its first frame at the In point specified downstream for the duration of the clip.
   - **Out point** - the source clip begins overwriting from its last frame at the Out point specified upstream for the duration of the clip.
5. Navigate to **Clip** > **Overwrite**, or press **M**, to overwrite from the In or Out point for the duration of the source clip.
   All shots under the source clip are overwritten.

### 3-Point Editing

Setting the output of a source clip and then editing the clip into a timeline at a specific point is sometimes referred to as 3-point editing. Using this method, you can insert and overwrite edits in an existing timeline or quickly construct scratch timelines from your source clips.

Firstly, set the output of your source clip using In and Out points in a clip Viewer, then set the reference In or Out point on your timeline to determine the clip's position. Finally, add the clip to the timeline using **Insert** or **Overwrite**.

**Tip:** You can set both In and Out points on the timeline, but bear in mind that there may be insufficient source frames for the range specified. If this is the case, blank frames are added and highlighted in red.

You can select a track before editing if you don't want to target the lowest available track. When inserting, even if the target track is empty, shots on all other unlocked tracks are rippled by the same amount.
1. Navigate to **Workspace** > **Editing** to display the 2-up Viewer layout.
2. Double-click the required source clip to load it into the left-hand clip Viewer.

3. Set the required frame range using In and Out points.

4. Double-click your sequence in the bin view to load it into the right-hand sequence Viewer.

5. Set In and/or Out points on the timeline to specify where the clip should be added and use **Insert** (N) or **Overwrite** (M) as required.

   As an example, assuming your clip Viewer and timeline are represented by the following image, and the Overwrite function is used:

   - **No In or Out points** - insert or overwrite at the current playhead position, for the range currently set in the clip Viewer.
• **In point but no Out point** - insert or overwrite from the In point position downstream, for the range currently set in the clip Viewer.

• **Out point but no In point** - insert or overwrite from the Out point position upstream, for the range currently set in the clip Viewer.
• **In and Out points** - insert or overwrite at the current In point position, for the duration set by the timeline's In and Out points. If there are insufficient source frames for the range specified, blank frames are added highlighted in red.
Versions and Snapshots

In addition to the regular project save and restore options, Nuke Studio can record the different states of your workflow as you progress using versions and snapshots.

**Versions** are children of clips. You can have any number of versions per clip as long as they follow the correct naming conventions, as shown in *Using Versions*. Versions can only be applied to source clips and shots and can be swapped in and out without overwriting existing work.

**Snapshots** are time-stamped copies of a sequence, allowing you to save its current state without the inconvenience of saving the entire project. When you restore a snapshot, a warning displays prior to the restore reminding you that edits since the snapshot was taken are lost. See *Using Snapshots* for more information.
Using Versions

Versions can be added to source clips and shots to allow greater flexibility in your workflow. You can have as many versions as required and cycle through them quickly using keyboard shortcuts.

**Note:** You cannot use versions when a clip is opened as a timeline, that is, by using the right-click **Open In > Timeline View** option.

The application relies on specific file naming or folder structure conventions to discover versions:

<table>
<thead>
<tr>
<th>Convention</th>
<th>Description</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>File name constants</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Clip name</td>
<td>The file name base must remain the same.</td>
<td>myClip_v1.0001.dpx</td>
</tr>
<tr>
<td></td>
<td></td>
<td>myClip_v2.0001.dpx</td>
</tr>
<tr>
<td></td>
<td></td>
<td>myClip_v3.0001.dpx</td>
</tr>
<tr>
<td>Version prefix</td>
<td>The delineation between the file name and version information must be either _ (underscore) or . (period) and remain the same for all versions.</td>
<td>myClip_v1.0001.dpx</td>
</tr>
<tr>
<td></td>
<td></td>
<td>myClip_v2.0001.dpx</td>
</tr>
<tr>
<td></td>
<td></td>
<td>myClip_v3.0001.dpx</td>
</tr>
<tr>
<td>File name variables</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Version padding</td>
<td>The version number padding in the clip name can be increased or decreased.</td>
<td>myClip_v1.0001.dpx</td>
</tr>
<tr>
<td></td>
<td></td>
<td>myClip_v002.0001.dpx</td>
</tr>
<tr>
<td></td>
<td></td>
<td>myClip_v03.0001.dpx</td>
</tr>
<tr>
<td>Frame padding</td>
<td>The frame padding in the clip name can be increased or decreased.</td>
<td>myClip_v1.01.dpx</td>
</tr>
<tr>
<td></td>
<td></td>
<td>myClip_v1.1.dpx</td>
</tr>
<tr>
<td></td>
<td></td>
<td>myClip_v1.0001.dpx</td>
</tr>
<tr>
<td>Extension</td>
<td>The file format is interchangeable. See <strong>Appendix C: Supported File Formats</strong> for</td>
<td>myClip_v1.01.png</td>
</tr>
<tr>
<td></td>
<td></td>
<td>myClip_v1.0001.dpx</td>
</tr>
<tr>
<td>Convention</td>
<td>Description</td>
<td>Example</td>
</tr>
<tr>
<td>---------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>----------------------------------</td>
</tr>
<tr>
<td>more information.</td>
<td></td>
<td>myClip_v1.mov</td>
</tr>
</tbody>
</table>

**Note:** If the file extension is a movie format, such as `.r3d` or `.mov`, the **Frame padding** can be omitted.

### Folder name constants

<table>
<thead>
<tr>
<th>Root folder</th>
<th>The root folder name must remain the same for all folders containing versions.</th>
<th>~/version/v1/myClip_v1.0001.dpx</th>
<th>~/version/v2/myClip_v2.0001.dpx</th>
<th>~/version/v3/myClip_v3.0001.dpx</th>
</tr>
</thead>
</table>

### Folder name variables

<table>
<thead>
<tr>
<th>Version padding</th>
<th>The version number padding in the folder name can be increased or decreased.</th>
<th>09_WF_Shot004_v1</th>
<th>09_WF_Shot004_v002</th>
<th>09_WF_Shot004_v03</th>
</tr>
</thead>
</table>
Versions in Bins

Versions behave similarly in both bins and sequences, and in both cases, you first have to ingest an existing version.

Ingest and locate the versioned clip, then:

1. Right-click and select **Version > Scan for Versions** to search for available versions.
   A dialog box lets you know how many versions were discovered.

2. Use the right-click **Version** menu to:
   • Go to the next **Version Up** or **Version Down**.
   • Go to the **Minimum** or **Maximum Versions**.

   **Tip:** You can also use the **Alt+Up/Down Arrow** keyboard shortcuts to increment versions or **Alt+Shift+Up/Down Arrow** to go to the maximum or minimum.

When you reach the end of the discovered versions, incrementing the version automatically scans for new versions that may have become available.

3. For source clips only, you can right-click the clip and select **Open In > Versions Bin** to display all discovered versions of the target clip.

   The versioning conventions may allow clips into the Version Bin that you weren’t expecting. You can disable versions by selecting them and pressing **D** or by selecting the **Set Active Version** of a clip using the right-click **Version** menu.

   The **Active Version** is the version displayed when you drag the source clip to the timeline, denoted by the orange **V** in the top left-hand corner of the thumbnail.
4. Once you've sorted all the available versions, select a clip in the bin view and press V to display all versions for that clip in a convenient window. Disabled versions are not displayed.

5. Select the required clip to set the **Active Version** and apply it to the clip.
Versions in Sequences

As mentioned previously, versions behave similarly in both bins and sequences, but swapping versions in sequences allows you to compare results more easily.

**Note:** You cannot use versions when a clip is opened as a sequence, that is, using the right-click **Open In > Timeline View** option.

Locate the ingested version clip and drag it to the timeline, right-click and select the **Version** menu:

- **Scan For Versions** to locate new versions of the clip.
- **Version Up** or **Version Down** to increment the version by one.
- Go to the **Minimum** or **Maximum Version**.

**Tip:** You can also use the **Alt + Up/Down Arrow** keyboard shortcuts to increment versions or **Alt+Shift+Up/Down Arrow** to go to the maximum or minimum.

Once you've scanned for versions, select a shot on the timeline and press **V** to display all available versions for that item in a convenient window.

Select the required shot version to set the **Active Version**.
Using Snapshots

Within a project you can save the current state of a sequence as a snapshot, including a comment or tag to describe that snapshot. You can see what snapshots exist for a sequence in the bin view and flip it back to any previously saved state.

An example workflow might appear as follows:

1. Two snapshots of the sequence (SQ) are recorded after edits. See Creating Snapshots.
2. Snapshot 1 is then restored. See Restoring Snapshots.
3. Further edits are made, then the sequence is recorded as snapshot 3.
Creating Snapshots

To create a snapshot for a sequence:

1. Locate the sequence in the bin view.
2. Right-click the sequence and select **Snapshot > Add Snapshot**.

The **Add new snapshot** dialog box displays.

3. Enter a comment, or use the default date and time supplied.
4. Click **Add** to create the snapshot.

Snapshots are indicated in the bin view with a camera icon containing the number of snapshots available.
Restoring Snapshots

To restore a snapshot:
1. Locate the sequence in the bin view.
2. Right-click the sequence and select **Snapshot > Restore Snapshot**.
3. Select the required snapshot from the list.
   A warning displays reminding you that edits since the snapshot was taken are lost.
4. Click **OK** to restore the sequence to the point at which the snap was recorded.
Exporting from Nuke Studio

This section deals primarily with shot management and export functionality when you're farming out shots or sequences to other artists. It also deals with the presets, which dictate how Create Comp passes data between the Timeline environment and Compositing environment.

The export suite can transcode, export clip selections from a timeline or bin, write out EDLs and XMLs, or bake out an entire timeline as a single clip in your required delivery format. The Export presets are also used to manage how Create Comp sends clips back and forth between Nuke Studio's Timeline environment and Compositing environment using Local and Project Presets.

Nuke Studio ships with several context-sensitive and ad hoc export options:

- **Exporting Sequences and Shots** - the process of preparing a sequence or individual shots for export and paving the way for VFX work to come back into Nuke Studio.
- **Transcoding** - converts one file format to another. You can transcode sequences, timeline selections, and clips from the bin view.
- **Ad Hoc Exports** - an umbrella covering exports that you might not perform on a per project basis, such as EDL or XML exports.
- **Create Comp** - edit or create the presets used during Create Comp, passing source clips or shots to the Compositing environment and sending rendered Write nodes from the Node Graph back to the Timeline environment.

With the addition of Python bindings to perform the same functions, this infrastructure provides a massive amount of flexibility, whether your pipeline is GUI or command line orientated.

**Note:** Nuke Studio is non-destructive and can slot into your pipeline if you setup your shot template to mirror the existing file structure.
Introduction to the Export Dialog

Nuke Studio uses presets and shot templates to perform export operations, including round-tripping and EDL/XML creation. The Export dialog controls what is exported and where, and whether or not to expect versioned clips as part of a round-trip from Nuke.

The Export dialog is accessed from the File menu, from the right-click bin and timeline menus, or by using the keyboard shortcut Ctrl/Cmd+Shift+E.

The Shot Template is also used to create the presets used during Create Comp, passing shots from the Compositing environment and sending rendered Write nodes from the Node Graph back to the Timeline environment.

Nuke Studio uses Content Presets in all shot templates, enabling you to create commonly used export conditions, which are then available across all projects. Some presets are only available with certain exporters, for example, the EDL Exporter preset cannot be used with Process as Shots exports.
You can filter your exports using the **Tracks for this Export** and **Filter by Tag** lists, exporting only certain tracks or shots marked with a particular tag. See **Using Tags** for more information.

For your convenience, Nuke Studio ships with a number of ready-made **Content Presets**, but you can edit these as required:

- **Transcode Images** - defines transcode parameters allowing you to save your most-used file type conversions.
- **Nuke Project File** - defines the script name and paths used by Nuke Read and Write nodes during a round-trip or **Create Comp**.
- **Nuke Annotations File** - defines the script name and paths used by Nuke Write nodes and Precomp group during a round-trip or **Create Comp**.
- **Render with** - selects how Nuke Studio renders your export: **Single Render Process** or **Frame Server**. This dropdown defaults to **Frame Server** using the number of slave processes specified in the **Preferences > Performance > Threads/Processes**. If you set this preference to **0**, Nuke Studio relies on external machines set up as render slaves. See **Using the Frame Server on External Machines** for more information.
- **EDL Exporter** - used to export a sequence to the EDL format.
- **SymLink Exporter** - creates symlinks to the location of the source files, rather than making copies.
- **XML Exporter** - used to export a sequence to XML format.
- **Copy Exporter** - creates copies of the source files to a specified location, rather than symlinking.
- **Audio Export** - copies any audio tracks to **.wav** files in a specified location.
Using Local and Project Presets

Presets are containers for export preferences, such as file structure and format, and filters for tracks, tags, and frame range. Two types of Presets are available to construct commonly used export tasks:

- **Local Presets** - these presets are used to set up round-trips between artists on different platforms and also to manage passing files between the Timeline and Compositing environment. See Create Comp for more details. Local Presets are saved in a Task Presets folder using the XML file format.

- **Project Presets** - you can drag-and-drop Local Presets into this panel to save the preset within a project .hrox file. This option is designed for collaborative work, allowing you to quickly share your export presets.

Using the Shot Template

The shot template sets up the folder hierarchy and naming conventions for export presets such as Basic Nuke Shot with Annotations and Transcode Clip DPX, and how Create Comp sends clips back and forth between the Timeline environment and Compositing environment. Any folders added to the template are created during export unless they already exist, in which case the export writes to the existing structure.

Nuke Studio ships with default templates for your convenience, but you can quickly create custom templates using folders and “tokens”, which are replaced with the relevant information during export.

**Tip:** Clicking an entry in the shot template displays a preview file path with the tokens resolved under the Version token number field.

Exports can resolve the following tokens:

<table>
<thead>
<tr>
<th>Token</th>
<th>Resolves to</th>
</tr>
</thead>
<tbody>
<tr>
<td>_nameindex</td>
<td>The index of the shot name in the sequence, preceded by _ (underscore), to avoid clashes with shots of the same name.</td>
</tr>
<tr>
<td>{ampm}</td>
<td>The local equivalent of either AM or PM.</td>
</tr>
<tr>
<td>{binpath}</td>
<td>The bin structure to preserve. Including this token recreates your bin structure up to the nearest parent bin.</td>
</tr>
<tr>
<td>Token</td>
<td>Resolves to</td>
</tr>
<tr>
<td>------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>{clip}</td>
<td>The name of the clip used in the shot processed.</td>
</tr>
<tr>
<td>{day}</td>
<td>The local weekday name, abbreviated to Mon, Tue, and so on.</td>
</tr>
<tr>
<td>{DD}</td>
<td>The day of the month as a decimal, 01, 02, and so on.</td>
</tr>
<tr>
<td>{event}</td>
<td>The timeline event number associated with the shot to process.</td>
</tr>
<tr>
<td>{ext}</td>
<td>The extension of the file to output, such as .dpx or .mov</td>
</tr>
<tr>
<td>{filebase}</td>
<td>The base of the clip name to process. For example, the filebase of Shot01_####.dpx is Shot01_.####.</td>
</tr>
<tr>
<td>{fileext}</td>
<td>The format of the clip to process, such as .dpx or .mov</td>
</tr>
<tr>
<td>{filehead}</td>
<td>The source clip filename not including frame padding or extension.</td>
</tr>
<tr>
<td></td>
<td>For example, the filehead of Shot01_####.dpx is Shot01.</td>
</tr>
<tr>
<td>{filename}</td>
<td>The source clip name of the media to process.</td>
</tr>
<tr>
<td>{filepadding}</td>
<td>The source filename padding, which you might use for formatting frame indices.</td>
</tr>
<tr>
<td>{filepath}</td>
<td>The full file path to the source media referenced in the export.</td>
</tr>
<tr>
<td>{fullbinpath}</td>
<td>The full bin structure to preserve. Including this token recreates the bin structure up to the project level.</td>
</tr>
<tr>
<td>{fullday}</td>
<td>The local full weekday name.</td>
</tr>
<tr>
<td>{fullmonth}</td>
<td>The local full month name.</td>
</tr>
<tr>
<td>{hierotemp}</td>
<td>The temp directory as specified in the Preferences.</td>
</tr>
<tr>
<td>{hour12}</td>
<td>The export start time hour component (12-hour clock).</td>
</tr>
<tr>
<td>{hour24}</td>
<td>The export start time hour component (24-hour clock).</td>
</tr>
<tr>
<td>{MM}</td>
<td>The month of the year as a decimal, 01, 02, and so on.</td>
</tr>
<tr>
<td>{minute}</td>
<td>The export start time minute component.</td>
</tr>
<tr>
<td>{month}</td>
<td>The local month name, abbreviated to Jan, Feb, and so on.</td>
</tr>
<tr>
<td>Token</td>
<td>Resolves to</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>{project}</td>
<td>The name of the parent project of the export item.</td>
</tr>
<tr>
<td>{projectroot}</td>
<td>The root export file path as specified in the Timeline Environment Project Settings.</td>
</tr>
<tr>
<td>{second}</td>
<td>The export start time second component.</td>
</tr>
<tr>
<td>{sequence}</td>
<td>The sequence name to process.</td>
</tr>
<tr>
<td>{shot}</td>
<td>The name of the shot to process.</td>
</tr>
<tr>
<td>{timestamp}</td>
<td>The export start time in the 24-hour clock format (HHMM).</td>
</tr>
<tr>
<td>{track}</td>
<td>The name of the track to process. Exporting EDLs using this token generates a separate EDL for each track.</td>
</tr>
<tr>
<td>{user}</td>
<td>The current username.</td>
</tr>
<tr>
<td>{version}</td>
<td>The string v#, defined by the number (#) set in the Version section of the export dialog</td>
</tr>
<tr>
<td>{YY}</td>
<td>The year of the century as a decimal, 01, 02, and so on.</td>
</tr>
<tr>
<td>{YYYY}</td>
<td>The year, including century.</td>
</tr>
</tbody>
</table>

**Tip:** Double-click the path column, right-click, and then choose Select Keyword to display a list of available export tokens, though only valid tokens for the current selection are listed.
Token substrings are valid if you need to extract a certain part of an evaluated string. For example, if `{shot}` resolves to JB100, then:

- `{shot [0:2] }` - resolves to JB
- `{shot [-3:] }` - resolves to 100

Similarly, anything within the outer brackets is evaluated as a Python string. For example, if `{shot}` resolves to JB_10_20, then:

- `{shot.split(’_’) [0] }` - resolves to JB
- `{shot.split(’_’) [2] }` - resolves to 20

### Custom Shot Templates

The shot template enables you to create as many Nuke Read and Write nodes as required for a project. A typical use case might be creating `.jpg` clips for review and `.dpx` resolution clips for finishing.

### Multi-format Exports

The following example describes how to build a shot template to export a sequence of `.mov` clips, create `.dpx` and `.jpg` Write nodes in Nuke, and bring the `.dpx` clips back into the timeline.

1. In the Export dialog, select Basic Nuke Shot in the Local Presets panel to auto-complete the shot template with the preset values.
2. Click Duplicate selected preset and give the new preset a name.
3. Rename the renders folder renders_dpx.
4. Select the nuke folder and click the folder icon to add a new folder. Name the new folder renders_jpg.
5. Select the renders_jpg folder and click to add a new entry.
6. Replace the `{filename}` token with `{shot}_comp{nameindex}_version.{##}.{ext}`, the same as the existing entry under renders_dpx.

**Note:** The `##` marks represent frame numbers for image sequences. If you were creating `.mov` clips, they'd be omitted.

The shot template should look something like this:
7. Click the **Content** column and select **Nuke Write Node**.

8. In the **Content** settings tab, use the **File Type** dropdown to select **jpeg**.

**Note:** When using a third party application to produce the VFX work, select **ExternalRender** instead of **Nuke Write Node**.
Notice that the settings available change depending on the File Type selected.

9. Click Nuke Project File in the shot template and check that both Write nodes are enabled.

10. Set up the rest of the export as described in Exporting Sequences and Shots and click Export.
Adding Burn-in Text to Exports

Nuke Studio can burn-in text during the export process using a simple Nuke gizmo. The gizmo is accessed from the **Nuke Write Node** preset’s **Content** panel under **Burn-in**.

The gizmo contains controls for the font style and fields denoting the position of the text. You can also add burn-in directly on timeline using the Burn-In soft effect. See **Soft Effects** for more information.

**Note:** The **Font** field only accepts the full file path and name of the font file to use. For example, on Mac `~/Library/Fonts/Verdana.ttf`

Click **Edit** to display the available controls.

You can mix-and-match the following methods to create burn-in text:

- Enter text manually, what you see is what you get in the burn-in.
- Use any of the tokens valid in the shot template as burn-in tokens. For example:
  
  `{shot}_comp`

  Extracts the shot name from the timeline and appends **_comp**.

  See **Using Local and Project Presets** for more information.

- Use metadata from tags applied to clips and shots. For example:
[metadata hiero/tags/Approved]

Extracts the Approved tag from the clip or shot. You can also append note to include any notes associated with the tag:

[metadata hiero/tags/Approved/note]

Note: You must precede spaces and slashes in the tag name with \ (backslashes) to enable Nuke Studio to process the tag name correctly. For example: [metadata hiero/tags/Blue\ Screen/note]

Tip: If you're not sure what metadata keys and values are available on a shot, you can add a Text soft effect containing the Tcl expression [metadata values] to display all metadata in the Viewer. See Soft Effects for more information.
Adding Additional Nodes During Export

Nuke Studio can include additional nodes, in any Nuke Project File or Transcode export in the Shot Template, by simply copying and pasting scripts from the Node Graph.

You can add nodes to shots, tracks, or sequences, or include them as unconnected ad hoc nodes in the script, filtered by tags if necessary.

1. In the Content tab, scroll down to the Additional Nodes control and click Edit.
   The Additional Nodes Setup dialog displays.

2. Click + to add an entry.

3. Click the Apply To field and select what the current entry applies to:
   - Shot - the additional nodes are added to the script for each shot in the export.
   - Track - the additional nodes are added to the script for each track in the export.
   - Sequence - the additional nodes are added to the script for the entire sequence.
   - Unconnected - the additional nodes are added to the script, but are not connected to the export node tree.
   - None - temporarily disables the current entry.

4. Select the Tags that you intend to use to filter which items receive the additional nodes.
   If you want to affect only the Reference track, for example, select the Reference tag. All items without that tag are ignored.

5. Copy and paste a node from the Node Graph into the Script panel.
Note: If you need more than one node, you might consider creating a Group in the Node Graph and pasting that into the Script panel.

6. Click OK to accept the additional nodes.
7. Select the Additional Nodes checkbox and complete the export process as described Exporting Sequences and Shots.
Using the Frame Server on External Machines

Although Nuke Studio is capable of rendering frames internally, running the Frame Server on an external machine can accelerate the process considerably by sharing work across a network of machines.

**Note:** The Frame Server requires a Nuke Studio license (nukestudio_i) on the main workstation, but only a Nuke render license (nuke_r) on the slave machines. Local Frame Server processes use ports 5558-5662.

If you want to use an interactive license (nuke_i) on the slave machines, add the --useInteractiveLicense argument to the runframeserver.py command described below.

Configuring the Frame Server on External Machines

Nuke Studio’s Frame Server can be set up on an external machine (or a number of machines) to render from your Nuke Studio session. To do this, you need to run the runframeserver.py script on the external machines, found inside the Python site-packages, with specific command line arguments.

**Warning:** In order for everything to work smoothly, you need to ensure that both your external slave machines and main Nuke Studio session can read and write files to a shared location, such as an NFS share.

Depending on platform this can be done by manipulating your default umask setting, but be aware that this alters the permissions of the created files.

Additionally, Mac and certain Linux distributions, such as RHEL, can not function as the main workstation if the firewall is blocking the communication port 5560. You can configure the firewall to allow certain ports through the firewall using the iptables command, but use caution when doing so. For example:

```
sudo iptables -I INPUT -p tcp --dport 5560 --syn -j ACCEPT
```
Please refer to the documentation on firewalls for your particular platform for more information.

The Frame Server uses a number of worker processes on the external machine, each of which requires allocated resources, such as threads, memory, and so on. There are a number of arguments that you must pass to `runframeserver.py` for the server to work correctly:

- **--numworkers** - this is the number of concurrent Nuke processes that are launched when you run this server render node.
- **--nukeworkerthreads** - the number of threads that each worker is allocated. This is similar to setting the `-m` argument when running Nuke from the command line.
- **--nukeworkermemory** - the amount of memory, in MB, allocated to each frame server worker.
- **--workerconnecturl** - the TCP port address of the main workstation you want to serve. For example:

```
tcp://bob:5560
```

where `bob` is the resolved hostname of a machine you wish to serve. You can also use an IP address.

**Tip:** To ensure that you're entering a valid URL, try using the `ping` command to see if you get a response.

- **--nukepath** - the path to the Nuke application on the slave workstation.

**Tip:** On Windows, if there are spaces in the file path, remember to place the path in quotes. For example, `--nukepath="C:\Program Files\Nuke11.3v6\Nuke11.3.exe"`

On a Linux slave machine, an example command prompt entry running from the install directory might look like this:

```
./python ./pythonextensions/site-packages/foundry/frameserver/nuke/runframeserver.py --numworkers=2 --nukeworkerthreads=4 --nukeworkermemory=8096 --workerconnecturl=tcp://bob:5560 --nukepath=./Nuke11.3
```

On a Windows slave machine, an example command prompt entry running from the install directory might look like this:

```
python.exe pythonextensions\site-packages\foundry\frameserver\nuke\runframeserver.py --numworkers=2 --nukeworkerthreads=4 --nukeworkermemory=8096 --workerconnecturl=tcp://bob:5560 --nukepath=Nuke11.3.exe
```
In the examples, we specify that the slave uses two Nuke workers, with four threads and 8 GB RAM each, and are slaved to the main Nuke Studio workstation running on **bob**.

---

**Tip:** If your slave machines run a different OS than your main Nuke Studio machine, you can use the **--remap** command line argument to convert file paths between them. The host file path is read first followed by the slave file path. Nuke Studio expects all file paths to use `/` (forward slash) between directories. For example:

```
--remap "P:/,/mnt/renders/"
```

converts host paths beginning with **P:/** (Windows style) to slave paths beginning with **/mnt/renders/** (Linux style).

---

You can check that the Frame Server and workers are connected by running the following lines in the Script Editor on the main workstation:

```python
from hiero.ui.nuke_bridge.FnNsFrameServer import frameServer
print [worker.address for worker in frameServer.getStatus(1).workerStatus]
```

Successful connections should report something similar to the following in the output panel:

```
['Worker 0 - henry.local - 192.168.1.11', 'Worker 0 - bob.local - 192.168.1.111', 'Worker 1 - henry.local - 192.168.1.11']
```

Where **henry.local** is the name of the remote slave, and **bob.local** is the name of the main Nuke Studio session.

---

**Note:** If the workers cannot contact the Frame Server, an exception is printed in the Script Editor's output panel.

---

### Frame Server Logs

Broker and Worker logging can help diagnose Frame Server issues. The logs are written to **NUKE_TEMP_DIR/logs** by default, and take the form:

- `broker.log`
- `worker-0.log`
- `worker-1.log`
- `worker-2.log`
**Note:** Running the Frame Server using Python, as described above, always writes log files to the specific OS temporary directory. For example, on Windows C:\temp is used.

**Tip:** You can use the FRAMESERVER\_LOG\_DIR environment variable to force Frame Server logs into a different location. See Environment Variables in the Nuke Online Help for more information.
Exporting Sequences and Shots

1. Select an entire sequence in the bin view, or shots in the timeline, and navigate to **File > Export...** The **Export** dialog displays.

2. Select **Process as Shots** from the **Export** dropdown.

3. Enter the **Export To** directory or click **Choose...** and browse to the location. The **Export To** directory is the starting point from which the shot template builds your shot hierarchy.

4. Select the **Basic Nuke Shot** preset under **Local Presets** to auto-complete the shot template or build a custom shot template by copying an existing template and editing as required using **Path** tokens, the **Contents** field, and the folder and +/- buttons.

   **Basic Nuke Shot** creates a folder for each clip, or shot, containing **nuke**, **script**, and **renders** folders.

   ![PATH](image)

   The tokens in the **Basic Nuke Shot** template break down as follows:
   - `{shot}` simply extracts the shot names as they appear in the timeline.
   - `{shot}_comp{_nameindex}_{version}.nk` extracts the shot name for each clip and the version selected in the **Tracks and Handles** controls. For example, **Shot01_comp_v03.nk**
   - `{shot}_comp{_nameindex}_{version}.####.{ext}` appends padding and the specified file extension. For example, **Shot01_comp_v03.0001.dpx**

   **Note:** The `{_nameindex}` token is included to avoid conflicts with non-unique shot names.

   **Tip:** Select a file entry in the shot template to display a preview of the file path with all the tokens resolved.

5. Proceed to **Nuke Project File Settings** to determine the Nuke script's behavior.
Nuke Project File Settings

Note: Custom shot presets can only be selected from the Project Settings if they contain a Nuke Project File and Nuke Write Node Content preset.

1. Click the Nuke Project File Content preset to display the script settings.
2. Select Write Nodes from the dropdown and check which path from the shot template should be used for the Nuke Write node. For example:
   `{shot}/nuke/renders/{shot}_comp_{nameindex}_{version}###.{ext}` to resolve the render path where Nuke Studio expects to find the files when they're rendered.

Note: If you included a Nuke Annotations File Content preset, enable the Annotations Precomp creator. See Annotations for more information.

3. If you're exporting retimed media, set how you want the Nuke script to handle the retime:
   - None - no retime is applied.
   - Motion - vector interpolation is used to calculate the in between frames. This is the most accurate retime method, but takes longer to render.
   - Frame - the nearest original frame is displayed.
   - Blend - a mix between two frames is used for the in between frames. This is quick to render and is useful when tweaking the timing in the Curve Editor before setting the method to Motion.

4. Soft Effects added to shots in your export are included in the resulting Nuke script by default. If you don't need the soft effects, disable Include Effects to omit them from the script. See Soft Effects for more information.

5. Select the required Reformattting options:
   - Plate Resolution - exports at the clip's original resolution, regardless of what is set in the timeline.
   - To Sequence Resolution - exports at the resolution set in the timeline Sequence panel Output Resolution dropdown.
   - Custom - activates the Reformat controls allowing you to customize the export resolution.

6. Enable Collate Shot Timings or Collate Shot Name to create additional Nuke Read nodes in the same script for clips that would normally be hidden by clips higher up the track hierarchy or clips on the same track with the same shot name.
Note: If you have a Read node selected, you can't enable the **Collate** functions.

For example:

- **Collate Shot Timings** - Items on track 1 that would otherwise be hidden by track 2.

![Timeline environment](image1) ![Compositing environment](image2)

Note: Shots on different tracks are not connected by default. If you want all the exported clips to be connected to the Nuke script Write node, enable **Connect Tracks**. This applies to stereo and multi-view sequences as well when you use separate files for the tracks per view. See [Stereoscopic and Multi-View Projects](#) for more information.

- **Collate Shot Name** - Two items on the same track with the same shot name.

![Timeline environment](image3) ![Compositing environment](image4)

7. If you want to add additional nodes to the script on export, enable **Additional Nodes** and click **Edit**. See [Adding Additional Nodes During Export](#) for more information.

8. Proceed to **Nuke Write Node Settings** to determine the Write node's behavior.
Nuke Write Node Settings

Note: Custom shot presets can only be selected from the Project Settings if they contain a Nuke Project File and Nuke Write Node Content preset.

1. Click the Nuke Write Node Content preset to display the write settings.
2. Set the following controls common to all file types:
   - Channels - set the channels to export using the dropdown. If you want to export a non-standard channel, type the name of the channel into the field manually.
   - Write Node Name - if you intend to create more than one Nuke Write node, define the name here. The default, Write_{ext}, appends the individual Write nodes with the file extension being written. You can, however, use any of the tokens Nuke Studio recognizes.
   - Colorspace - use the dropdown to set the colorspace to render, such as linear, REDLog, or raw.
3. Select the file type to render using the dropdown and complete the relevant fields, dependent on the file type selected.

Note: Selecting mov from the dropdown provides additional QuickTime specific controls, allowing you to choose a codec, encoder, and in some cases, YCbCrMatrix. The matrix control enables you to use the new Rec 601 and Rec 709 or the Legacy encoding methods, which are the methods used previously in Nuke. There’s also an Advanced dropdown containing mov32 and mov64 encoder specific controls.

Similarly, selecting exr provides an additional metadata dropdown allowing you to export or round-trip selected metadata along with your .exr output.

4. Create Directories is enabled by default, which enables the corresponding control in the .nk script's Write node. This control allows Nuke to create the required directories when you render out a new version from the Write node. Disabling this control causes versioned renders to fail because the target directories don't exist. You can manually create the correct directories or enable Create Directories in the script's Write node if this happens.
5. Use the Reformat controls to determine how the Write node is set up in the Nuke script:
   - None - the clip or sequence resolution is used, no additional formatting is applied during export.
   - To Sequence Resolution - exports at the resolution set in the timeline Sequence panel Output Resolution dropdown. This option also allows you to set the Filter used to resize the output.
Note: The filters available are generally listed in order of quality and processing time. Cubic can be faster to render, but Lanczos4 may be produce better results. See Choosing a Filtering Algorithm for more information on filters in Nuke.

- **To Scale** - activates all the Reformat controls, except Format, allowing you to customize the export resolution.
- **Custom** - activates all the Reformat controls, except Scale, allowing you to customize the export resolution.

<table>
<thead>
<tr>
<th>Reformat Control</th>
<th>To Sequence Resolution</th>
<th>To Scale</th>
<th>Custom</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Format</strong></td>
<td>- sets the format to render out in Nuke, such as 1920x1080 HD 1080.</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Scale</strong></td>
<td>- sets the proportion by which to scale the output format.</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Resize</strong></td>
<td>- sets the method by which you want to preserve or override the original aspect ratio:</td>
<td></td>
<td></td>
</tr>
<tr>
<td>• <strong>width</strong></td>
<td>- scales the original until its width matches the format's width. Height is then scaled in such a manner as to preserve the original aspect ratio.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>• <strong>height</strong></td>
<td>- scales the original until its height matches the format's height. Width is then scaled in such a manner as to preserve the original aspect ratio.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>• <strong>fit</strong></td>
<td>- scales the original until its smallest side matches the format's smallest side. The original's longer side is then scaled in such a manner as to preserve original aspect ratio.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>• <strong>fill</strong></td>
<td>- scales the original until its longest side matches the format's longest side. The input's shorter side is then scaled in such a manner as to preserve original aspect ratio.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>• <strong>distort</strong></td>
<td>- scales the original until all its sides match the lengths specified by the format. This option does not preserve the original aspect ratio, so distortion may occur.</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Center</strong></td>
<td>- when enabled, transform image to the center of the output. When disabled, the image is aligned with the bottom-left corner of the output.</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Filter** - sets the filtering algorithm used to transform pixels. See *Choosing a Filtering Algorithm* for more information on filters in Nuke.

6. You can apply text burn-in to the media using a Nuke gizmo. Enable **Burn-in Gizmo** and click **Edit** to define the information applied. See *Adding Burn-in Text to Exports* for more information.

7. Proceed to **Tracks, Range, and Handles Settings** to select which items are processed during export.
Tracks, Range, and Handles Settings

The tracks, tags, and handles controls in the Export dialog allow you to select the frame range or shots to export.

1. Set the Version number for the export, if applicable. Use the arrows to increment the version number and the +/- buttons to increase or decrease the padding. You can also type directly into the numeric field.

   **Note:** See Using Versions for more information on how versioning works in Nuke Studio.

2. Select Tracks for this export by enabling or disabling the tracks in the list. Nuke Studio exports all tracks by default.

3. Enable or disable tags using the Filter by tag panel. Click the checkbox to cycle through the available tag states.

4. If you're exporting a sequence, set the Range controls as required:
   - Select Whole Sequence or In/Out Points to export only the selected frames.
   - Set how clip Start Frames are derived using the dropdown menu:
     - Sequence - use the sequence's start frame.
     - Custom - specify a start frame for all clips using the field to the right.

5. If you're exporting shots, set the Handles controls as required:
   - Clip Length - exports the full clip length available, as if the clip was opened as a Viewer.
   - Cut Length - exports only the cuts included on the timeline.

   **Note:** Selecting Cut Length allows you to add handles to each clip, up to the maximum available source clip length.

   - Check Apply Retimes to export any retimes present on the timeline.

   **Note:** When Apply Retimes is disabled, which is the default state for Create Comp, any TimeWarp soft effects are not included in the resulting Nuke script. When the new shot is
created through **Create Comp** or **Build Track from Export Tag**. TimeWarp soft effects are copied from the original shot to the new one.

- **Set how clip Start Frames** are derived using the dropdown menu:
  - **Source** - use the source clip's start frame.
  - **Custom** - specify a start frame for all clips using the field to the right.

6. **Set how Nuke Studio should render your export using the Render with dropdown.** Nuke Studio provides the following options:
   - **Frame Server** - uses multiple Nuke processes to speed up render times and shares a render queue with any Nuke Comp renders in the timeline, improving resource management. See [Using the Frame Server on External Machines](#) for more information.
   - **Single Render Process** - uses a single Nuke process to render your export. Rendering QuickTimes falls back to this setting, but it's also used when a problem is detected with the Frame Server.
   - **Custom Render Process** - uses a custom render process. Nuke Studio requires a Python script to pass exports to your render farm of choice. Scripts must be located in specific directories, dependent on platform, as listed in [Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts](#). If no scripts exist, the dropdown only contains the default render processes.

7. **Click Export.**

   The **Export Queue** window displays an estimate of how long each component of the export is expected to take.

   Once the export is complete, the file structure specified in the shot template is created. You can then import the Nuke clips on a separate track when they're ready.

---

**Tip:** Click the magnifying glass icon to reveal the file structure in a browser window.

When clips are exported from Nuke Studio, they are marked with a Nuke tag flagging which clips have an export history. Clips tagged in this way can be used to build VFX tracks quickly as described in [Building VFX Tracks and Comp Clips](#).
Building VFX Tracks and Comp Clips

When the compositing work is complete, the clips are ready to re-ingest. The shot template defines where the Nuke files reside, so all you need to do is instruct Nuke Studio to build tracks from previous exports. See Building Tracks From Export Structure for more information.

Alternatively, if you have a history of rendered VFX clips, different versions and so on, you can also build tracks from export tags to select from a list of available clips. This method allows you to add Comp Clips to the timeline, which act as containers for Nuke .nk scripts, or placeholders for Nuke renders. See Building Comp Clips From Export Tags or Building Render Placeholders From Export Tags for more information.

Building Tracks From Export Structure

1. Select the required clips on the timeline and right-click to display the context sensitive menu.

   **Tip:** You may find it easier to select clips in the spreadsheet and then right-click on the timeline.

2. Click Build Track > From Export Structure.
   
   The Build Track From Export Structure dialog displays.

3. Enter a Track Name or use the default VFX.
4. Select an Export Preset using the dropdown menu. In this case, select the same preset used during the export.

5. Enter the file path of the Export Root directory or click Choose and browse to the location.

**Note:** The root directory is the location entered in Export To when exporting the project.

6. Select the Content preset you intend to ingest from the shot template. In this case, the Nuke Write Node.

7. Click Build to create the VFX track.

**Note:** Nuke Studio warns you if no selection is made in the Content column.

Nuke Studio automatically creates a new track containing the VFX clips, if they exist, or offline place holders if the clips are a work in progress. If a shot already exists in any of the target tracks, a new track is created to hold the new shots.

The clips are automatically updated when work is complete as long as they are saved with the expected name and location, as specified in the shot template.
Building Comp Clips From Export Tags

When you build a track from an export tag, Nuke Studio imports Comp Clips containing the Nuke script, by default. Comp Clips are shots that reference Nuke scripts, rather than being placeholders for offline clips as in the Building Render Placeholders From Export Tags workflow.

1. Select the required clips on the timeline and right-click to display the context sensitive menu.

   **Tip:** You may find it easier to select clips in the spreadsheet and then right-click on the timeline.

2. Click **Build Track > From Export Tag**.

   The **Build Track From Export Tag** dialog displays.

3. Enter a **Track Name** or use the default **VFX**.

4. Select the required export tag in the left-hand panel to display tag information in the right-hand panel.

   Nuke Studio imports the .nk Comp Clip by default.

   ![Build Track From Export Tag Dialog]

   If you just want to import the offline renders when they're finished, disable the **Create Comp Clips** checkbox. See Building Render Placeholders From Export Tags for more information.

5. Click **Build** to create the VFX track.

   Nuke Studio automatically creates a new track containing the Comp Clips. If a shot already exists in any of the target tracks, a new track is created to hold the new shots.
6. You can double-click Comp Clips to open them in Nuke Studio's Compositing environment to make edits as required.

Building Render Placeholders From Export Tags

When you build a track from an export tag, you can choose to import the renders from the .nk script, rather than Comp Clips which contain the Nuke script.

1. Select the required clips on the timeline and right-click to display the context sensitive menu.

   **Tip:** You may find it easier to select clips in the spreadsheet and then right-click on the timeline.

2. Click **Build Track > From Export Tag**.
   
   The **Build Track From Export Tag** dialog displays.

3. Enter a **Track Name** or use the default **VFX**.

4. Select the required export tag in the left-hand panel to display tag information in the right-hand panel.

5. Disable the **Create Comp Clips** checkbox to import the offline renders when they're finished.
If you want to import the .nk Comp Clips, enable the **Create Comp Clips** checkbox. See [Building Comp Clips From Export Tags](#) for more information.

6. Click **Build** to create the VFX track.

Nuke Studio automatically creates a new track containing the VFX clips, if they exist, or offline place holders if the clips are a work in progress.

If a shot already exists in any of the target tracks, a new track is created to hold the new shots.
Exporting Multi-View Source Clips

Multi-view exports are similar to regular exports, but the **DPX Multi-View** or **Multi-View Nuke Shot** templates are used to create the required export tree using %v functionality, just like Nuke's Node Graph. The **DPX Multi-View** example preset is designed for exporting sequences and the **Multi-View Nuke Shot** preset is designed for shots.

See [Stereoscopic and Multi-View Projects](#) for more information on working with multi-view footage.

1. Right-click on the sequence or shot that you want to export.
   - Sequences are exported from the bin view.

   ![Sequence in Bin View](image)

   Shots can be a multi-view or single views split into separate tracks. See [Displaying Views in the Timeline](#) for more information.

   ![Multi-View Timeline](image)

   For separate tracks, you can right-click a single track to export all views or select all the per-track views and right-click.

2. Select **Export**.
The **Export** dialog is displayed.

3. Select **Process as Sequence** or **Process as Shots** from the **Export** dropdown.
   
   If you're exporting a sequence, select the **Log10 Cineon DPX Multi-View** example preset. If you're exporting shots, select the **Multi-View Nuke Shot (%v)** example preset.

4. Set the **Nuke Project File** and **Nuke Write Node** presets as described in **Nuke Project File Settings** and **Nuke Write Node Settings**.

   If you're using independent files per track, that is without importing multi-view files or using %V functionality, the separate tracks in the export script are not connected by default. If you want the tracks to be connected in the script, enable **Connect Tracks** in the **Nuke Project File** preset.

5. Set the track and handle preferences as described in **Tracks, Range, and Handles Settings** and then click **Export**.

   Once the export is complete, the file structure specified in the shot template is created. You can then import the Nuke clips on a separate track when they're ready.

   **Tip:** Click the magnifying glass icon in the **Export Queue** to reveal the file structure in a browser window.

When clips are exported from Nuke Studio, they are marked with a Nuke tag flagging which clips have an export history. Clips tagged in this way can be used to build VFX tracks quickly as described in **Building VFX Tracks and Comp Clips**.
Transcoding

Transcoding in Nuke Studio uses a background render process to convert one file format to another. You can transcode sequences, timeline selections, and clips from the bin view.

Transcoding a Sequence

1. Select a sequence in the bin view and navigate to File > Export...
   The Export dialog displays.
2. Select Process as Sequence and the preset you intend to use, or use the default dpx preset.
3. Enter the Export To directory or click Choose... and browse to the location.
4. Click the Content column in the shot template to display the transcode options.
5. Set the following controls common to all file types:
   • Channels - set the channels to export using the dropdown. If you want to export a non-standard channel, type the name of the channel into the field manually.
   • Colorspace - use the dropdown to set the colorspace to render, such as linear, REDLog, or raw.
6. Select the File Type to render using the dropdown and complete the relevant fields, dependent on the File Type selected.

   ![Note: Selecting mov from the dropdown provides additional QuickTime specific controls, allowing you to choose a codec, encoder, and in some cases, YCbCrMatrix. The matrix control enables you to use the new Rec 601 and Rec 709 or the Legacy encoding methods, which are the methods used previously in Nuke. There’s also an Advanced dropdown containing mov32 and mov64 encoder specific controls. Similarly, selecting exr provides an additional metadata dropdown allowing you to export or round-trip selected metadata along with your .exr output.]

7. Use the Reformat controls to determine how the Write node is set up in the Nuke script:
   • None - the clip or sequence resolution is used, no additional formatting is applied during export.
   • To Scale - activates all the Reformat controls, except Format, allowing you to customize the export resolution.
   • Custom - activates all the Reformat controls, except Scale, allowing you to customize the export resolution.
Reformat Control

<table>
<thead>
<tr>
<th>To Scale</th>
<th>Custom</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Format</strong> - sets the format to render out in Nuke, such as <strong>1920x1080 HD 1080</strong>.</td>
<td></td>
</tr>
<tr>
<td><strong>Scale</strong> - sets the proportion by which to scale the output format.</td>
<td></td>
</tr>
</tbody>
</table>

**Resize** - sets the method by which you want to preserve or override the original aspect ratio:

- **width** - scales the original until its width matches the format’s width. Height is then scaled in such a manner as to preserve the original aspect ratio.
- **height** - scales the original until its height matches the format’s height. Width is then scaled in such a manner as to preserve the original aspect ratio.
- **fit** - scales the original until its smallest side matches the format’s smallest side. The original’s longer side is then scaled in such a manner as to preserve original aspect ratio.
- **fill** - scales the original until its longest side matches the format’s longest side. The input’s shorter side is then scaled in such a manner as to preserve original aspect ratio.
- **distort** - scales the original until all its sides match the lengths specified by the format. This option does not preserve the original aspect ratio, so distortion may occur.

**Center** - when enabled, transform image to the center of the output. When disabled, the image is aligned with the bottom-left corner of the output.

**Filter** - sets the filtering algorithm used to transform pixels. See [Choosing a Filtering Algorithm](#) for more information on filters in Nuke.

8. Complete the general controls common to all file types:
   - Select the **Retime Method** to apply, if applicable.
   - Soft Effects added to shots in your export are included in the resulting Nuke script by default. If you don’t need the soft effects, disable **Include Effects** to omit them from the script. See [Soft Effects](#) for more information.
   - **Include Audio** - when enabled, any audio tracks are exported alongside the video.
   - **Delete .WAV File** - when disabled, delete the .wav file used to create the export. This control only applies to formats that support audio, such as .mov files.

9. Enable the **Burn-in Gizmo** to burn-in text using a Nuke gizmo. Click **Edit** to define the information applied during burn-in. See [Adding Burn-in Text to Exports](#) for more information.
10. Specify any **Additional Nodes** required during export by clicking **Edit**. See **Adding Additional Nodes During Export** for more information.

11. Check **Keep Nuke Script** if you require the `.nk` files after the transcode operation.

**Note:** The following controls may improve render times for certain exports:

- For `.dp` exports, you can enable **Read All Lines**, which can speed up transcode times for I/O heavy scripts.
- For machines with multiple CPU sockets using a **Single Render Process**, you may find that limiting the render with **Use Single Socket** may improve render times.

---

**Tracks and Range Settings**

1. Set the **Version** number for the export, if applicable. Use the arrows to increment the version number and the +/- buttons to increase or decrease the padding. You can also type directly into the numeric field.

**Note:** See **Using Versions** for more information on how versioning works in Nuke Studio.

2. Select **Tracks for this export** by enabling or disabling the tracks in the list. Nuke Studio exports **all tracks** by default.

3. If you set in and out point on the sequence, enable In/Out Points to export only the selected frames.

4. Set how clip **Start Frames** are derived using the dropdown menu:
   - **Sequence** - use the sequence's start frame.
   - **Custom** - specify a start frame for all clips using the field to the right.

5. Set how Nuke Studio should render your export using the **Render with** dropdown. The following options are available:
   - **Frame Server** - uses multiple Nuke processes to speed up render times and shares a render queue with any Nuke Comp renders in the timeline, improving resource management. See **Using the Frame Server on External Machines** for more information.
   - **Single Render Process** - uses a single Nuke process to render your export. Rendering QuickTimes falls back to this setting, but it's also used when a problem is detected with the Frame Server.
   - **Custom Render Process** - uses a custom render process. Nuke Studio requires a Python script to pass exports to your render farm of choice. Scripts must be located in specific directories,
dependent on platform, as listed in Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts. If no scripts exist, the dropdown only contains the default render processes.

6. Click Export.
   The Export Queue window displays an estimate of how long each component of the export is expected to take.
   Once the export is complete, the file structure specified in the shot template is created containing the transcoded files.

Tip: Click the magnifying glass icon to reveal the exported file in a browser window.
Transcoding a Sequence as Shots

1. Select the required sequence in the bin view and navigate to File > Export...
   The Export dialog displays.

2. Select Process as Shots and use the default, Transcode Shots DPX, or build a shot template using the Path and Contents fields and the folder and +/- buttons.
   The default:

   ![Export Structure](image)

   Creates a folder for each shot, containing a clip with the {shot} name and the required file padding (####) and extension {ext}.

3. Enter the Export To directory or click Choose... and browse to the location.
   The Export To directory is the starting point from which the shot template builds your shot hierarchy.

4. In the Content tab, complete the File Type specific and general controls common to all file types as described in Transcoding a Sequence.

5. Click the Tracks and Handles tab, select the Tracks For This Export by enabling or disabling the tracks in the list. Nuke Studio exports all tracks by default.

6. Enable or disable tags using the Filter by Tag panel. Click the checkbox to cycle through the available tag states.

7. Set the Range and Handles, as required:
   - Clip Length - exports the full clip length available, as if the clip was opened as a Viewer.
   - Cut Length - exports only the cuts included on the timeline.

   **Note:** Selecting Cut Length allows you to add handles to each clip, up to the maximum available source clip length.

8. Check Apply Retimes to export any retimes present on the timeline.

   **Note:** When Apply Retimes is disabled, which is the default state for Create Comp, any TimeWarp soft effects are not included in the resulting Nuke script. When the new shot is
created through Create Comp or Build Track from Export Tag, TimeWarp soft effects are copied from the original shot to the new one.

9. Set how clip Start Frames are derived using the dropdown menu:
   - **Source** - use the source clip's start frame.
   - **Custom** - specify a start frame for all clips using the field to the right.

10. Set the Version number for the export, if applicable. Use the arrows to increment the version number and the +/- buttons to increase or decrease the padding. You can also type directly into the numeric field.

**Note:** See Using Versions for more information on how versioning works in Nuke Studio.

11. Set how Nuke Studio should render your export using the Render with dropdown. Nuke Studio provides the following options:
   - **Frame Server** - uses multiple Nuke processes to speed up render times and shares a render queue with any Nuke Comp renders in the timeline, improving resource management.
     See Using the Frame Server on External Machines for more information.
   - **Single Render Process** - uses a single Nuke process to render your export. Rendering QuickTimes falls back to this setting, but it's also used when a problem is detected with the Frame Server.
   - **Custom Render Process** - uses a custom render process. Nuke Studio requires a Python script to pass exports to your render farm of choice. Scripts must be located in specific directories, dependent on platform, as listed in Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts. If no scripts exist, the dropdown only contains the default render processes.

12. Click Export.

   The Export Queue window displays an estimate of how long each component of the export is expected to take.

   Once the export is complete, the file structure specified in the shot template is created. You can then import the Nuke clips on a separate track when they're ready.

**Tip:** Click the magnifying glass icon to reveal the file structure in a browser window.
Transcoding Timeline Selections

Transcoding an entire timeline can be time consuming, or even unnecessary, if all you're looking for is a new version of a selection of shots.

To transcode a selection of clips from a timeline:
1. Select the required shots on the timeline.
2. Right-click a highlighted item and select Export...

The Export dialog displays.
3. Refer to Transcoding a Sequence to complete the export.
Transcoding from the Bin View

To transcode directly from the bin view:
1. Select the bin(s) to export from the bin view.
2. Right-click a highlighted bin and select Export...

The Export dialog displays.
3. Select Process as Clips and modify the shot template, if required.
4. Follow the steps under Transcoding a Sequence to complete the export.
Ad Hoc Exports

This section covers exports that you might not perform on a per project basis, such as the EDL or XML Exporters and Copy Exporter. Exporters are available for sequences, shots, and clips as described in the following table.

<table>
<thead>
<tr>
<th>Exporter</th>
<th>Source</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Sequences</td>
</tr>
<tr>
<td>EDL Exporter</td>
<td></td>
</tr>
<tr>
<td>XML Exporter</td>
<td></td>
</tr>
<tr>
<td>Audio Exporter</td>
<td></td>
</tr>
<tr>
<td>Copy Exporter</td>
<td></td>
</tr>
<tr>
<td>SymLink Exporter</td>
<td></td>
</tr>
</tbody>
</table>

AdHoc Exports | Tracks and Range Settings
Exporting EDLs and XMLs

Nuke Studio supports export to EDL and XML using very similar methods, the main difference being that EDL doesn't support multiple video tracks in a single file whereas XML does.

**Note:** Nuke Studio can read AAF files, but not write them out.

To export to EDL or XML:
1. Select a sequence in the bin view and navigate to File > Export...
   
The Export dialog displays.
2. Select Process as Sequence from the Export dropdown.
3. Select the CMX 3600 EDL or Final Cut Pro XML preset, or duplicate one and create your own preset.

**Note:** EDLs only support one video track per file. If you have more than one track, include the `{track}` token in the shot template to write out an EDL for each track preset.

For example, `{filename}_{{track}}.{ext}` might produce a separate EDL for each track on your timeline called myTimeline_Video1.edl, myTimeline_Video2.edl, and so on.

4. Enter the Export To directory or click Choose... and browse to the location.
   
The Export To directory is the starting point from which the shot template builds your shot hierarchy.
5. If you're exporting to EDL, set the additional EDL Exporter controls in the Content tab, if required:
   - **Reel Name** - define the reel name written into the EDL, independent of the clip's reel name.
     
Enter text or standard shot-level tokens in this field. See Using the Shot Template for more information.
     
     If the field is left blank, the reel name from the clip is used or the name of the shot, if no reel name exists.
   - **Truncate Reel Name** - restricts the Reel name to eight characters.
   - **Use Absolute Path** - adds the full file path for each clip to the EDL comments field.
   - **From Clip Name** - define the text appended to “from” comment fields in EDLs, such as *FROM CLIP NAME*. Text and/or standard shot-level tokens are valid in this field: \{shot\}, \{clip\}, \{track\}, \{sequence\}, \{event\}, \{fps\}, and the default \{filename\}.

OR
If you're exporting to XML, you can enable **Include Markers** to convert any frame tags present in the sequence to markers in Final Cut Pro or Premiere. See [Tagging Using the Viewer](#) for more information on adding tags to frames.

6. Click the **Tracks and Range** tab and select the **Tracks For This Export** by enabling or disabling the tracks in the list. Nuke Studio exports **all tracks** by default.

7. If you set in and out point on the sequence, enable **In/Out Points** to export only the selected frames.

8. Set how clip **Start Frames** are derived using the dropdown menu:
   - **Sequence** - use the sequence's start frame.
   - **Custom** - specify a start frame for all clips using the field to the right.

9. Set the **Version** number for the export, if applicable. Use the arrows to increment the version number and the +/- buttons to increase or decrease the padding. You can also type directly into the numeric field.

```
Note: See **Using Versions** for more information on how versioning works in Nuke Studio.
```

10. Set how Nuke Studio should render your export using the **Render with** dropdown. Nuke Studio provides the following options:

   - **Frame Server** - uses multiple Nuke processes to speed up render times and shares a render queue with any Nuke Comp renders in the timeline, improving resource management.
     See [Using the Frame Server on External Machines](#) for more information.
   - **Single Render Process** - uses a single Nuke process to render your export. Rendering QuickTimes falls back to this setting, but it's also used when a problem is detected with the Frame Server.
   - **Custom Render Process** - uses a custom render process. Nuke Studio requires a Python script to pass exports to your render farm of choice. Scripts must be located in specific directories, dependent on platform, as listed in [Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts](#). If no scripts exist, the dropdown only contains the default render processes.

11. Click **Export**.

   The **Export Queue** window displays an estimate of how long each component of the export is expected to take.

   Once the export is complete, the file structure specified in the shot template is created.

```
Tip: Click the magnifying glass icon to reveal the exported file in a browser window.
```
Using the Audio Exporter

The Audio Exporter allows you write audio to separate .wav files. You can extract audio from whole sequences, shots, and source clips.

Exporting Audio from Sequences

1. Select a sequence in the bin view and navigate to File > Export...

The Export dialog displays.

2. Select Process as Sequence from the Export dropdown.

3. Select the Log10 Cineon DPX preset, duplicate it, and give it a name.

4. Enter the Export To directory or click Choose... and browse to the location.

The Export To directory is the starting point from which the shot template builds your shot hierarchy.

5. Build a custom shot template using Path tokens, the Contents field, and the folder and +/- buttons.

An example shot template is shown below:

![Example Shot Template]

6. Set the Version number for the export, if applicable. Use the arrows to increment the version number and the +/- buttons to increase or decrease the padding. You can also type directly into the numeric field.

Note: See Using Versions for more information on how versioning works in Nuke Studio.

7. Select Tracks for this export by enabling or disabling the tracks in the list. Nuke Studio exports all tracks by default.

8. Set the Range controls as required:
   • Select Whole Sequence or In/Out Points to export only the selected frames.
• Set how clip **Start Frames** are derived using the dropdown menu:
  • **Sequence** - use the sequence’s start frame.
  • **Custom** - specify a start frame for all clips using the field to the right.

9. Set how Nuke Studio should render your export using the **Render with** dropdown. Nuke Studio provides the following options:
  • **Frame Server** - uses multiple Nuke processes to speed up render times and shares a render queue with any Nuke Comp renders in the timeline, improving resource management.
    See [Using the Frame Server on External Machines](#) for more information.
  • **Single Render Process** - uses a single Nuke process to render your export. Rendering QuickTimes falls back to this setting, but it’s also used when a problem is detected with the Frame Server.
  • **Custom Render Process** - uses a custom render process. Nuke Studio requires a Python script to pass exports to your render farm of choice. Scripts must be located in specific directories, dependent on platform, as listed in [Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts](#). If no scripts exist, the dropdown only contains the default render processes.

10. Click **Export**.

The **Export Queue** window displays an estimate of how long each component of the export is expected to take.

Once the export is complete, the file structure specified in the shot template is created.

**Tip:** Click the magnifying glass icon to reveal the exported file in a browser window.

---

**Exporting Audio from Shots**

1. Select the required shots in the timeline and navigate to **File > Export**...
   
   The **Export** dialog displays.

2. Select **Process as Shots** from the Export list.

3. Select the **Transcode Shots DPX** preset, duplicate it, and give it a name.

4. Enter the **Export To** directory or click **Choose**... and browse to the location.
   
   The **Export To** directory is the starting point from which the shot template builds your shot hierarchy.

5. Build a custom shot template using **Path** tokens, the **Contents** field, and the folder and +/- buttons.
   
   An example shot template is shown below:
6. Set the **Version** number for the export, if applicable. Use the arrows to increment the version number and the +/- buttons to increase or decrease the padding. You can also type directly into the numeric field.

   **Note:** See **Using Versions** for more information on how versioning works in Nuke Studio.

7. Select **Tracks for this export** by enabling or disabling the tracks in the list. Nuke Studio exports all tracks by default.

8. Enable or disable tags using the **Filter by tag** panel. Click the checkbox to cycle through the available tag states.

9. If you're exporting a sequence, set the **Range** controls as required:
   - Select **Whole Sequence** or **In/Out Points** to export only the selected frames.
   - Set how clip **Start Frames** are derived using the dropdown menu:
     - **Sequence** - use the sequence's start frame.
     - **Custom** - specify a start frame for all clips using the field to the right.

10. If you're exporting shots, set the **Handles** controls as required:
    - **Clip Length** - exports the full clip length available, as if the clip was opened as a Viewer.
    - **Cut Length** - exports only the cuts included on the timeline.

    **Note:** Selecting **Cut Length** allows you to add handles to each clip, up to the maximum available source clip length.

11. Set how clip **Start Frames** are derived using the dropdown menu:
    - **Source** - use the source clip's start frame.
    - **Custom** - specify a start frame for all clips using the field to the right.

12. Set how Nuke Studio should render your export using the **Render with** dropdown. Nuke Studio provides the following options:
• **Frame Server** - uses multiple Nuke processes to speed up render times and shares a render queue with any Nuke Comp renders in the timeline, improving resource management. See [Using the Frame Server on External Machines](#) for more information.

• **Single Render Process** - uses a single Nuke process to render your export. Rendering QuickTimes falls back to this setting, but it's also used when a problem is detected with the Frame Server.

• **Custom Render Process** - uses a custom render process. Nuke Studio requires a Python script to pass exports to your render farm of choice. Scripts must be located in specific directories, dependent on platform, as listed in [Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts](#). If no scripts exist, the dropdown only contains the default render processes.

13. Click **Export**.

The **Export Queue** window displays an estimate of how long each component of the export is expected to take.

Once the export is complete, the file structure specified in the shot template is created.

---

**Tip:** Click the magnifying glass icon to reveal the exported file in a browser window.

---

### Exporting Audio from Source Clips

1. Select the required source clips and navigate to **File > Export...**

   The **Export** dialog displays.

2. **Process as Clips** is selected automatically from the Export list.

3. Select the **Transcode Clips DPX** preset, duplicate it, and give it a name.

4. Enter the **Export To** directory or click **Choose...** and browse to the location.

   The **Export To** directory is the starting point from which the shot template builds your shot hierarchy.

5. Build a custom shot template using **Path** tokens, the **Contents** field, and the folder and +/- buttons.

   An example shot template is shown below:
6. Set the **Version** number for the export, if applicable. Use the arrows to increment the version number and the +/- buttons to increase or decrease the padding. You can also type directly into the numeric field.

**Note:** See *Using Versions* for more information on how versioning works in Nuke Studio.

7. Set how clip **Start Frames** are derived using the dropdown menu:
   - **Source** - use the source clip’s start frame.
   - **Custom** - specify a start frame for all clips using the field to the right.

8. Set how Nuke Studio should render your export using the **Render with** dropdown. Nuke Studio provides the following options:
   - **Frame Server** - uses multiple Nuke processes to speed up render times and shares a render queue with any Nuke Comp renders in the timeline, improving resource management.
   - **Single Render Process** - uses a single Nuke process to render your export. Rendering QuickTimes falls back to this setting, but it’s also used when a problem is detected with the Frame Server.
   - **Custom Render Process** - uses a custom render process. Nuke Studio requires a Python script to pass exports to your render farm of choice. Scripts must be located in specific directories, dependent on platform, as listed in *Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts*. If no scripts exist, the dropdown only contains the default render processes.

9. Click **Export**.
   
   The **Export Queue** window displays an estimate of how long each component of the export is expected to take.
   
   Once the export is complete, the file structure specified in the shot template is created.

**Tip:** Click the magnifying glass icon to reveal the exported file in a browser window.
Using the Copy Exporter

Copying media from various locations is very time consuming and can waste disk space. The Copy Exporter allows you to consolidate sequences containing only your project media in a named file structure using the shot template.

To copy media to a named location:
1. Select a sequence in the bin view and navigate to File > Export...
   The Export dialog displays.
2. Select Process as Shots from the Export dropdown.
3. Select the Transcode Shots DPX preset, duplicate it, and give it a name.
4. Enter the Export To directory or click Choose... and browse to the location.
   The Export To directory is the starting point from which the shot template builds your shot hierarchy.
5. Build a custom shot template using Path tokens, the Contents field, and the folder and +/- buttons.
   An example shot template is shown below:

![Shot Template Example]

6. Set the Version number for the export, if applicable. Use the arrows to increment the version number and the +/- buttons to increase or decrease the padding. You can also type directly into the numeric field.

   **Note:** See Using Versions for more information on how versioning works in Nuke Studio.

7. Set how clip Start Frames are derived using the dropdown menu:
   - Source - use the source start frame.
   - Custom - specify a start frame for all clips using the field to the right.

8. Set how Nuke Studio should render your export using the Render with dropdown. Nuke Studio provides the following options:
• **Frame Server** - uses multiple Nuke processes to speed up render times and shares a render queue with any Nuke Comp renders in the timeline, improving resource management. See [Using the Frame Server on External Machines](#) for more information.

• **Single Render Process** - uses a single Nuke process to render your export. Rendering QuickTimes falls back to this setting, but it's also used when a problem is detected with the Frame Server.

• **Custom Render Process** - uses a custom render process. Nuke Studio requires a Python script to pass exports to your render farm of choice. Scripts must be located in specific directories, dependent on platform, as listed in [Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts](#). If no scripts exist, the dropdown only contains the default render processes.

9. Click **Export**.

   The **Export Queue** window displays an estimate of how long each component of the export is expected to take.

   Once the export is complete, the file structure specified in the shot template is created.

   **Tip:** Click the magnifying glass icon to reveal the exported file in a browser window.
Using the SymLink Generator

The SymLink Generator allows you to create symbolic links to your project media in a named file structure using the shot template.

**Note:** Windows only: Symbolic links are only supported by Windows Vista, or later. If you're linking across file systems, the remote file servers must also be running Windows Vista, or later. Additionally, you may need administrator privileges and a local NTFS drive to create symbolic links.

To create symbolic links to a named location:
1. Select a sequence in the bin view and navigate to **File > Export...**
   The **Export** dialog displays.
2. Select **Process as Shots** from the **Export** dropdown.
3. Select the **Transcode Shots DPX** preset, duplicate it, and give it a name.
4. Enter the **Export To** directory or click **Choose...** and browse to the location.
   The **Export To** directory is the starting point from which the shot template builds your shot hierarchy.
5. Build a custom shot template using **Path** tokens, the **Contents** field, and the folder and +/- buttons.
   An example shot template is shown below:

   ![Example Shot Template](image)

6. Set the **Version** number for the export, if applicable. Use the arrows to increment the version number and the +/- buttons to increase or decrease the padding. You can also type directly into the numeric field.

   **Note:** See **Using Versions** for more information on how versioning works in Nuke Studio.

7. Set how clip **Start Frames** are derived using the dropdown menu:
   - **Source** - use the source start frame.
• **Custom** - specify a start frame for all clips using the field to the right.

8. Set how Nuke Studio should render your export using the **Render with** dropdown. Nuke Studio provides the following options:

- **Frame Server** - uses multiple Nuke processes to speed up render times and shares a render queue with any Nuke Comp renders in the timeline, improving resource management. See [Using the Frame Server on External Machines](#) for more information.

- **Single Render Process** - uses a single Nuke process to render your export. Rendering QuickTimes falls back to this setting, but it's also used when a problem is detected with the Frame Server.

- **Custom Render Process** - uses a custom render process. Nuke Studio requires a Python script to pass exports to your render farm of choice. Scripts must be located in specific directories, dependent on platform, as listed in [Loading Gizmos, NDK Plug-ins, and Python and Tcl Scripts](#). If no scripts exist, the dropdown only contains the default render processes.

9. Click **Export**.

The **Export Queue** window displays an estimate of how long each component of the export is expected to take.

Once the export is complete, the file structure specified in the shot template is created.

**Tip:** Click the magnifying glass icon to reveal the exported file in a browser window.
Advanced Compositing with NukeX and Nuke Studio

These pages explain in detail key feature of NukeX and Nuke Studio. For more information on the differences between the various applications, see Nuke Products. These are the topics covered:

- **VectorGenerator, Kronos, and MotionBlur** - VectorGenerator, Kronos, and MotionBlur use Foundry's advanced motion estimation technology to produce images containing motion vector fields, slow down or speed up footage, and add motion blur. For more information, see Retiming and Motion Blur.

- **LensDistortion** - The LensDistortion node gives you multiple ways to analyze image sequences and lens grids, resulting in a lens model and the ability to warp and un-warp in order to compensate for lens distortion. For more information, see Working with Lens Distortion.

- **PlanarTracker** - The PlanarTracker is a powerful tool for tracking surfaces that lie on a plane in your source footage. You can use your tracking results to replace the tracked plane with another image for instance. For more information, see Tracking with PlanarTracker.

- **CameraTracker** - With the fully integrated 3D CameraTracker node, you can do your own camera solves and create reference geometry and cards positioned at tracked points in the 3D scene. For more information, see Camera Tracking.

- **MatchGrade** - The MatchGrade node allows you to automatically calculate a grade to match the colors in the **Source** input to the colors in the **Target** input. For more information, see Using MatchGrade.

- **Smart Vector Toolset** - The Smart Vector Toolset allows you to work on one frame in a sequence and then use motion vector information to accurately propagate work throughout the rest of the sequence. For more information, see Using the Smart Vector Toolset.

- **DepthGenerator** - The DepthGenerator node provides a method to produce a per-frame Z-depth map from the input 2D footage. It additionally requires a camera solve which can be obtained using the CameraTracker node. For more information, see Generating Depth Maps.

- **PointCloudGenerator** - You can create dense point clouds from your footage using the PointCloudGenerator and CameraTracker. For more information, see Creating Dense Point Clouds.

- **PoissonMesh** - With the PoissonMesh node, you can use a dense point cloud to create a 3D mesh from your 2D footage. For more information, see Using the PoissonMesh Node.

- **ModelBuilder** - ModelBuilder provides an easy way to create 3D models for a 2D shot, given a tracked camera. You can build a model by creating shapes and then editing them, and align models over your 2D footage by dragging vertices to their corresponding 2D location. For more information, see Using ModelBuilder.
• **Particles** - The Particle node set is a solution for creating particles in a 3D environment. You can use the Particle nodes for emitting, manipulating and displaying limitless types of particles in your 3D scene. For more information, see *Creating 3D Particles*.

• **PrmanRender** - PrmanRender is a render node that works together with Pixar’s PhotoRealistic RenderMan® Pro Server software to give you an even better quality render result. PrmanRender is an alternative to the ScanlineRender node for rendering 3D scenes, and it gives you control over features such as shadows, reflections, refractions and depth-of-field. For more information, see *PrmanRender*.

• **FurnaceCore** - This plug-in bundle consists of Foundry’s best Furnace tools, including regraining, rig-removal, and more. For more information, see *Using F_DeFlicker2*, *Using F_ReGrain*, *Using F_WireRemoval*, *Using F_Align*, *Using F_RigRemoval*, and *Using F_Steadiness*.

**Note:** The following FurnaceCore nodes have been replaced by other nodes and can no longer be found in the **FurnaceCore** menu:
- F_DeGrain and F_Denoise were replaced by Denoise (Filter > Denoise) in Nuke 6.3.
- F_Kronos was replaced by Kronos (Time > Kronos) in Nuke 7.0.
- F_MotionBlur was replaced by MotionBlur (Filter > MotionBlur) in Nuke 7.0.
- F_VectorGenerator was replaced by VectorGenerator (Time > VectorGenerator) in Nuke 7.0.

• **BlinkScript** - The BlinkScript node runs Foundry’s Blink framework enabling you to write your code once and run it on any supported device. For more information, see *Using the BlinkScript Node* on page 1817.
Retiming and Motion Blur

This chapter looks at creating motion vectors using VectorGenerator, retiming sequences using Kronos, and adding motion blur using MotionBlur.
VectorGenerator

VectorGenerator allows you to produce images containing motion vector fields. A vector field for an image in a sequence has the same dimensions as the image, but contains an \((x,y)\) offset per pixel. These offsets show how to warp a neighboring image onto the current image. Clearly, as most of the images in a sequence have two neighbors, each can have two vector fields:

1. The backward vector field: the \(x\) and \(y\) offsets per pixel that, when applied to the previous frame in the sequence, allow you to reconstruct an approximation to the current frame.
2. The forward vector field: the \(x\) and \(y\) offsets needed to transform the next frame into an approximation to the current one.

The output from VectorGenerator is stored in the vector channels. The images below show the different vector images for a clip.

When viewing 'forward' or 'backward' motion vectors, the Viewer represents the \(x\) values as amounts of red and \(y\) values as amounts of green. Motion vectors can be positive or negative, where zero represents no motion.

In general, once you have generated a sequence of motion vector fields that describe the motion in a particular clip well, they are suitable for use in any nodes which can take vector inputs. These include Kronos and MotionBlur. If you are going to be using more than one of these effects in your project, it might be worth generating the vector fields beforehand with VectorGenerator, so that they can be reused.
Quick Start

Here's a quick overview of the workflow:

1. Add VectorGenerator to your node tree. See Connecting VectorGenerator below.
2. View and refine the results. See Viewing and Refining the Results.

Tip: You can check Use GPU if available to have the node run on the graphics processing unit (GPU) rather than the central processing unit (CPU).

For more information on the minimum requirements, please see Windows, Mac OS X and macOS, or Linux or refer to the Nuke Release Notes available in Help > Release Notes.

You can select the GPU to use in the Preferences. Press Shift+S to open the Preferences dialog, make sure you're viewing the Preferences > Performance > Hardware tab, and set default blink device to the device you want to use. You must restart Nuke for the change to take effect.

If you are using a render license of Nuke, you need to add --gpu on the command line.
Connecting VectorGenerator

To connect VectorGenerator:

1. Select **Time > VectorGenerator** to insert a VectorGenerator node after the sequence from which you want to generate motion vectors.

2. Attach a Viewer to the output of the VectorGenerator node.

3. If your sequence is composed of a foreground object moving over a background, the motion estimation is likely to get confused at the edge between the two. To fix this, add a matte of the foreground region to the **Matte** input. Then, use **Matte Channel** in the VectorGenerator properties to select which component of the matte to use.
   This helps the motion estimation algorithm inside VectorGenerator understand what is foreground and background in the image, so that the dragging of pixels between overlapping objects can be reduced.
   White areas of the matte are considered to be foreground, and black areas background. Gray areas are used to attenuate between foreground and background.

4. If you supplied a foreground matte in the previous step, you can set **Output** in the VectorGenerator properties to:
   - **Foreground** - to output the vectors for the foreground regions.
   - **Background** - to output the vectors in the background regions.

5. Proceed to **Viewing and Refining the Results**.
Viewing and Refining the Results

To view and refine the results:

1. Select the required vector calculation type from the Method dropdown:
   - **Local** - uses local block matching to estimate motion vectors. This method is faster to process, but can lead to artifacts in the output.
   - **Regularized** - uses semi-global motion estimation to produce more consistent vectors between regions.

   **Note:** Scripts loaded from previous versions of Nuke default to Local motion estimation for backward compatibility. Adding a new VectorGenerator node to the Node Graph defaults the Method to Regularized motion estimation.

2. To view the generated motion vector fields, click the channels dropdown menu on top of the Viewer and select:
   - **motion** - to view both forward and backward motion vectors.
   - **forward** - to view the forward motion vectors.
   - **backward** - to view the backward motion vectors.

3. If the calculated motion vector fields do not produce the results you're after when used with other nodes (such as Kronos and MotionBlur), try adjusting Vector Detail in the VectorGenerator properties.

   This determines the resolution of the vector field. The larger vector detail is, the greater the processing time, but the more detailed the vectors should be. A value of 1.0 generates a vector at each pixel. A value of 0.5 generates a vector at every other pixel. For some sequences, a high vector detail near 1.0 generates too much unwanted local motion detail, and often a low value is more appropriate.

4. If you're using the Regularized vector calculation method, adjust the Strength control to determine the strength of pixel matching between frames. Higher values allow you to accurately match similar pixels in one image to another, concentrating on detail matching even if the resulting motion field is jagged. Lower values may miss local detail, but are less likely to provide you with the odd spurious vector, producing smoother results.

   **Note:** The default value should work well for most sequences.

5. If you're using the Local vector calculation method, adjust the Smoothness control to improve your results. A high smoothness can miss lots of local detail, but is less likely to provide you with...
the odd spurious vector, whereas a low smoothness concentrates on detail matching, even if the resulting field is jagged.

**Note:** The default value should work well for most sequences.

6. If there are variations in luminance and overall flickering in your Source sequence, enable **Flicker Compensation** to avoid problems with your output.

Examples of variable luminance include highlights on metal surfaces, like vehicle bodies, or bodies of water within a layer that reflects light in unpredictable ways.

**Note:** Enabling **Flicker Compensation** increases rendering time.

7. By default, VectorGenerator analyzes motion based on the brightness of the image. Specifically this is the mean of the red, green, and blue channels. However, you can bias this using the **Weight Red**, **Weight Green**, and **Weight Blue** controls under **Tolerances**. For example, if you set the **Weight Red** and **Weight Green** parameters to zero, VectorGenerator only looks for motion in the blue channel.

8. Once you’re happy with the results, we recommend that you insert a Write node after VectorGenerator to render the original images and the vector channels as an **.exr** file. This format allows for the storage of an image with multiple layers embedded in it. Later, whenever you use the same image sequence, the motion vector fields are loaded into Nuke together with the sequence.
Kronos

Kronos is NukeX's retimer, designed to slow down or speed up footage. It works by calculating the motion in the sequence in order to generate motion vectors. These motion vectors describe how each pixel moves from frame to frame. With accurate motion vectors, it is possible to generate an output image at any point in time throughout the sequence by interpolating along the direction of the motion.

Simple mix of two frames to achieve an in-between frame.

Kronos vector interpolation of the same two frames.

By default, Kronos is set to perform a half-speed slow down. This is achieved by generating a new frame at position 0.25 and 0.75 between the original frames at 0 and 1. Frames are created at a quarter and three quarters instead of zero (an original frame) and a half so as not to include any original frames in the re-timed sequence. This avoids the pulsing that would otherwise be seen on every other frame on a half-speed slow down, and can introduce motion blur.

Kronos only interpolates between input frames, as it cannot extrapolate images before the first frame or after the last frame. A retime with a constant speed $s$ "stretches" the output time by a factor $1/s$, and generates the required images to fill in all intervals between the input frames.

In the following table, $|$ denotes where we need to have images to fill the video sequence. In this example, we assume that the input sequence has 5 frames, denoted by $X$, and we want to retime it with constant speed 0.5 (the default setting). This operation corresponds to stretching the time between the input frames by a factor of 2, which leaves a number of gaps that Kronos fills by generating the images denoted by $O$.

| Samples | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|
| Input | $X$ | $X$ | $X$ | $X$ | $X$ |
| Input Stretched | $X$ | $X$ | $X$ | $X$ | $X$ | $X$ |
| Output | $X$ | $O$ | $X$ | $O$ | $X$ | $O$ | $X$ | $O$ | $X$ |
The output sequence is \texttt{number_of_intervals / speed + 1} frames long. For example, if speed is 0.5 and the input frame is in the range \([1, 10]\), the output sequence length will be \((10 - 1) / 0.5 + 1 = 19\) frames long.

Example Input Speed Calculation

Let's say that the input sequence is \([t_{\text{start}}, t_{\text{end}}]\). Retiming the input sequence in the \textbf{Timing > Input Speed} mode varies the speed at which time flows in the input sequence, so that a point in time \(\tau\), in the input time frame, maps to the output time \(t_o\) as follows:

\[
t_o(\tau) = t_{\text{start}} + \int_{t_{\text{start}}}^{\tau} \frac{1}{s(x)} dx
\]

The start frame \(t_{\text{start}}\) is the same for both the input and output sequence. If the speed parameter \(s\) is not animated, and therefore \(s\) is a constant, we get:

\[
t_o(\tau) = t_{\text{start}} + \frac{1}{s} \int_{t_{\text{start}}}^{\tau} dx = t_{\text{start}} + \frac{1}{s} \cdot (\tau - t_{\text{start}})
\]

So, if we want to know where the last frame of the sequence maps to, we use:

\[
t_o(t_{\text{end}}) = t_{\text{start}} + \frac{1}{s} \cdot (t_{\text{end}} - t_{\text{start}})
\]

Let's look at an example. If the input sequence is in the range \([1, 10]\) and speed is 0.5, the output sequence has a range \([1, 19]\), as follows:

\[t_{\text{o_start}} = 1\]
\[t_{\text{o_end}} = 1 + 2 \cdot (10 - 1) = 19\]
Quick Start

Here's a quick overview of the workflow:
1. Create a node tree with Kronos. See Connecting Kronos.
2. Retime your footage. See Retiming a Sequence.
3. If you're not happy with the results, adjust the vector generation parameters. See Refining the Results.
4. If necessary, add motion blur to the retimed footage. See Adding Motion Blur.

**Tip:** You can check **Use GPU if available** to have the node run on the graphics processing unit (GPU) rather than the central processing unit (CPU).

For more information on the minimum requirements, please see Windows, Mac OS X and macOS, or Linux or refer to the Nuke Release Notes available in Help > Release Notes.

You can select the GPU to use in the Preferences. Press `Shift+S` to open the Preferences dialog, make sure you're viewing the Preferences > Performance > Hardware tab, and set **default blink device** to the device you want to use. You must restart Nuke for the change to take effect.

If you are using a render license of Nuke, you need to add **--gpu** on the command line.
Connecting Kronos

To connect Kronos:

1. Select **Time > Kronos** to insert a Kronos node after the sequence you want to retime. The **Input Range** is set automatically by the source clip when you first create the node. After that, it is only updated if you click **Reset**.

2. If you intend to use **Output Speed** or **Frame** timing, attach a Viewer to the output of the Kronos node.
   OR
   If you intend to use **Input Speed** timing, attach a Viewer before the Kronos node.

3. Enable **Use GPU if available** to render output on the **Local GPU** specified, if available, rather than the CPU.
   For more information on the minimum requirements, please see Windows, Mac OS X and macOS, or Linux or refer to the Nuke Release Notes available in Help > Release Notes.

4. Select the channels you want to apply the retime to using the **channels** dropdown. Kronos retimes **all** channels by default.

5. If your sequence is composed of a foreground object moving over a background, the motion estimation is likely to get confused at the edge between the two. To fix this, add a matte of the foreground region to the **Matte** input. Then, use **Matte Channel** in the Kronos properties to select which component of the matte to use.
   This forces the motion of the foreground to be calculated separately to the motion of the background and so should produce fewer artifacts in the retimed sequence.

6. If the motion in your input sequence has been estimated before (for example, using **VectorGenerator** or third-party software), you can supply the motion vectors to the Background Vectors (**BgVecs**) and Foreground Vectors (**FgVecs**) inputs to save processing time.

   **Note:** The **BgVecs** input appears as an arrowhead on the side of the node.

   If you have separate vectors for the background and foreground, you should connect them to the appropriate inputs and supply the matte that was used to generate them to the **Matte** input. If you have a single set of vectors, you should connect it to the **FgVecs** input.

7. You can set **Output** in the Kronos controls to:
   • **Result** - displays both the foreground and background retimed footage.
   • **Matte** - displays the retimed **Matte** input.
   • **Foreground** - displays the retimed foreground. The background regions outside the matte input may show garbage.
• **Background** - displays the retimed background. The foreground regions inside the matte input may show garbage.

8. If your **Source** sequence is very noisy and interfering with the motion estimation, you can supply a smoothed version of the sequence in the **MoSrc** input. When a **MoSrc** input is connected, motion vectors are calculated from that sequence and applied to the **Source** sequence.

9. Proceed to **Retiming a Sequence**.
Retiming a Sequence

Kronos allows you to retime a sequence by speed or by frame. Speed retimes describe the retiming by a percentage of full speed, either against the output or input frames. For example, the default setting **Output Speed = 0.5** is equal to a 50% retiming on the output frames. Frame retimes describe the retiming in terms of ‘at frame 100 in the output clip, I want to see frame 50 of the source clip’.

**Note:** **Input Range** in the Kronos properties defines which frames are used for the retiming. When you first create the node, the range is automatically set to the frame range of the **Source** clip. If you change the **Source** clip later, you need to click the **Reset** button to make sure **Input Range** matches the current input.

Retiming a Sequence Using Output Speed

To apply a constant retiming:

1. Ensure that the Viewer is connected to the Kronos node and **Timing** is set to **Output Speed**.
2. Select the required interpolation **Method** using the dropdown:
   - **Frame** - the nearest original frame is displayed.
   - **Blend** - a mix between two frames is used for the in-between frame. This is quick to render and is useful when tweaking the timing on a curve before setting the **Method** to **Motion**.
   - **Motion** - vector interpolation is used to calculate the in-between frame.
3. Enter a value for the **Output Speed** control. Values below 1 slow down the clip and values above 1 speed up the clip. The default value is **0.5**, creating a half-speed slow down. Quarter-speed would be **0.25**.
4. Play through the sequence to view the retiming.
5. If you’re not happy with the results of the retiming, try **Refining the Results**.
6. If you’d like to add motion blur to your retimed footage, proceed to **Adding Motion Blur**.

To use varying retimes:

**Tip:** If you want to retime at a certain event in the clip, you may find that using **Input Speed** is easier to predict where keyframes should be placed.

1. Ensure that the Viewer is connected to the Kronos node and **Timing** is set to **Output Speed**.
2. Select the required interpolation **Method** using the dropdown:
   - **Frame** - the nearest original frame is displayed.
   - **Blend** - a mix between two frames is used for the in-between frame. This is quick to render and is useful when tweaking the timing on a curve before setting the **Method** to **Motion**.
   - **Motion** - vector interpolation is used to calculate the in-between frame.
3. Move the playhead to the frame you want to start retiming and enter the required **Output Speed**.
4. Click the Animation button and select **Set Key**.
5. Move the playhead to the frame you want to change the retime value and enter the required **Output Speed**.
   A new keyframe is added automatically.
6. Once you've added all the required keyframes, click the Curve Editor tab to view your keyframes in context of the sequence.
7. If you're not happy with the results of the retime, try Refining the Results.
8. If you'd like to add motion blur to your retimed footage, proceed to Adding Motion Blur.

## Retiming a Sequence Using Input Speed

To apply a constant retime:
1. Ensure that the Viewer is connected to the tree before the Kronos node and **Timing** is set to **Input Speed**.
2. Select the required interpolation **Method** using the dropdown:
   - **Frame** - the nearest original frame is displayed.
   - **Blend** - a mix between two frames is used for the in-between frame. This is quick to render and is useful when tweaking the timing on a curve before setting the **Method** to **Motion**.
   - **Motion** - vector interpolation is used to calculate the in-between frame.
3. Enter a value for the **Input Speed** control. Values below 1 slow down the clip and values above 1 speed up the clip. The default value is 0.5, creating a half-speed slow down. Quarter-speed would be 0.25.
4. Connect the Viewer to the Kronos node, and then play through the sequence to view the retime.
5. If you're not happy with the results of the retime, try Refining the Results.
6. If you'd like to add motion blur to your retimed footage, proceed to Adding Motion Blur.

To use varying retimes:
1. Ensure that the Viewer is connected to the tree before the Kronos node and **Timing** is set to **Input Speed**.
2. Select the required interpolation **Method** using the dropdown:
   - **Frame** - the nearest original frame is displayed.
Retiming a Sequence | Retiming a Sequence Using Frame

- **Blend** - a mix between two frames is used for the in-between frame. This is quick to render and is useful when tweaking the timing on a curve before setting the **Method** to **Motion**.
- **Motion** - vector interpolation is used to calculate the in-between frame.

3. Move the playhead to the frame you want to start retiming and enter the required **Input Speed**.
4. Click the Animation button and select **Set Key**.
5. Move the playhead to the frame you want to change the retime value and enter the required **Input Speed**.
   A new keyframe is added automatically.
6. Once you've added all the required keyframes, connect the Viewer to the Kronos node, and then click the **Curve Editor** tab to view your keyframes in context of the output sequence.
7. If you're not happy with the results of the retime, try **Refining the Results**.
8. If you'd like to add motion blur to your retimed footage, proceed to **Adding Motion Blur**.

Retiming a Sequence Using Frame

To apply a constant retime:

1. Ensure that the Viewer is connected to the Kronos node and **Timing** is set to **Frame**.
2. Select the required interpolation **Method** using the dropdown:
   - **Frame** - the nearest original frame is displayed.
   - **Blend** - a mix between two frames is used for the in-between frame. This is quick to render and is useful when tweaking the timing on a curve before setting the **Method** to **Motion**.
   - **Motion** - vector interpolation is used to calculate the in-between frame.
3. Go to a frame in the timeline, and set **Frame** to the input frame you want to appear at that output position. For a constant retime, you'll need two key frames to retime the clip.
   For example, to slow down a 50 frame clip by half, you can set **Frame** to 1 at frame 1, and to 50 at frame 100. To do a four times slow down, set **Frame** to 1 at frame 1, and to 25 at frame 100.
4. Play through the sequence to view the retime.
5. If you're not happy with the results of the retime, try **Refining the Results**.
6. If you'd like to add motion blur to your retimed footage, proceed to **Adding Motion Blur**.

To use varying retimes:

1. Ensure that the Viewer is connected to the Kronos node and **Timing** is set to **Frame**.
2. Select the required interpolation **Method** using the dropdown:
   - **Frame** - the nearest original frame is displayed.
   - **Blend** - a mix between two frames is used for the in-between frame. This is quick to render and is useful when tweaking the timing on a curve before setting the **Method** to **Motion**.
   - **Motion** - vector interpolation is used to calculate the in-between frame.
3. Go to a frame in the timeline, and set **Frame** to the input frame you want to appear at that output position. You'll need at least two key frames to retime the clip.

For example, to speed up a 50 frame clip and then return to normal speed at the end:

- Set **Frame** to 1 at frame 1,
- Set **Frame** to 10 at frame 20,
- Set **Frame** to 40 at frame 30,
- Set **Frame** to 50 at frame 50.

These keyframes produce the following curve in the Curve Editor.

4. If you're not happy with the results of the retime, try Refining the Results.

5. If you'd like to add motion blur to your retimed footage, proceed to Adding Motion Blur.
Refining the Results

To refine the results:

1. To have the motion vectors displayed in the Viewer, expand the Advanced control and enable Overlay Vectors. Forward motion vectors are drawn in red, and backward motion vectors in blue.

   ![Overlay Vectors enabled.](image)

   **Note:** Motion vectors displayed in the Viewer are added to your output if you don't turn off the overlay before rendering.

2. To set the spacing between motion vectors displayed on the Viewer, adjust Vector Spacing. The default value of 20 means every 20th vector is drawn. Note that Vector Spacing only affects the Viewer overlay rather than the retimed result.

   ![A Vector Spacing value of 4. A Vector Spacing value of 40.](image)

3. Adjust Vector Detail to vary the density of the vector field. The larger vector detail is, the greater the processing time, but the more detailed the vectors should be. A value of 1 generates a vector at each pixel. A value of 0.5 generates a vector at every other pixel. For some sequences, a high vector detail near 1 generates too much unwanted local motion detail, and often a low value is more appropriate.
Areas of unwanted local motion detail. Lower Vector Detail is more appropriate in this case.

4. Vector fields usually have two important qualities: they should accurately match similar pixels in one image to another and they should be smooth rather than noisy. Often, it is necessary to trade one of these qualities off against the other. A high Strength misses lots of local detail, but is less likely to provide you with the odd spurious vector. A low Strength concentrates on detail matching, even if the resulting field is jagged. The default value of 1.5 should work well for most sequences.

5. Set the type of resampling applied when retiming:
   - Bilinear - the default filter. Faster to process, but can produce poor results at higher zoom levels. You can use Bilinear to preview a retime before using one of the other resampling types to produce your output.
   - Lanczos4 and Lanczos6 - these filters are good for scaling down, and provide some image sharpening, but take longer to process.

6. If the overall brightness in your Source footage changes between frames, enable Flicker Compensation. This allows Kronos to take into account variations in luminance and overall flickering, which could otherwise cause problems with your output.

Examples of variable luminance include highlights on metal surfaces, like vehicle bodies, or bodies of water that reflect light in unpredictable ways.
Note that using **Flicker Compensation** increases rendering time.

7. In order to reduce processing time, much of the motion estimation is done on luminance only - that is, using monochrome images. In most cases this is perfectly acceptable, but the parameters in the **Tolerances** group allow you to concentrate on a particular feature in an image by adding bias to individual colours. You may, for example, wish to increase **Weight Red** to allow the algorithm to concentrate on getting the motion of a primarily red object correct, at the cost of the rest of the objects in a shot.

8. If you'd like to add motion blur to your retimed footage, proceed to **Adding Motion Blur**.
Adding Motion Blur

To add motion blur:

1. In the Shutter controls, select a suitable Shutter Time, depending on the amount of blur you wish to add. This sets the equivalent shutter time of the retimed sequence. For example, a shutter time of 0.5 is equivalent to a 180 degree mechanical shutter, so at 24 frames per second the exposure time is 1/48th of a second.

   The larger the value, the more motion blur is produced.

   ![A low Shutter Time value.](image1) ![A high Shutter Time.](image2)

   Alternatively, you can enable Automatic Shutter Time to have Kronos automatically calculate the shutter time throughout the sequence. Note that this only produces motion blur when the retimed speed is greater than the original speed.

2. Render the sequence to see the motion blurred result.

3. If you can see that the motion blur has been created from a few discrete images, try increasing Shutter Samples. This results in more in-between images being used to generate the motion blur and so produces a smoother blur but takes longer to render.
MotionBlur

MotionBlur adds realistic motion blur to a sequence.

Before MotionBlur.  
After MotionBlur.

It uses the same techniques and technology as the motion blur found in Kronos, but presents the controls in a less complex, more user friendly way. However, if you need precise control over the motion vectors used for adding blur, or a large temporal range (that is, a very high shutter time), you should use Kronos.
Quick Start

Here's a quick overview of the workflow:

1. Create a node tree with MotionBlur. See Connecting MotionBlur.
2. Adjust the amount and quality of the motion blur produced. See Adjusting MotionBlur Controls.

Tip: You can check **Use GPU if available** to have the node run on the graphics processing unit (GPU) rather than the central processing unit (CPU).

For more information on the minimum requirements, please see Windows, Mac OS X and macOS, or Linux or refer to the Nuke Release Notes available in Help > Release Notes.

You can select the GPU to use in the Preferences. Press **Shift+S** to open the Preferences dialog, make sure you're viewing the Preferences > Performance > Hardware tab, and set **default blink device** to the device you want to use. You must restart Nuke for the change to take effect.

If you are using a render license of Nuke, you need to add **--gpu** on the command line.
Connecting MotionBlur

To connect MotionBlur:

1. Select **Filter > MotionBlur** to insert a MotionBlur node after the clip you want to add motion blur to.

2. If your sequence is composed of a foreground object moving over a background, the motion estimation is likely to get confused at the edge between the two. To fix this, add a matte of the foreground region to the **Matte** input. Then, use **Matte Channel** in the MotionBlur controls to select which component of the matte to use.
   
   This forces the motion of the foreground to be calculated separately to the motion of the background and so should produce fewer artifacts in the motion blur.

3. If the motion in your input sequence has been estimated before (for example, using **VectorGenerator** or third-party software), you can supply the motion vectors to the Background Vectors (**BgVecs**) and Foreground Vectors (**FgVecs**) inputs to save processing time.
   
   If you have separate vectors for the background and foreground, you should connect them to the appropriate inputs and supply the matte that was used to generate them to the **Matte** input. If you have a single set of vectors, you should connect it to the **FgVecs** input.

4. Attach a Viewer to the output of the MotionBlur node.

5. Proceed to **Adjusting MotionBlur Controls** below.
Adjusting MotionBlur Controls

To adjust MotionBlur controls:

1. Select a suitable **Shutter Time**, depending on the amount of blur you wish to add. This sets the equivalent shutter time of the retimed sequence. For example, a shutter time of 0.5 is equivalent to a 180 degree mechanical shutter, so at 24 frames per second the exposure time is 1/48th of a second.
   
   The larger the value, the more motion blur is produced.

   ![A low and high Shutter Time](image)

2. Set the method of calculating motion estimation vectors:
   
   - **Local** - uses local block matching to estimate motion vectors. This method is faster to process, but can lead to artifacts in the output.
   
   - **Regularized** - uses semi-global motion estimation to produce more consistent vectors between regions.

   **Note:** Scripts loaded from previous versions of Nuke default to **Local** motion estimation for backward compatibility. Adding a new MotionBlur node to the Node Graph defaults the **Method** to **Regularized** motion estimation.

3. Use **Vector Detail** to alter the density of the calculated motion vector field. A value of 1 generates a vector at each pixel, whereas a value of 0.5 generates a vector at every other pixel.

   Bear in mind that higher **Vector Detail** values require longer to process, but can pick up finer movement and enhance your results in some cases.

4. Set the type of re-sampling applied when retiming:

   - **Bilinear** - the default filter. Faster to process, but can produce poor results at higher zoom levels. You can use **Bilinear** to preview a motion blur before using one of the other re-sampling types to produce your output.
   
   - **Lanczos4** and **Lanczos6** - these filters are good for scaling down, and provide some image sharpening, but take longer to process.

5. Render the sequence to see the motion blurred result.
6. If you can see that the motion blur has been created from a few discrete images, try increasing **Shutter Samples**. This results in more in-between images being used to generate the motion blur and so results in a smoother blur.
Working with Lens Distortion

Lens distortion can make composting work more difficult because the characteristics of the lens can cause areas of the footage to warp, making correct placement of assets problematic. One way around this problem is to undistort your plate before starting work on a comp and then redistort and merge only the VFX work to produce the final comp.

The original plate

The undistorted plate and comp

The comp after redistortion

It's a good idea to shoot grids with the same lenses you use to create your footage as this make estimating distortion easier. If you have the lens properties, such as Focal Length and Sensor Size, and a grid shot with the same lens you'll find this process much less painful.

Nuke's LensDistortion node allows you to undistort or distort an image according to several radial distortion models that ship with Nuke or a custom model you define yourself. You can calculate the warp for use on the input image or output the warp to an STMap for use elsewhere in the script.

Quick Start

Here's a quick overview of the workflow:
1. Read in an input sequence, connect it to a LensDistortion node (Transform > LensDistortion), and connect the output to a Viewer.

2. If you have a grid shot using the same lens as your sequence, use Grid Detection to calculate the distortion, and then apply the warp to your sequence. See Estimating Lens Distortion Using a Grid and Removing Lens Distortion from an Image for more information.

3. If you do not have a grid, or Grid Detection did not work, draw features and lines manually to estimate distortion and undistort the sequence. See Estimating and Removing Lens Distortion Using Lines for more information.

4. Add your VFX work to the undistorted sequence.

5. Redistort the VFX work and merge it over the original plate. See Applying Lens Distortion to an Image for more information.
Estimating Lens Distortion Using a Grid

Grid analysis estimates the distortion from a checkerboard or thin line grid. As a general rule, if you have a grid you can use to calculate your lens distortion, you should use grid analysis. Grids are constructed from Features and Links, which can be calculated separately to allow for manual corrections or all at once automatically.

To estimate distortion using a grid:
1. Read in your grid and connect a LensDistortion node, followed by a Viewer.
2. Set the lens Type and Projection using the dropdowns in the LensDistortion Properties panel. In this example, the lens is Spherical and Rectilinear, but there are a number of other presets included.
3. If you’re using a fisheye lens, you need to 'defish' the lens. You can do this by selecting the correct Projection setting, such as Fisheye Equisolid, entering the Focal Length and Sensor Size.

   **Tip:** If you don’t know the Focal Length, switch the Output Mode to Undistort and adjust the Focal Length until the curved lines in the image appear approximately straight. Don’t forget to switch back to Mode > STMap before continuing.

4. Set the Distortion model preset to use for the estimation. Nuke ships with several models for use with CaraVR and 3DEqualizer, as well as a NukeX Classic model.

   The model you choose sets appropriate Distortion Model controls and populates the read only Distortion Equation fields showing the math involved in the estimation.
5. The LensDistortion node adds a keyframe on the current frame automatically, but you can add more using the + or Key All buttons.

Adding keyframes can produce a better result for long sequences, but it takes longer to calculate the result.

6. Click Detect to start the grid calibration. By default, Nuke looks for Features on the grid and then creates Links between those features to create the distortion grid.

   **Tip:** For difficult shots, you can make adjustments between Features and Links detection to improve the results. See Adjusting Grid Detection Parameters for more information.

A grid overlay is displayed when the detection step is complete.

7. Click Solve to estimate the distortion using the grid.

   Nuke estimates the distortion and displays a new overlay and overall Solve Error.
Green lines represent links that fall within the **Threshold** value and red lines those that fall outside the **Threshold**. You can hover over links to display their solve error.

8. Clicking **Solve** again can improve the result in some cases. You can also refine the detection and linking before solving again using the controls on the **Grid Detection** tab. See **Adjusting Grid Detection Parameters** for more information.

9. When you’re happy with the solve, switch the Output **Mode** to **Undistort**.

**Note:** You can also output the distortion as an STMap for use in other images. STMaps contain the pre-computed warp data in the **motion** channel, allowing you apply the warp quickly and easily. See **Working with STMaps** for more information.

Nuke takes the estimated distortion and uses the result to 'straighten' the feature links.
You may notice that the grid expands beyond the bounding box of the footage, which can mean you're losing image data.

10. Set the **Format** control to **Reformat** and select a larger format to include the grid. For example, if your original grid is UHD_4K, you might reformat to 8K_LatLong.

Reformatting the grid deforms it into a characteristic bow tie shape. Any areas of the image where there is no data are overscanned, meaning the last pixel available is duplicated to the edge of the format. The areas at the top and bottom of the image in the center show overscan.

11. If you don’t want to reformat the image, or only a small amount of the image is missing, you can bring more image data into the format bounds using the **Adjust Format** controls.

12. Enable **Override** and then use the **Add Pixels** and **Adjust Aspect** controls to enlarge the format as required.
The new format is called `lens_distortion_adjusted` by default, but you can enter any name you choose in the `adjustedFormatName` control. The new name is then added to the `Project Settings > Format` dropdown after adjustment to avoid changing Nuke's default formats.

**Tip:** You can also preserve overscan data by increasing the bounding box. The `Add Pixels` control allows you to add an equal number of pixels on all sides of the image or you can edit the bounding box manually by enabling `Override` and using the `outputBBox` controls.

13. Proceed to Removing Lens Distortion from an Image or Adjusting Grid Detection Parameters to fine tune your grid.
Adjusting Grid Detection Parameters

The default settings on the LensDistortion node's Grid Detection tab are suitable for a wide range of input images, but in some cases, you'll need to make some adjustments in order to achieve the required coverage of Features and viable Links. You'll need to click Detect again after making changes to the Grid Detection controls to update the results.

**Tip:** For particularly hard solves, you can split the Features and Links detection into two steps, rather than using Grids, which performs both steps automatically.

Feature Detection

The Features in a solve form the building blocks for accurate Links, so it's important to get the maximum coverage possible, while still maintaining quality.

Increasing the maximum Number of Features can improve coverage relatively easily. You can use the Detection Threshold control to reject bad features automatically. If you enter a low detection threshold value, features are detected evenly on all parts of the image, even if they would otherwise have been rejected.

![Low Number of Features vs High Number of Features](image)

The Patch Size control determines how much bias is placed on detecting features on saddle points. High values force features towards saddle points, which might not be desirable.
The **Feature Separation** control sets the distribution of features in relation to each other. High values spread features at even distances over the image. It is important that the features do not cluster together, if this is the case, try increasing this value.

Increasing the separation too far can reduce the number of features detected dramatically, so use caution.

**Link Detection**

The **Links** in a solve depend on solid **Features** detection, so it's important to make sure your features coverage is good before linking. **Links** provide the curvature data from the lens so that the solve can be as accurate as possible.

The **Angle Threshold** control sets how much offset tolerance is allowed between potential features before they can be linked. Try increasing the **Angle Threshold** if there are missing links between features, but higher values can introduce links between features that may be incorrect.

The image on the right shows some potentially poor links on the left-hand side.
The **Distance Threshold** control determines how far apart Links can be before they are merged into a single, averaged link. Try increasing the **Distance Threshold** if features are removed incorrectly after **Links Detection**.

The **Peak Threshold** control determines how much directional tolerance is allowed between potential features before they can be linked. Reducing the Peak Threshold can increase viable links with low contrast lens reference grids.

Proceed to **Removing Lens Distortion from an Image** when you're satisfied with your grid.
Removing Lens Distortion from an Image

After estimating the lens distortion from a reference grid, you can use the warp to remove the distortion from your sequence before performing comp work. The easiest way to do this is to copy and paste the LensDistortion node that was used to generate the warp and connect it to your sequence.

**Note:** You can also use STMaps to remove and apply distortion. See [Working with STMaps](#) for more information.

1. Read in the sequence from which the reference grid was taken.
2. Copy and paste the LensDistortion node containing the grid analysis (described in [Estimating Lens Distortion Using a Grid](#)), then connect it to the sequence and a Viewer.

The distortion is removed from the sequence in the Viewer.
You may notice that the lines expand beyond the bounding box of the footage, which can mean you're losing image data.

3. Set the **Format** control to **Reformat** and select a larger format to include the grid. For example, if your original grid is UHD_4K, you might reformat to 8K_LatLong.

Reformatting the grid deforms it into a characteristic bow tie shape. Any areas of the image where there is no data are overscanned, meaning the last pixel available is duplicated to the edge of the format. The areas at the top and bottom of the image in the center show overscan.

You can now apply your VFX work before redistorting the plate back into its original state.

4. If you don't want to reformat the image, or only a small amount of the image is missing, you can bring more image data into the format bounds using the **Adjust Format** controls.

5. Enable **Override** and then use the **Add Pixels** and **Adjust Aspect** controls to enlarge the format as required.
**Note:** You can use expressions to control the number of pixels added as a percentage of the format by right-clicking the **Add Pixel** control and selecting **Add Expression**. For example, `input.width*0.05` adds pixels equal to 5% of the input format's width.

The new format is called **lens_distortion_adjusted** by default, but you can enter any name you choose in the **adjustedFormatName** control. The new name is then added to the **Project Settings > Format** dropdown after adjustment to avoid changing Nuke's default formats.

**Tip:** You can also preserve overscan data by increasing the bounding box. The **Add Pixels** control allows you to add an equal number of pixels on all sides of the image or you can edit the bounding box manually by enabling **Override** and using the **outputBBox** controls.

6. You can now track, matchmove, composite, and so on in the undistorted space before redistorting the image. See **Applying Lens Distortion to an Image** for more information.
Estimating and Removing Lens Distortion Using Lines

Line analysis estimates the distortion from lines drawn manually along features in the input that are known to be straight. This can be useful if there is no grid available or if you have a grid for your sequence but the grid analysis failed, for instance due to bad lighting.

The LensDistortion node has five tools for selecting and drawing features and lines in the Viewer:

- **Select** - a multi-purpose tool used to select, move, and delete features and lines.
- **Select Feature** - used to select, move, and delete features. You cannot affect lines using this tool.
- **Select Line** - used to select, move, and delete lines. You cannot affect features using this tool.
- **Add Feature** - used to add, select, move, and delete features. You cannot affect lines using this tool.
- **Add Line** - used to add and delete lines. You can connect existing features or create new lines using this tool.

To estimate lens distortion using lines, do the following:

1. Examine the source footage for distorted lines that are known to be straight. If you have a grid, this is simple, but real life examples include walls, railings, roads, and so on.
2. Select the **Add Feature** tool and start to add points along the distorted lines by clicking in the Viewer.

**Tip:** You can also draw lines immediately using the **Add Line** tool, but adding features first can be more accurate.
3. When the features are complete for a particular line in the image, switch to the **Add Line** tool.

4. Select the first feature in the Viewer and then click the next feature to add a line between the two.

   **Tip:** When you’ve finished drawing a line, either select a different tool or press **Enter** to finish the current line before starting another.

5. The solve requires at least as many lines as there are **Distortion Parameters** to calculate the distortion. In the case of **NukeX Classic**, two vertical and two horizontal to cover the **Denominator** and **Centre** parameters. Good line drawing practices include:
- **Drawing lines with three or more features** - lines with only two points are ignored.
- **Drawing longer lines** - they contain more useful information on the curvature resulting from lens distortion.
- **Distributing lines evenly** - avoid biasing the solve by covering as much of the image as possible.

The following example shows a typical set of lines.

6. Click **Solve** to calculate the distortion, there's no need to run the **Detect** step with manual features and lines.

   Green lines represent links that fall within the **Threshold** value and red lines those that fall outside the **Threshold**. You can hover over links to display their solve error.

   **Tip:** Clicking **Solve** again can improve the result in some cases.

7. When you're happy with the solve, switch the Output **Mode** to **Undistort**.
Note: You can also output the distortion as an STMap for use in other images. STMaps contain the pre-computed warp data in the motion channel, allowing you apply the warp quickly and easily. See Working with STMaps for more information.

Nuke takes the estimated distortion and uses the result to 'straighten' the feature links.

You may notice that the lines expand beyond the bounding box of the footage, which can mean you're losing image data.

8. Set the Format control to Reformat and select a larger format to include the grid. For example, if your original grid is UHD_4K, you might reformat to 8K_LatLong.

Reformatting the feature lines deforms it into a characteristic bow tie shape. Any areas of the image where there is no data are overscanned, meaning the last pixel available is duplicated to the edge of the format. The area at the bottom of the image in the center shows overscan.

You can now apply your VFX work before redistorting the plate back into its original state.
9. If you don’t want to reformat the image, or only a small amount of the image is missing, you can bring more image data into the format bounds using the Adjust Format controls.

10. Enable Override and then use the Add Pixels and Adjust Aspect controls to enlarge the format as required.

**Note:** You can use expressions to control the number of pixels added as a percentage of the format by right-clicking the Add Pixel control and selecting Add Expression. For example, `input.width*0.05` adds pixels equal to 5% of the input format's width.

The undistorted format  

The adjusted format

The new format is called `lens_distortion_adjusted` by default, but you can enter any name you choose in the `adjustedFormatName` control. The new name is then added to the Project Settings > Format dropdown after adjustment to avoid changing Nuke’s default formats.

**Tip:** You can also preserve overscan data by increasing the bounding box. The Add Pixels control allows you to add an equal number of pixels on all sides of the image or you can edit the bounding box manually by enabling Override and using the outputBBox controls.

11. You can now track, matchmove, composite, and so on in the undistorted space before redistorting the image. See Applying Lens Distortion to an Image for more information.
Applying Lens Distortion to an Image

The LensDistortion node's Redistort mode allows you to warp your VFX work to match the original plate, which avoids the increased processing time the filter hit on the entire image would incur.

**Note:** You can also use STMaps to remove and apply distortion. See [Working with STMaps](#) for more information.

1. Copy and paste the LensDistortion node containing the grid analysis (described in [Estimating Lens Distortion Using a Grid](#)), then connect it downstream of your VFX work.
2. Change the **Mode** control to **Redistort** to reverse the warp that was applied during the undistortion of the plate.
3. Merge the redistorted VFX work back over the plate to complete the comp.

The undistorted plate and comp

The comp work after redistortion
Working with STMaps

STMaps allow you to warp an image or sequence according to the pre-calculated distortion from a LensDistortion node, stored in the motion layer. The motion channels represent the absolute pixel positions of an image normalized between 0 and 1, which can then be used on another image to remove or add distortion without the warp estimation step. See Estimating Lens Distortion Using a Grid and Estimating and Removing Lens Distortion Using Lines for more information on warp estimation.

To remove distortion using the STMap node, do the following:

1. Read in your sequence and add an STMap node downstream. Make sure the src input is connected to the sequence.
2. Copy and paste the LensDistortion node containing the grid analysis with the Mode control set to STMap, OR
   Read in the file containing the motion layer, if you wrote it out separately.
3. Connect the stmap input to the source of the motion layer.
4. Set the STMap UV channels control to the motion layer to remove the distortion.
   You can now track, matchmove, composite, and so on in the undistorted space before redistorting the image.
5. After the comp work is complete, you can warp the VFX work by applying the backward layer in the UV channels control using another STMap node.
6. The final step is merging the warped VFX work back over the original image to create the comp.

The output from nodes placed before the second STMap node are distorted by the same warp as the sequence.

The undistorted plate and comp

The comp after redistortion
Tracking with PlanarTracker

PlanarTracker is a powerful tool for tracking surfaces that lie on a plane in your source footage. Planar tracking is often better than tracking individual points (with the Tracker node for instance) as it takes a great deal more information into account and gives you a more accurate result. A rigid surface, like a wall or a side of an object, are good planes to track. You can use your tracking results to replace the tracked plane with another image for instance. You can define the region to track by creating and animating roto shapes.
Tracking a Plane

Before you can track a plane, you need to draw one using the Roto node.

Drawing a Plane to Track

You can use the PlanarTracker to track rigid objects and objects that deform slightly throughout the track. As the PlanarTracker tries to fit a plane to the object to be tracked, rigid objects obtain better track results than objects that deform.

For instance, a wall or a flat side of an object are good planes, but you can also get good results tracking faces or people. It's also important that the plane you’re tracking has some texture and that the plane isn't completely obscured at any point of the tracking. Tracking surfaces without texture and few features to track is not likely to produce good results.

1. You can do one of the following:
   • Insert a PlanarTracker node by either selecting Transform > PlanarTracker, or by pressing tab, typing PlanarTracker, and pressing Return. This inserts a Roto node that is already in PlanarTracker mode. You can use this to draw a Bezier shape, which is automatically added as a track object. The shape's boundary appears in purple, denoting this, and the shape is automatically added to a layer called PlanarTrackLayer1 in the stroke/shape list.
   OR

   • Create a Roto or RotoPaint node and use it to draw a Bezier shape around the plane you want to track. Your new shape’s boundaries appear in red in the Viewer, and a Bezier shape item appears in the stroke/shape list. The shape remains as a normal roto shape until it is converted into a track object.
2. If you’re drawing more than one shape, you can arrange them in the stroke/shape list to tell PlanarTracker that they are different layers. Order your shapes from closest to camera (top of the list) to furthest away (bottom of the list), and PlanarTracker automatically holds out any track layers above the current one.

3. Make sure you’re still on the same frame as you used to draw the Bezier shape.

**Note:** When you select a frame to draw a roto shape on, that frame becomes your reference frame. When you proceed with tracking your plane it’s important that you’re always starting on the same reference frame. Move to your reference frame using the **Go to Reference Frame** button in the course of the process you may decide to change your reference frame. You can do this by clicking **Set Reference Frame** in the Viewer.

### Tracking the Footage

You can now proceed to track the plane you’ve drawn:

Depending on whether you have chosen to insert a PlanarTracker node, or a Roto (or RotoPaint) node, do one of the following:

If you have chosen to insert a PlanarTracker node:
1. Use the tracking tools above the Viewer to track the Roto shape.

![Tracking tools interface](image)

2. See Tracker Menu Options in the Viewer for more information.

If you have chosen to insert a Roto or a RotoPaint node:

1. Right-click the Roto shape and select one of the following options:
   - **planar-track this shape** - Select this to convert the roto shape into a track object that doesn't contain any tracking data (if it's not already).
   - **planar-track this shape (fwd)** - Select this to track your shape while the track is played forwards.
   - **planar-track this shape (bkwd)** - Select this to track your shape while the track is played backwards.

2. If you selected one of the latter two options, your shape is now tracked throughout the timeline. A progress dialog is displayed while the shape is tracked.

   If you selected the first option to convert your roto shape into a track object without any tracking data, you can use the tracking tools above the Viewer to track forwards or backwards. See Tracker Menu Options in the Viewer for more information. When you select any of the options above the tracker menu tools are displayed above the Viewer.

   The shape is now automatically added to a layer called **PlanarTrackLayer1** in the stroke/shape list and is displayed in purple in the Viewer, signifying that it has been converted into a track object.

3. In the stroke/shape list, a purple rectangle appears in the **PT** column next to your shape(s). You can toggle the purple rectangle on and off to revert the shape to a normal roto shape instead of a track object.

   **Note:** You can only toggle a shape between being a roto shape or a track object, after converting the Roto shape into a track object.

4. Select the **Tracking** tab in the Roto node properties to display the tracking data, as shown below.
Tracker Menu Options in the Viewer

You can use the tracker menu options to perform more advanced tracking. These options are only visible in the Viewer after you have converted a shape into a track object.

Track Motion Controls

Before tracking an object, you can use the track motion controls to specify the type of movement that the PlanarTracker can expect. The following options are available:

- **Translation** - Select this so the tracker expects translation.
- **Rotate** - Select this so the tracker expects rotation.
- **Scale** - Select this so the tracker expects scaling.
- **Shear** - Select this so the tracker expects shearing.
• **Perspective** - Select this so the tracker expects changing perspective.

## Tracking Controls

Use the **Tracking buttons** to track backwards or forwards throughout the whole footage, or if you want, select a specified frame range.

You can also track on a frame by frame basis. For example, if the plane you’re tracking is visible throughout the footage, forward tracking might be all you need. If the plane is only partially visible over a part of the footage though, it might be a good idea to first track the part with the whole plane visible, and then track the rest of the footage separately.

With the Tracking controls you can track and re-track your footage. For more information see [Tracking and Stabilizing](#).

## Clear Tracking Data Controls

You can clear tracking information that you’ve already created with the clear buttons.

- **clear all** - clear all tracking information created by PlanarTracker.
- **clear backwards** - clear all tracking information backwards from the current frame.
- **clear forwards** - clear all tracking information forwards from the current frame.
Planar Tracker Surface Controls

After you have tracked your shape or shapes, you can review your results by scrubbing back and forth on the timeline. You can adjust the planar surface shape if, for example, your roto shape drifts in the course of the tracking, or you simply want to change it. To do this, you can use the planar surface controls.

- **center planar surface** - Select this to center the planar surface in the Viewer during playback.
- **display planar surface** - Select this to display the boundary of the planar surface.
- **enable planar surface editing** - Select this so that you can edit the planar surface by dragging the corner points in the Viewer.
- **display grid** - Select this to display a grid over the planar surface.
- **set planar surface to image bounding box** - Select this to change the planar surface to be the same as the image bounding box.

Reference Frame Controls

You can use the reference controls above the Viewer to either go to the current reference frame, or change the reference frame to the current frame.

Keyframe Controls

You can set and remove keyframes, and go to previous or next keyframes using the keyframe controls above the Viewer.
Layer Dropdown

You can use the layer dropdown to quickly select various PlanarTracker layers, or to add a new layer.

CornerPin Dropdown

You can use the CornerPin dropdown to insert a CornerPin node. The CornerPin2D tool is designed to map the four corners of an image sequence to positions derived from tracking data. In practice, this allows you to replace any four-cornered feature with another image sequence.

The CornerPin dropdown provides you with several different kinds of CornerPin you can choose from:

- **relative** - to warp the image according to the relative transform between the current frame and the reference frame. The image remains unchanged in the reference frame. You can also pick the baked version of the relative CornerPin node. A baked node has the keyframe values copied from PlanarTracker, rather than having them expression linked.
• **absolute** - to use the to and from controls to place the image exactly within the selected plane. This may skew the image throughout the footage. This attaches and automatically sets its format to the dimensions of any currently selected node. You can also pick the baked version of the absolute CornerPin node. A baked node has the keyframe values copied from PlanarTracker, rather than having them expression linked.

• **stabilize** - to apply an inverse transform on the image, which effectively locks the image in its place and stabilizes it in the footage. This type of corner pinning is most useful for drift corrections and making sure your tracking results are reliable. You can also select **Tracker** to create a baked Tracker node for further reducing jitter. You can also pick the baked version of the stabilize CornerPin node. A baked node has the keyframe values copied from PlanarTracker, rather than having them expression linked.

• **Tracker** - creates a Tracker node with four tracks set to each of the tracked corner points. These tracked points give you access to the Tracker's comprehensive functions, such as track averaging and enhanced smoothing controls. See **Tracking and Stabilizing** for more information.

---

### Tracking Tab Menu Options

Select the **Tracking** tab in the Roto node's properties to access more tracking options.

#### Settings

<table>
<thead>
<tr>
<th><strong>track channels</strong></th>
<th>Use this dropdown to select the channel set on which you want to track.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>image channels</strong></td>
<td>Use this to select an individual channel from the channel set. You do not need to select an individual channel, and instead can leave it as none, the default.</td>
</tr>
<tr>
<td><strong>pre-track filter</strong></td>
<td>Before image patches are compared, the selected filter is applied. You can select one of the following options:</td>
</tr>
<tr>
<td></td>
<td>• <strong>none</strong> - This disables all pre-filtering, which allows you to have full control of tuning the input image for tracking.</td>
</tr>
<tr>
<td></td>
<td>• <strong>adjust contrast</strong> - This stretches the image contrast to better suit the tracking algorithm. This option is recommended.</td>
</tr>
<tr>
<td></td>
<td>• <strong>grayscale</strong> - This converts any input RGB channels to a grayscale image for faster processing.</td>
</tr>
<tr>
<td><strong>adjust for luminance</strong></td>
<td>If your footage changes brightness over time, either gradually or</td>
</tr>
</tbody>
</table>
changes

suddenly, enabling this option performs extra pre-filtering to help compensate for the changes. This may make the track slower and less accurate and therefore should only be selected when you need to handle changes in luminance.

clamp super-white, sub-zero footage

Select this to clamp the tracking patch pixel values to lie between 0 and 1.

Note: If you want to track using the full dynamic range of your super-white or sub-zero footage, disable this option.

hide progress bar

As it is possible to stop a track using the tools above the Viewer, you can choose to hide the progress dialog that appears when you track a shape.

Export

You can use the CornerPin dropdown to select the type of corner pin node you want to insert. See CornerPin Dropdown for more information about the different kinds of CornerPin. After you have selected the required type of CornerPin, you can press Create to insert it.

Select the link output checkbox to link to the PlanarTracker output so that exported nodes are updated with the track.

Correction

TheCornerPin points are populated automatically when you track an object. When you draw a roto shape and convert it into a track object, Nuke automatically places 4 corner pins around the shape. These are the points that are tracked.

You can correct the four automatically placed points by offsetting any or all of the four points. To offset a point, simply click and drag it in the Viewer to the correct position.

Reference Frame

This is the frame used as a reference to compare all other frames containing tracking data. It defaults to the first frame used to track. You can change this by entering a new frame number in the reference frame input field.
Reusing a Track Result

You can reuse an entire tracked plane or a single track.

Reusing a Tracked Plane

You can use a plane you've already tracked to analyze a larger plane situated on the same plane.
1. Make sure you're in the same frame as you used to draw the previous shape.
2. Draw a shape on the same plane as your previously tracked shape. Make sure the new shape is in the same PlanarTrackLayer as the first one in the stroke/shape list.
3. PlanarTracker has now placed your new roto shape on the same plane as the old one. If you now scrub in the timeline, you should find that your new shape sticks with the camera movement.

Reusing a Single Track

You can also reuse your tracked points to verify that the overall results are sticking to your footage.
1. Track a plane as described under Tracking a Plane, and adjust the results if necessary.
2. When you're happy with the tracking results for that plane, you can drag a corner of your tracked shape and place it on a specific detail that may not be getting tracked correctly. Note that you're moving the roto shape locally, and not affecting the actual track results.
3. Scrub through the frames to see if the point stays on the detail as you expect.
4. If drifting occurs, drag the point to correct it.
Placing an Image on the Planar Surface

When you've tracked a planar surface on your footage, you might want to place an image on it. To place an image on the tracked planar surface, do the following:

1. Ensure you're on the reference frame you've drawn the roto shape on, and select the PlanarTrackLayer you want to use in the Create New Track dropdown on top of the Viewer.

2. Check show plane and correct plane on the properties panel to make your planar surface visible and to enable modifying it. You can also click the corresponding buttons above the Viewer.

3. A rectangle indicates your planar surface in the Viewer. If the rectangle is very large, you can click the Resize Planar Surface to Image button in the Viewer.

4. Drag the corners of the rectangle to cover the area over which you want to place an image. Click the Show grid lines button in the Viewer to use a guide grid in positioning your rectangle.
This shows grid lines corresponding with the current plane and it helps with realigning your plane.

5. Scrub in the timeline to make sure the planar surface sticks to the area you want.

If you need to adjust it, you can do one of the following:

• Adjust the points in the reference frame to change the planar surface over the whole footage. This way you're adjusting the actual dimensions of the planar surface rectangle. The rectangle appears in yellow.

• Adjust the points in other frames to change the planar surface in the current frame and its adjacent frames. This way you're correcting small drifts in the planar surface rectangle without changing its real dimensions. The rectangle appears in blue.
6. The values in the **CornerPin points** show how your plane has warped from the reference frame to the current frame. Adjust them in the **Curve Editor** by selecting **Curve Editor** tab above the Node Graph.

**Note:** While you can drag the **CornerPin points** data to another node or control by simply dragging selecting all the points and dragging on any of the animation buttons. If you press **Ctrl/Cmd+drag**, this creates an expression link to the **CornerPin points** data instead of placing them in the new location.

7. Read in the image you want to place on the tracker planar surface.
8. To convert the new image to the same format, insert a Reformat node and select the correct image format from the **output format** dropdown.
9. You can now add a CornerPin node to help you place an image on the plane. Click the **Create CornerPin2D Node** dropdown in the Viewer (or use the **Export** dropdown in the properties panel) and select the type of corner pinning you want to use.

![CornerPin2D dropdown](image)

See the 'Tracker Menu Options in the Viewer' section in **Tracking a Plane** for more information about the different types of CornerPin.

10. Insert a Merge node and connect the CornerPin node to the **A** input, and the Roto node to the **B** input. Your node tree should now look similar the one shown below:
11. You can now make any other required changes to your image. For example, you can adjust the mix control in the Merge node's properties, to change the opacity of the image.

12. When you're happy, close all the node property panels to see the result clearly in the Viewer. You can scrub through the timeline and make sure the position of the image is correct throughout.
Camera Tracking

Nuke’s CameraTracker node is designed to provide an integrated camera tracking or match-moving tool, which allows you to create a virtual camera whose movement matches that of your original camera. Tracking camera movement in a 2D footage enables you to add virtual 3D objects to your 2D footage.

Introduction

With the CameraTracker node, you can track the camera motion in 2D sequences or stills to create an animated 3D camera or a point cloud and scene linked to the solve. You can automatically track features, add User Tracks or tracks from a Tracker node, mask out moving objects using a Bezier or B-spline shape, and edit your tracks manually. CameraTracker can solve the position of several types of cameras as well as solve stereo sequences.

Quick Start

The tracking process is outlined below, whether you intend to track a sequence or a set of stills:

1. Connect the CameraTracker node to the sequence you want to track. See Connecting the CameraTracker Node.

2. Mask out any areas of the image that may cause CameraTracker problems, such as movement within the scene or burn-in. See Masking Out Regions of the Image.

3. If you're tracking stereoscopic or multi-view images, set the Principal View on the CameraTracker or Settings tabs. See Working with Multi-View Scripts for more information.

4. Set the camera parameters, such as Focal Length and Film Back Size, if they are known. These are described under Setting Camera Parameters.

5. Set the Source dropdown to Sequence or Stills, and then:
   - If you intend to track a continuous Sequence of frames, set the tracking preferences using the Settings tab Features and Tracking controls. See Tracking in Sequence Mode for more information.
   - If you're using Stills, you can track all frames in the same way as sequence tracking, or a subset of Reference Frames using the +/- keyframe buttons above the Viewer or in the properties panel. See Tracking in Stills Mode for more information.

6. You can place User Tracks to improve difficult solves, use an entirely manual tracking approach, or set 3D survey points. You can use 3D survey points to tie your sequence to a known 3D world, such as those created using stills. See Working with User Tracks for more information.
Tip: 3D survey points have replaced the ProjectionSolver workflow, but you can still add ProjectionSolver nodes by pressing X in the Node Graph and entering `ProjectionSolver` as a Tcl command.

7. Click **Track** to begin tracking the sequence.
8. Solve the Camera position by clicking **Solve** and refine it, if necessary. For more information, see [Solving the Camera Position](#).
9. Set the ground plane, if required, and adjust your scene. See [Adjusting the Scene](#).
10. Select what to export from the solve using the **Export** dropdown and click **Create**.
    You can export an animated camera, a stereoscopic or multi-view rig, a 3D scene and point cloud, lens distortion, or cards. See [Using Solve Data](#).
11. If you have multiple footage sources of the same scene or content available, you can also use survey points to solve each of your sources and then register them all in the same world. See [Combining Solves](#).
12. Add your 3D virtual objects to the footage. See [Placing Objects in the Scene](#).
13. By default, any 3D objects you added to your footage do not have lens distortion applied to them. As a result, they can look like they weren't shot with the same camera. To fix this, see [Accounting for Lens Distortion](#).
Connecting the CameraTracker Node

To connect the CameraTracker node:

1. Read in and select the clip you want to track.
2. Click 3D > CameraTracker.
3. If you want to omit a part of the scene from being tracked, connect a matte to the Mask input. Note that, unlike the Source input, this input is hidden and appears as a small triangle on the left hand side of the node. For more information about masking, see Masking Out Regions of the Image.
4. Click Image > Viewer to insert a Viewer node and connect it to the CameraTracker node.
Masking Out Regions of the Image

Tracking works best on fixed, rigid parts of the scene so that each track can create a single, fixed 3D point. The solver uses these 3D points to work out the camera path. Moving elements and burn-ins do not have a fixed 3D point in the world and should be masked out before tracking.

To mask regions of your sequence, attach a matte to the Mask input to define image regions that should not be tracked. You can also use the source input's alpha channel as a matte.

1. If you want to use a separate matte for masking, connect a Roto node to the CameraTracker Mask input.
2. Scrub through the sequence and keyframe the roto shapes to cover the areas you don’t want to track.

You don’t have to be too accurate with the mask, it’s only intended to cover areas that are likely to cause CameraTracker problems. For example, in the image shown, the actors and the copyright burn-in are masked.

3. In the Properties panel, set Mask to the component you want to use as a mask:
   - None - Track features in the whole footage.
   - Source Alpha - use the alpha channel of the source clip to define which areas to ignore.
   - Source Inverted Alpha - use the inverted alpha channel of the source clip to define which areas to ignore.
   - Mask Luminance - use the luminance of the mask input to define which areas to ignore.
• **Mask Inverted Luminance** - use the inverted luminance of the mask input to define which areas to ignore.

• **Mask Alpha** - use the mask input alpha channel to define which areas to ignore.

• **Mask Inverted Alpha** - use the inverted mask input alpha channel to define which areas to ignore.

4. Track as normal using the automated Analysis **Track** button. See **Tracking in Sequence Mode**.

**Note:** There is no need to mask areas of the image when tracking manually - you specify where User Tracks are placed.
Working with Multi-View Scripts

CameraTracker can track and solve stereoscopic or multi-view projects in much the same way as single view projects.

1. Connect the CameraTracker node as described in Connecting the CameraTracker Node.
2. Use the CameraTracker or Settings tab Principal View dropdown to select the view used to create the tracks. Tracks in any other views present in the script are calculated from the Principal View.
3. Follow the workflow described under Camera Tracking to track and solve the camera.
4. If you create any User Tracks, note that they currently only support at most two views. To select which views are to be used as your left and right views for user tracks, use the User Track Views control on the UserTracks tab of the CameraTracker properties.
5. To visualize both cameras for your views, create a multi-view rig with a camera for each of your views by selecting Camera rig from the Export section of the CameraTracker tab. See Creating Camera Nodes for more information.
Setting Camera Parameters

Camera settings relate to the physical aspects of the camera used on set. Accurate physical camera data produces a better camera track and solution.

1. Select the motion type of the on set camera from the CameraTracker tab Camera Motion dropdown menu. This is linked to a control of the same name on the Settings tab.
   - Rotation Only - select this option if the camera is static and rotating, for example, if you’re using a tripod mounted camera for nodal pans.
   - Free Camera - select this option if the camera is both translating and rotating.
   - Linear Motion - select this if the camera has a straight, linear path.
   - Planar Motion - select this if the camera has a flat path, moving in a two-dimensional plane only.

2. If you have already used a separate LensDistortion node (see Working with Lens Distortion on page 1) to remove lens distortion from your footage, you can leave the Lens Distortion control set to No Lens Distortion.
   Otherwise, set Lens Distortion to Unknown Lens before you solve the camera position to force CameraTracker to calculate the distortion.

3. Select the Focal Length type for the camera from the dropdown menu:
   - Known - select this option if the focal length is available and enter a value in the Length control.
   - Approximate Varying - select this option if an approximate focal length is available and enter keyframed focal length values in the Length control.
   - Approximate Constant - select this option if an approximate focal length is available and there is no zoom, and enter a focal length value in the Length control.

   **Note:** CameraTracker attempts to refine the focal length during the solve if you select an Approximate option.

   - Unknown Varying - select this option if the focal length is unknown and changing.
   - Unknown Constant - this is the default option. Use this option if the focal length is unknown and there is no zoom.

4. Either choose a Film Back Preset from the dropdown to populate the Film Back Size controls, or if your camera isn’t in the list, enter the Film Back Size manually.
Tip: You can add your own defaults by editing the ..//NukeScripts/camerapresets.py file in the Nuke installation package.
Tracking in Sequence Mode

In **Sequence** mode, CameraTracker tracks the footage attached to the **Source** input and defines a set of 2D feature tracks that correspond to fixed points in the scene.

**Note:** If you intend to remove lens distortion manually using a separate LensDistortion node, you should do that before you track the sequence. See *Working with Lens Distortion*. Otherwise, set **Lens Distortion** to **Unknown Lens** before you solve the camera position to force CameraTracker to calculate the distortion.

Before tracking, use CameraTracker's properties panel to control Viewer output and determine tracking behavior:

1. On the **CameraTracker** tab, ensure that **Source** is set to **Sequence**.
2. If you intend to mask out parts of your image, set the **Mask** control to the matte source.
   For example, if you're using the alpha channel from a Roto node attached to the **Mask** input, select **Mask Alpha**. For more information, see *Masking Out Regions of the Image*.

**Note:** For stereoscopic or multi-view footage, set the **Principal View** on the **CameraTracker** or **Settings** tab to the main tracking view. Any masking should be applied to this view, which is then used to calculate the secondary camera.

3. Use the **Range** dropdown to determine which frames are analyzed:
   - **Input** - the default value, sets the frame range to the length of the sequence attached to the **Source** input.
   - **Global** - sets the tracking frame range to the range set in the Project Settings **frame range** controls.
     If no frame range is defined, the frame range of the first image you read in is used as the **Global** frame range.
   - **Custom** - sets a tracking range of frames described by the **from** and **to** fields.
4. On the **Settings** tab, define the starting points for feature tracking:
   - **Number of Features** - define the number of features you want to track in each frame.
     Ideally, you should use more than 100 tracks per frame. In most cases, the default 150 should be sufficient, but in difficult sequences you may consider using a higher number.
   - **Detection Threshold** - set the distribution of features over the input image.
If you enter a low detection threshold value, features are tracked evenly on all parts of the image and vice versa.

Low threshold distribution  High threshold distribution

- **Feature Separation** - set the distribution of features in relation to each other.
  To force feature separation and spread features evenly over the image at even distances, enter a high feature separation value.

Low separation value  High separation value

5. Check **Refine Feature Locations** to lock detected features to local corners. If you activate this, CameraTracker finds the closest corner point in your footage and locks the feature to it.

6. Check **Preview Features** to view the current distribution of tracking features.
  Preview comes in handy when you want to tweak the tracking parameters further before tracking - it updates dynamically when controls are adjusted.
  It's important to make sure that the previewed features are distributed evenly and not clustered together too densely before tracking using the distribution and separation controls.
7. Once you're happy with the feature distribution in the preview, click **Track** to begin analyzing the sequence.

8. CameraTracker begins reading frames sequentially and tracking the features present. Tracks that don't meet the quality thresholds set in the **Settings** tab **Tracking** controls are reseeded so that the number of tracks remains constant.

See **Troubleshooting Sequence Tracks** for more details on adjusting these controls.

When the playhead reaches the end of the sequence, it begins a verifying pass by reading the frames sequentially backwards. Any tracks that were reseeded due to error are tracked back past the point where they were created, if they remain viable.
Viewing Track Data

Once tracking is complete, scrub through the timeline to examine the tracked features. The points represent the features and the vectors the track length calculated for the associated feature. Hover over a point to display its length, in frames.

Consider masking out areas where tracks are consistently seeded and rejected, such as highly reflective surfaces, and adjusting the Settings tab Tracking controls to increase track reliability.

See Masking Out Regions of the Image and Troubleshooting Sequence Tracks for more information.

Reviewing AutoTracks Curves

Detailed tracking information is displayed in curves in the properties panel on the AutoTracks tab. You can select all track curves at once or get a more detailed view of a single curve, such as num tracks.

Tip: Pressing F with focus on a curve maximizes the selected curve in the space available.
Selecting all track curves.

Framing a single curve.

Curves can indicate areas of the sequence where tracking encountered problems. For example, the number of tracks curve on the right shows a significant dip on the current frame, as indicated by the playhead position.

In this instance, you could try adjusting the **Number of Features** or **Minimum Length** and retracking. See **Troubleshooting Sequence Tracks**
Troubleshooting Sequence Tracks

Some sequences are inevitably going to cause problems. There are a number of pre-tracking checks and post-tracking refinement controls to assist CameraTracker.

Pre-Tracking Checks

• Play through the sequence before tracking and mask out any problem areas in the scene. Large moving objects can confuse the tracking and solving process as can appear to be fixed areas of the scene.

  See Masking Out Regions of the Image for more information.

  **Tip:** If you're compositing on moving elements, such as faces, try tracking and solving on just the moving element using masks. You'll then get a camera that moves around the element as if it was fixed in a single position.

• Textureless areas of the sequence, such as greenscreens, can cause features to cluster together in other areas, affecting the track and solve.

  Turn on Preview Features and use the distribution and separation controls to even out the features. See Tracking in Sequence Mode

• Regular edges, such as striped patterns, in the scene can confuse CameraTracker.

  Again, turn on Preview Features and use the Detection Threshold to force CameraTracker to use corner-like images textures. See Tracking in Sequence Mode

• You can improve tracking data by adding User Tracks manually, see Working with User Tracks for more information.

Reviewing and Refining Tracking Data

You can use the threshold controls on the AutoTracks tab to dynamically reject tracks and remove them, before solving, to improve accuracy. Using the num tracks curve as an example:

1. Click the AutoTracks tab in the properties and select the num tracks curve.

  **Tip:** Press F in the curve display to fit the selected track(s) to the available screen space.
In the example, you can see that the number of tracks has dropped significantly around frames 180-200.

2. Move the Viewer playhead to the affected area and examine the tracked features in the Viewer.

3. If you increase the Min Length control in the properties panel, you’ll start to see tracks turn red as they fall below the threshold.

4. Click Delete Rejected to remove all the tracks that fall below the specified threshold.

   Tip: You can also remove tracks manually, by selecting them in the Viewer, right-clicking, and choosing tracks > delete selected.

5. You can then retrack the affected frame range by clicking Update Track. See Retracking Partial Frame Ranges for more information.
Re-tracking Using Tracking Settings

Once you have refined the tracking data, review the sequence by scrubbing the playhead to make sure you have tracks across the image on the rigid fixed background. If you see lots of tracks that drift or jump, select them manually, right-click, and go to **tracks > delete selected**.

![Image showing the tracking interface with options for delete selected, extract user track, and center selected.]  

You can also add User Tracks to improve tracking data before solving, see **Working with User Tracks**.

You can improve a set of feature tracks using the following controls and then retrack to improve your chances of getting a good solve:

- **Minimum Length** - sets a threshold value for the minimum acceptable track length. Tracks that fail to stick to the associated feature for this amount of frames are rejected.

  In long slow camera moves, it is best to solve from long tracks generated in the shot. If there are lots of short tracks, this can lead to noise on the calculated camera. Try removing the short tracks before solving.

- **Track Threshold** - CameraTracker's tolerance to change along the track is determined by this control.

  Reducing this threshold makes tracking more tolerant to image changes, potentially producing longer tracks.

- **Track Smoothness** - sets the threshold for smooth track generation. Adjusting this value can be useful in preventing poor tracks in complex sequences.

  Increasing the smoothness value removes tracks that fail over time.
• **Track Consistency** - sets the threshold for how inconsistent a feature track can be before CameraTracker discards it and reseeds in a different location.

  Higher values allow for less inconsistency and vice versa.

  Click **Delete Rejected** to remove all the tracks that fall below the specified thresholds.
Extending Existing Camera Tracks

CameraTracker allows you to extend an existing set of tracking data by adding new frames, such as when more frames become available from a shoot.

**Note:** You can update tracking data as often as you like before solving, but once you've solved the camera position, updating tracking data should only be used to add a relatively small number of frames. See Updating Solves with Extended Tracking Data for more information.

To add new frames to your tracking data:

1. Read the new frames into Nuke using the Read node's `frame range` controls.
2. Open the CameraTracker properties panel and click **Update Track**. A dialog displays allowing you to set the frame range to update.
3. Set the required frame range and click **OK**.
   
   CameraTracker tracks the specified range and combines the tracking data with the existing tracks. For example, if your original `AutoTracks > num tracks` curve appeared as shown on the left, the image on the right represents the new tracking data after the update is complete.

   ![Original frame range track data. Updated track including the extended frame range.](image)

   Tip: If the transition between the ranges seems abrupt, you can retrack the frames around the join using the method described in Retracking Partial Frame Ranges.

4. Proceed to **Solving the Camera Position**.
Retracking Partial Frame Ranges

After refining your feature points, you may not need to retrack the entire sequence. You can use Update Track to analyze a specific frame range.

1. Refine your features using the controls described previously.
2. Click Update Track.
   A dialog displays allowing you to set the frame range to update.
3. Set the required frame range and click OK.
   CameraTracker retracks the selected range and combines the tracking data with any existing tracks.

Note: You may notice that CameraTracker performs some analysis outside your selected frame range. This can be necessary to extend existing tracks into the update range, avoid creating duplicate tracks, and to combine the new tracks with existing tracking data.
Tracking in Stills Mode

In Stills mode, CameraTracker tracks the reference frames in the Source input and analyzes the input in two stages. Tracking defines the set of 2D feature tracks that correspond to fixed rigid points in the scene, then the solver calculates the camera path and projection that creates a 3D point for each reference frame.

**Note:** If you intend to remove lens distortion manually using a separate LensDistortion node, you should do that before you track the stills. See Working with Lens Distortion. Otherwise, set Lens Distortion to Unknown Lens before you solve the camera position to force CameraTracker to calculate the distortion.
Still Photography Guidelines

Still tracking relies on good input to produce good tracking output, so it’s vital that you capture good still photographs that CameraTracker can interpret correctly.

CameraTracker has different requirements depending on the subject of the stills. For example, stills for a near-flat scenes are captured differently to those required for an interior set or model.

General Guidelines

• Don’t crop your stills or transform them in anyway, such as rotation.
• Avoid dramatic changes in scale and angle between stills.
• To ensure you capture every part of the scene in 3 to 4 images, aim for a maximum change of 20-25% in content between frames.
• Avoid unnecessary redundancy where still frames contain large portions of the previous frame.
• Avoid tracking stills containing multiple occlusions - features with complex overlap can result in drastic changes in content between frames.

Shooting Near-flat Scenes

The best stills for near-flat scenes are captured head-on, facing the subject. In Nuke terms, that would mean taking your stills from the camera positions shown on the right.

![Incorrect camera positioning.](image1) ![Correct camera positioning.](image2)

Shooting 3D Objects

When moving around an object, a photo every 15-25 degrees should be adequate, so a minimum of 16 shots for a full 360. Of course, taking more can improve the result.
Shooting Interiors

For enclosed spaces, such as interiors, capture stills from the center facing outward and then around the perimeter facing inward, rather than from different points within the space.

Incorrect camera positioning from different points.

Capturing stills facing outward. Capturing stills facing inward.
Tracking Still Frames

Before you do anything else, you need to tell CameraTracker which still frames you want it to track and how it should distribute the tracking features on each frame. Then, you can track your still frames.

Selecting the Frames to Track

1. On the CameraTracker tab of the CameraTracker properties, ensure that Source is set to Stills.
2. Use the Range dropdown to determine which frames are analyzed:
   - **Input** - the default value, sets the frame range to the length of the sequence attached to the Source input.
   - **Global** - sets the tracking frame range to the range set in the Project Settings frame range controls.
     If no frame range is defined, the frame range of the first image you read in is used as the Global frame range.
   - **Custom** - sets a tracking range of frames described by the from and to fields.
   - **Reference Frames** - allows you to manually set keyframes on the frames you want to analyze.
3. If you set Range to Reference Frames, do one of the following:
   - To set all frames in the input as reference frames, set the Add dropdown menu to Add All. Any missing frames on the input are skipped.
   - To set a specific frame range as reference frames, set the Add dropdown menu to Add Range. In the dialog that opens, enter the frame range in the Frame range field and click OK.
   - To set individual frames as reference frames, set the Thumbnails dropdown menu to All at the top of the Viewer. Then, scrub to a frame you want to set as a reference frame and click the add reference frame button . The current frame is set as a keyframe and displayed in the thumbnail gallery at the bottom of the Viewer.
Scrub to the next frame you want to add as a reference frame and repeat the process until you're happy with your set of reference frames.

**Tip:** If necessary, you can also use the delete reference frame button or the Delete dropdown menu in the CameraTracker properties to remove frames from the set of frames to analyze.

4. Proceed to *Adjusting the Features to Track* below.

**Adjusting the Features to Track**

1. If you intend to mask out parts of your image (such as burn-ins), set the Mask control to the matte source. For example, if you're using the alpha channel from a Roto node attached to the Mask input, select Mask Alpha. For more information, see Masking Out Regions of the Image.

**Note:** For stereoscopic or multi-view footage, set the Principal View on the CameraTracker or Settings tab to the main tracking view. Any masking should be applied to this view, which is then used to calculate the secondary camera.

2. On the Settings tab, define the starting points for feature tracking:
   - **Number of Features** - define the number of features you want to track in each frame.
Ideally, you should use more than 200 tracks per frame. In most cases, the default 250 should be sufficient, but in difficult solves you may consider using twice that number.

• **Detection Threshold** - set the distribution of features over the input image.
  If you enter a low detection threshold value, features are tracked evenly on all parts of the image and vice versa.

  ![Low threshold distribution](image1) ![High threshold distribution](image2)

• **Feature Separation** - set the distribution of features in relation to each other.
  To force feature separation and spread features evenly over the image at even distances, enter a high feature separation value.

  ![Low separation value](image3) ![High separation value](image4)

3. Check **Refine Feature Locations** to lock detected features to local corners. If you activate this, CameraTracker finds the closest corner point in your footage and locks the feature to it.

4. Check **Preview Features** to view the current distribution of tracking features.
  Preview comes in handy when you want to tweak the tracking parameters further before tracking - it updates dynamically when controls are adjusted.
  It's important to make sure that the previewed features are distributed evenly and not clustered together too densely before tracking using the distribution and separation controls.
5. Once you’re happy with the feature distribution in the preview, proceed to Tracking the Selected Frames below.

**Tracking the Selected Frames**

1. In the CameraTracker properties, click Track to begin analyzing the still frames.
   CameraTracker begins by detecting viable features, aligning the reference frames, and then tracking forwards. Tracks that don’t meet the quality thresholds set in the Settings tab Tracking controls are reseeded so that the number of tracks remains constant.
   When the playhead reaches the end of the sequence, it begins a verifying pass by reading the frames sequentially backwards. Any tracks that were reseeded due to error are tracked back past the point where they were created, if they remain viable.

2. Proceed to Viewing Reference Frames and Track Data.
Viewing Reference Frames and Track Data

Once tracking is complete, you can review how CameraTracker connected the analyzed frames:

1. Do one of the following:
   - Switch the **Thumbnails** control above the Viewer to **Tracked**. CameraTracker displays the frames that are connected to the current frame in the thumbnail gallery at the bottom of the Viewer. This allows you to simply step through the sequence frame-by-frame to evaluate the connections.
   - Switch the **Thumbnails** control above the Viewer to **All** to have CameraTracker display all frames in the tracked range in the thumbnail gallery. This allows you to hover the cursor over a thumbnail to view its connections to adjacent reference frames highlighted in orange. Disconnected frames are highlighted in red. See [Disconnected Frame Sets](#) for more information.

   ![Image of tracking connections](image)

   **Tip:** You can click-and-drag to scroll the gallery left and right, or click a thumbnail to move the playhead to the corresponding reference frame. To adjust the size of the thumbnail gallery, drag the top of the gallery up and down.

2. To display track information, hover over a feature or marquee select several features.

   **Note:** If you make multiple selections, you won't see the track lengths in frames, only the tracks themselves.
Reviewing AutoTracks Curves

Detailed tracking information is displayed in curves in the properties panel on the AutoTracks tab. You can select all track curves at once or get a more detailed view of a single curve, such as num tracks.

Tip: Pressing F with focus on a curve maximizes the selected curve in the space available.

Curves can indicate areas of the sequence where tracking encountered problems. For example, the number of tracks curve on the right shows a significant dip on the current frame, as indicated by the playhead position.

In this instance, you could try adjusting the **Number of Features** or **Minimum Length** and retracking. See Troubleshooting Still Tracks
Disconnected Frame Sets

Auto-tracking stills is not perfect and may throw out some disconnected frames, most likely due to a large change in viewpoint. After tracking, set the **Thumbnails** control above the Viewer to **All** and hover over images in the thumbnail strip. Thumbnails highlighted in red were not matched with the reference frames around them and are flagged with an error message in the Viewer.

CameraTracker can deal with disconnected frames in two ways:

- work with the connected set by deleting the unconnected reference frames using the delete button or **Delete** dropdown and tracking again, or
- if all the frames in the sequence are needed, create User Tracks that span the sets to connect them.

Make sure to define the tracks in as many frames as possible, and then track again based on the User Tracks. See **Linking Still Reference Frames**.

**Tip:** When a camera track starts and ends in the same place, CameraTracker may not automatically connect the end frame with the starting frame. You can set the **Thumbnail** control to **Tracked** to review the connections created between stills, and then add User Tracks between the two frames and click **Update Track**. See **Working with User Tracks** for more information.
Troubleshooting Still Tracks

Some stills sequences are inevitably going to cause problems. There are a number of pre-tracking checks and post-tracking refinement controls to assist CameraTracker.

You can improve tracking data by adding User Tracks. See Working with User Tracks for more information.

Pre-tracking Checks

• For standard sequences, avoid too much or too little redundancy between still frames, that is, frames that contain large portions of the previous frame. As a guide, try to capture every part of the scene in 3 to 4 images, aiming for a maximum change of 20-25% in content between frames.
• Try to capture the same object in at least 3, but preferably 4 photos.

For example, when moving around an object, a photo every 15-25 degrees would be adequate (approximately 16 shots for a full 360).

Post-Tracking Refinements

You can refine a set of feature tracks using the following controls and then retrack to improve your chances of getting a good solve:

• **Minimum Length** - sets a threshold value for the minimum acceptable track length. Tracks that fail to stick to the associated feature for this amount of frames are rejected.

In long slow camera moves, it is best to solve from long tracks generated in the shot. If there are lots of short tracks, this can lead to noise on the calculated camera. Try removing the short tracks before solving.

• **Track Threshold** - CameraTracker’s tolerance to change along the track is determined by this control.

Reducing this threshold makes tracking more tolerant to image changes, potentially producing longer tracks.

**Tip:** Press F in the curve display to fit the selected track(s) to the available screen space.
Retracking Partial Frame Ranges

After refining your feature points, you may not need to retrack the entire sequence. You can use **Update Track** to analyze a specific frame range.

1. Refine your features using the controls described above.
2. Click **Update Track**.
   A dialog displays allowing you to set the frame range to update.
3. Set the required frame range and click **OK**.
   CameraTracker retracks the selected range and combines the tracking data with any existing tracks.

**Note:** You may notice that CameraTracker re-detects features on existing frames before re-tracking. This can happen when the cached feature data has been purged and needs to be rebuilt.
Working with User Tracks

User Tracks are placed manually, rather than being automatically seeded by CameraTracker, and can be used to improve or even replace auto tracking data. They can also be used to link unmatched reference frames together when tracking stills.

You can create User Tracks before tracking and solving to lock the camera to a particular part of the shot or afterwards to help improve the results.

These pages cover the following topics:
• Adding and positioning User Tracks in the Viewer. See Adding and Positioning User Tracks.
• Track User Tracks in several different ways. See User Tracking Methods.
• Using User Tracks to improve auto-tracking data. See Tracking Assists.
• Tracking a scene manually using only User Tracks. See Tracking a Scene Manually.
• Using User Tracks to link reference frames when tracking stills. See Linking Still Reference Frames.
• Assigning User Tracks as 3D survey points in order to tie your sequence to a known 3D world, such as those created using stills. See Assigning 3D Survey Points.
Adding and Positioning User Tracks

You can add as many User Tracks as required, depending on what you intend to accomplish. For example, auto-track assist may only require one or two User Tracks, whereas manual tracking would require at least eight User Tracks.

1. Enable the fast-add button at the top of the Viewer and click in the Viewer to add User Tracks or click Add Track on the UserTracks tab in the properties panel.

**Note:** Clicking Add Track places the new track in the center of the current Viewer.

**Tip:** You can quickly add User Tracks by holding Ctrl/Cmd+Alt and clicking in the Viewer.

Track zoom windows display in the top left of the Viewer.

There are three zoom windows initially:

- **Reference frame** (green) - the first frame in the track, which remains constant, allowing you to locate the feature more accurately in subsequent frames.
- **Current frame** (orange) - the playhead position. Use this window to adjust the tracking anchor on the current frame by comparing it to the reference frame.
• **Keyframe** (blue) - the first keyframe in the sequence. Adding more keyframes adds a zoom window per keyframe.

**Tip:** You can click on a keyframe zoom window to instantly jump to that frame.

**Tip:** You can hide the zoom windows by setting the **Zoom Window** Viewer control to **Off**.

2. Drag the crosshair in the center of the anchor over the feature to be tracked or manually adjust the track \( x \) and \( y \) fields in the UserTracks list.
3. Refine the track position by clicking and dragging in the current frame zoom window.

**Tip:** You can change the magnification of zoom windows by holding **Shift** and dragging the magnifying glass cursor away from the center of the window.

To resize a zoom window, drag its lower right corner to a new location.

4. Once you're happy with the track's position, proceed to **User Tracking Methods**.
User Tracking Methods

User Tracks can be tracked automatically or manually, extracted from auto-tracking data, and imported from a Tracker node.

Auto User Tracks

The easiest way to get started with User Tracks is using the Autotrack feature, but you can create tracks totally independent of auto-tracking. After placing your tracks on the required features in the sequence:

1. Click on the UserTracks tab in the properties panel to display the track table.
2. Select all the tracks you want to calculate in the table and then click Autotrack.
   CameraTracker attempts to track the selected features, in a similar way to Auto Tracks, except that when a track fails it is not reseeded.
3. You can examine features during and after tracking to determine if they failed:
   • During tracking - the anchors are colored yellow until they fail.
   • After tracking - any failed tracks don’t have keyframes all the way through the sequence.
   The image shows a good track, user 1, highlighted in yellow and a failed track, user 3, which has started to drift on frame 116.

4. If a track fails or drifts, adjust the position of the tracking anchor by dragging the anchor in the zoom window.
5. Scrub through the sequence a few frames and repeat. Continue on through to the end of the sequence.

6. At each frame, a new keyframe window is added to the right of the zoom window.

Manual User Tracks

Manual tracking can produce the best results, but is time consuming because all keyframes are tracked manually by you, rather than CameraTracker.

**Tip:** Generally, you would only manually create User Tracks in stills. Nuke's Tracker node can create reliable tracks for continuous sequences which you can then import as User Tracks.

After placing your tracks on the required features in the sequence:
1. Click on the **UserTracks** tab in the properties panel to display the track table.
2. Select the required track from the tracks list.
3. Scrub the playhead forward a few frames and move the tracking anchor to the correct position in the new frame using the current zoom window.
   A new keyframe zoom window is added to the Viewer.

4. At the top of the Viewer, set **Thumbnails** to **User Tracks**.
   All keyframes for the currently selected user track are displayed in the thumbnail gallery.

5. To see a User Track's reference frame, hover over the track in the Viewer. This can help you quickly decide if the object you're tracking is in the current frame or not.
6. When you've finally keyframed your User Track in all of the desired frames, the tick marks on the Viewer's timeline are set on all of those keyframes. By using Alt+left or right arrow, you can quickly jump to the previous or next keyframe. Doing this while looking at the zoom window helps show you what keyframes your User Track is slightly off in, making this a very useful trick for getting perfectly placed User Tracks.

Extracting User Tracks

After auto-tracking, you can use the data to extract User Tracks. CameraTracker considers User Tracks to be important and tries to lock down on them over auto-tracks. As a result, extracting User Tracks from an auto-track can be useful if you have an auto-track that appears to be reliable.

You might also extract auto-tracks to User Tracks if you want to export some auto-tracks to be used in a Tracker node for 2D work.

1. Right-click on an auto-track in the Viewer and select tracks > extract user track.
   The selected track is prefixed with extr. and placed in the User Tracks table.
2. If a track fails or drifts, adjust the position of the tracking anchor by dragging the anchor using the zoom window.
3. Scrub through the sequence a few frames and repeat. Continue on through to the end of the sequence.
4. At each frame, a new keyframe window is added to the right of the zoom window.

Importing a Tracker

You can quickly create User Tracks by importing keyframes and tracking data from a Tracker node within the same script.

1. Click on the UserTracks tab in the properties panel to display the track table.
2. Click Import Tracker.
3. The Select Tracker to Import From dialog displays.
4. Select the required Tracker node from the tracker node dropdown.
   A new User Track is added to the tracks list.
Tracking Assists

Auto-tracking data can be improved by seeding User Tracks in a sequence, as CameraTracker assumes manually placed tracks are superior to auto-tracking. Re-tracking after adding User Tracks forces CameraTracker to recalculate auto-tracks using more accurate data.

1. Track your sequence using the steps described in Tracking in Sequence Mode or Tracking in Stills Mode
2. Track one or two User Tracks using the methods described in User Tracking Methods.

**Tip:** Adding tracks is time consuming, but the more User Tracks you add, the better the result is likely to be.

3. Either re-track the entire sequence by clicking Track,
   OR
   If you only added a few keyframes to a particularly troublesome area of the sequence, click Update Track to re-track just that range of frames.
4. If the tracking data requires more work, try adjusting some of the controls described in Troubleshooting Sequence Tracks or Troubleshooting Still Tracks.
5. Proceed to Solving the Camera Position.
Tracking a Scene Manually

Sometimes, a scene can be difficult to track automatically. For example, you might be tracking a set of still photos of a scene with a very wide baseline between images. Alternatively, you might already have 2D track data from an earlier process that you want to get a solve from. In cases like this, it can be easier to only set up and work with User Tracks as your track data.

The key thing to remember is that you typically need at least 8 to 16 User Tracks per frame to get a reliable solve.

1. Use the steps described in Manual User Tracks to create the required number of manual tracks.
2. Proceed to Solving the Camera Position.
Linking Still Reference Frames

During stills tracking, the links between reference frames can fail. This can occur for a number of reasons, such as a lack of overlap between stills. You can help CameraTracker by adding User Tracks on features common to the frames before auto-tracking.

1. Use the steps described in Working with User Tracks to create at least four User Tracks in as many frames as possible.
   For example, in the stills shown, the tops of the pillars and the stone work on the corner of the building.

2. Move to the next frame and reposition the User Tracks to the corresponding features in the still. Continue throughout the sequence of stills.

3. Once all your frames have been linked in this way, click Track.
   CameraTracker uses the reliable information from the User Tracks to create a set of Auto Tracks.

4. Proceed to Solving the Camera Position.

**Note:** The above workflow describes how to use User Tracks to help connect frames that weren't automatically tracked. Sometimes, there are so many poorly tracked frames that instead of hoping that linking frames and clicking Track works, it might be easier to start from scratch and perform an entirely manual track. See Tracking a Scene Manually for more details.
Assigning 3D Survey Points

If you create a User Track, you can assign it as a known 3D survey point. This tells CameraTracker which points in your 2D footage go with their counterparts on your 3D model, allowing it to solve the camera to match the known 3D points and achieve the best results.

**Note:** 3D survey points have replaced the ProjectionSolver workflow, but you can still add ProjectionSolver nodes by pressing X in the Node Graph and entering `ProjectionSolver` as a Tcl command.

**Tip:** You can create a 3D model for your footage using ModelBuilder or an external application.

To create 3D survey points in your scene:

1. To define the full rotation, translation, and scale between the scene and your survey points, create and track at least three User Tracks. If you track more than three, you'll get the best fit result. See Working with User Tracks for more information.
2. Switch the Viewer to 3D mode, either by pressing Tab in the Viewer or using the View selection dropdown.
3. Set the Thumbnail control above the Viewer to All to display all reference frame thumbnails as an overlay in the Viewer.
4. Use the selection mode button above the Viewer to swap to Vertex selection mode.
5. Select a vertex on the 3D model and the User Track you want to be a survey point in the thumbnail gallery.
6. Right-click on the User Track and select user tracks > snap to 3D vertex selection.
7. In the CameraTracker properties panel, click the **UserTracks** tab to display a list of tracks. The user track table displays the pixel error when matching the 3D vertex to the User Track. You can use this to review how well the camera solve fits the 3D survey points. If the solver could not match a point, there will be a high error value. If this happens, try looking at the 2D feature positions to check they correspond to the same point and also double check the 3D vertex on the model.

8. Repeat the process for the required User Tracks in the scene, and then enable the **s** checkbox for each track in the User Tracks table. The **s** designates the points as known 3D survey positions during the solve, allowing CameraTracker to create a camera positioned correctly for the geometry.

**Tip:** If you have multiple footage sources of the same scene or content available, you can also use survey points to solve each of your sources and then register them all in the same world. See *Combining Solves*. 
Solving the Camera Position

When you're happy with the features that you've tracked, you can solve the camera position. CameraTracker uses the tracking information to calculate the camera position and add position information to the tracked feature points in the Viewer.

In the CameraTracker properties panel, click Solve. The solver selects a subset of keyframes positioned relatively far apart so that there is sufficient parallax in the images to define the 3D points for the tracks.

When the solve is complete, the Viewer displays the tracked features with solve data applied.
Viewing Solve Data

Once the solve data is placed in the Viewer, you can zoom in to display the points more clearly. You can control what appears in the Viewer using the Display controls on the Settings tab:

- **Show tracks** - show or hide the 2D tracking information.
- **Show projected 3D points** - show or hide the 2D position of the 3D points.
- **Show key tracks only** - only show the longest tracks used to calculate the solve.
- **Show 3D marker** - show or hide 3D markers at each point in the 3D Viewer.

A traffic light scheme applies to the 2D tracks to help find good thresholds for track rejection in the AutoTracks tab - amber by default, green for good tracks, and red for poor tracks.

The circles and crosses are reprojected solved auto-tracks. The closer the 3D point circles are to the feature points, the better the solve.

Hover over a point to display its solve information.

CameraTracker also creates a point cloud in the 3D Viewer allowing you to cross-check the relative positions of the 2D and solved 3D points. Select some points in 2D and then press Tab in the Viewer to switch between 2D and 3D space to check the point positions.
Several points selected in the 2D sequence. The same points shown in the 3D Viewer.

Reviewing AutoTracks Curves

Detailed solve information is displayed in curves in the properties panel on the AutoTracks tab. You can select all solve curves at once or get a more detailed view of a single curve, such as error - rms (root mean square).

**Tip:** Pressing F with focus on a curve maximizes the selected curve in the space available.

Curves can indicate areas of the sequence where solving encountered problems. For example, the root mean square error curve on the right shows a significant blip on the current frame, as indicated by the playback position. In this instance, you could try adjusting the Keyframe Spacing or Smoothness and updating the solve. See Troubleshooting Solves

The curves in the graph show the following solve information:
• **Solve Error** - displays the constant SolveError parameter.
• **error min** - the minimum reprojection error at each frame (in pixels).
• **error rms** - the root mean reprojection error at each frame (in pixels).
• **error track** - the maximum root mean reprojection error calculated over the track lifetime at each frame (in pixels).
• **error max** - the maximum reprojection error at each frame (in pixels).
• **Max Track Error** - displays the constant Max RMS Error parameter.
• **Max Error** - displays the Max Error threshold parameter.

### Reviewing Solved User Tracks

When a User Track is solved, you can review its error in the User Track table on the UserTracks tab. If the error is high, try reviewing the 2D x,y feature track. Alternatively, you can uncheck the User Track’s e (enable) column to remove it from the scene before clicking Solve again.

The User Track's 3D position (X,Y,Z) is also shown in the table, as well as in the 3D Viewer.

> **Tip:** You can create a User Track after solving the camera in order to extract a specific 3D position in the shot. To produce an accurate 3D point, the User Track must be defined in 3 or more frames with good parallax. When you’ve created the User Track, select it in the User Tracks table and click the Update XYZ button to triangulate the 3D position from the current solve.

> **Tip:** After auto-tracking, you might want to select an auto-track and extract it as a User Track. This hints to the solver that this is an important track and should be locked down on. This is particularly useful if a particular part of the scene is not solving properly, despite the auto-tracks appearing to have tracked well in that region.

For more information, see [Extracting User Tracks](#).

### Previewing Matchmove Quality

After calculating the solve, you may want to review its quality by previewing how well objects added to the 3D scene are going to stick to your footage. If you set Lens Distortion to Unknown Lens before creating the solve, there are two ways to do this:
**Method 1**

1. In the 2D Viewer, right-click on a track point (or several selected points, ideally on the same plane) and select **create > cube**, for example.
   CameraTracker creates a Cube node and places it in the average position of the selected points. You are going to use the cube to test how well it sticks to the input footage.

   **Tip:** If necessary, you can adjust the size of the cube using the **uniform scale** control in the Cube properties.

2. In the CameraTracker properties, set the **Export** menu to **Scene+** and click **Create**.
   CameraTracker creates a 3D Scene with a Camera, PointCloud, ScanlineRender, and a LensDistortion node set to undistort the input.

3. Connect your Cube node to one of the Scene node's inputs.

4. View the output from ScanlineRender.
   The cube you created in step 1 and the point cloud CameraTracker generated are displayed on top of the undistorted footage in the 2D Viewer.
5. Scrub through the timeline to see whether the cube and the point cloud stick to the footage. If they do, delete any nodes you no longer need (such as the Cube node) and proceed to Adjusting the Scene. If they don't, proceed to Troubleshooting Solves.

**Method 2**

1. In the CameraTracker properties, set the Export menu to Camera and click Create. CameraTracker creates a Camera node that emulates the camera used on set.

2. In the 2D Viewer, right-click on a track point (or several selected points, ideally on the same plane) and select create > cube, for example. CameraTracker creates a Cube node and places it in the average position of the selected points. You are going to use the cube to test how well it sticks to the input footage.
Tip: If necessary, you can adjust the size of the cube using the uniform scale control in the Cube properties.

3. Enable **Undistort Input**. CameraTracker removes lens distortion from the input footage.
4. Make sure that both the CameraTracker and Camera properties panels are open.
5. Press **Tab** on the Viewer to switch to the 3D view.
6. Set the camera look through menu in the top right corner of the Viewer to the camera you created in step 1 and click the **button to lock the 3D view to that camera.
7. At the top of the Viewer, set the Viewer composite dropdown menu to **over**, so that you are viewing your CameraTracker node over the same CameraTracker node. This overlays the input image currently undistorted by CameraTracker over the point cloud in the 3D Viewer.

8. Scrub through the timeline to see whether the cube you created in step 2 and the point cloud CameraTracker generated stick to the footage. If they do, delete any nodes you no longer need (such as the Cube node) and proceed to **Adjusting the Scene**. If they don’t, proceed to **Troubleshooting Solves**.
Troubleshooting Solves

CameraTracker has several troubleshooting workflows available to improve solve accuracy, but ultimately, good solves rely on good tracking data.

Using Curve Thresholds to Delete Tracks

You can use the threshold controls on the AutoTracks tab to dynamically reject tracks and remove them to improve accuracy. Using the error - rms curve as an example:

1. Click the AutoTracks tab in the properties and select the error - rms curve.

   ![](image)

   **Tip:** Press F in the curve display to fit the selected track(s) to the available screen space.

   In the example, you can see that the error curve has increased significantly around frame 195.

2. Move the Viewer playhead to the affected area and examine the tracked features in the Viewer.

3. If you decrease the Max Track Error control in the properties panel, you'll start to see tracks turn red as they fall below the threshold.
Max Track Error set to 6.  

Max Track Error set to 2.

4. Click **Delete Rejected** to remove all the tracks that fall below the specified threshold.

**Tip:** You can also remove tracks manually, by selecting them in the Viewer, right-clicking, and choosing **tracks > delete selected**.

5. You can then resolve the affected frame range by clicking **Update Solve**.

6. Improve the solve further using the **Settings** tab **Solving** controls:
   - **Camera motion** - set the camera movement that CameraTracker should account for during the solve calculation.
   - **Keyframe Separation** - adjusts the separation value between keyframes.
     High separation values generate fewer keyframes with a greater spread and are typically used for slower camera motion.
     Low values generate more keyframes with a tighter spread and are typically used for rapid camera motion.

**Note:** When enabled, the **Reference Frame** control determines where the first keyframe is placed.

- **Smoothness** - higher values can help smooth a camera path by adding 'weight' to iron out the path.
- **Reference Frame** - allows you to specify the start frame for updating camera solves, as well as determining where the first keyframe is placed.

**Note:** Check **Set reference frame** to enable this control.
Refining a Solve

On the AutoTracks tab, in the Refinement section, there are three refinement controls to help improve your solve. If the Error and per frame controls on the CameraTracker tab show a relatively high value, try refining the solve using the inlier tracks. The inliers are defined by the curve thresholds and can refine the focal length, camera position or camera rotation (or a combination of these). You can manually edit the camera solve first on the Output tab, then select:

• **Focal Length** - Check to refine the camera’s focal length.
• **Position** - Check to refine the camera position.
• **Rotation** - Check to refine the camera rotation.

Finally, click **Refine Solve**.
Updating Solves with Extended Tracking Data

CameraTracker allows you to add more frames to an existing camera solve using updated tracking data. It is usually not possible to extend the range very far (about 10-15% of the original range), because the existing solve locks the 3D points in place, so matching new 2D tracking data to 3D points can produce high tracking errors, which CameraTracker rejects.

For example, if you've calculated a solve from frames 1-10, the 3D point for a given track is based on the data from only those frames. Updating that track by extending it to frame 20 attempts to fit the 2D point from frames 11-20 to the calculated 3D point. This generally doesn't produce a good fit, and the solve error for the extended range is increased.

To update a solve with updated tracking data:

1. Track the additional frames as described in Extending Existing Camera Tracks.
   - If there are a lot of rejected tracks, highlighted in red, try using fewer frames in the tracking update.
2. Click Update Solve to calculate the new tracking data.
   - If there are a lot of rejected tracks, you may find it easier to retrack the entire range and solve the camera position from the new, complete data. Re-solving from scratch uses all of the parallax information over all frames (rather than the initially solved frames), and so you often get a lower overall solve error.
Adjusting the Scene

When the tracking and solving processes are complete, you can use the solve data to align and scale the scene before adding cameras, point clouds, objects, and so on.

You can:

• Set the ground plane, axes, or origin for the scene. See Setting the Ground Plane and Axes.

• Manually adjust the scene using the Scene tab's Scene Transform controls. See Transforming the Scene Manually.

• Use known distances, measured on set, to scale a scene more accurately. See Using Scene Constraints.
Setting the Ground Plane and Axes

A solved camera has no notion of where the ground plane is in the scene, which can produce an unexpected offset and angle relative to where you'd expect the ground to be in 3D space.

Setting the ground plane or set of axes is designed to provide CameraTracker with a sensible frame of reference. While doing so is not strictly necessary, it can simplify working in the 3D environment.

Before placing the ground plane, scan through your sequence to find a frame with good track and solve data in an area that you know to be at ground level.

1. Hover over likely points to display the track and solve data, typically points with RMS error less than 1.
2. Hold Shift and select your chosen points, from multiple frames if necessary.
3. Switch between the 2D and 3D Viewers to confirm the position of the selected points.

4. Switch back to the 2D Viewer and right-click a selected point to display the track options menu.
5. Select ground plane > set to selected to orient the camera in relation to the selected points.

Points designated as the ground plane are highlighted in magenta and the camera is reoriented in relation to the new ‘ground’. 

Ground plane points selected in the 2D Viewer. The same points viewed in the 3D Viewer.
Setting Axes Individually

In some situations, you may not have a clearly defined ground plane to reference. Using CameraTracker, you can set the X, Y, and Z axes separately instead of grouping them on the ground plane.

1. Select two or more points that you know to be on the required axis using the 3D point cloud to assist you.
2. Right-click a point and select `ground plane > set X, Y, or Z` as required.

   Points designated as an axis are highlighted in yellow and the camera is reoriented in relation to the new axis.

Setting the Origin Within a Scene

Defining the origin within a layer can assist you when placing objects in your composition - if you know where 0, 0, 0 is located, you can easily place objects precisely where you want them on any axis.

1. Select a single track as an origin point from your solve data.
2. Right-click the point and select `ground plane > set origin`.

   The point designated as the origin is highlighted in yellow and the point cloud is translated in relation to the new origin.
Point before the origin shift.  
Point after the origin shift.
Transforming the Scene Manually

In addition to defining the ground plane, axes, and origin automatically, you can manually adjust the scene using the Scene tab's Scene Transform controls. These controls operate in exactly the same way as other transform tools in Nuke.

To transform a scene, use the following controls:

- **rotation order** - set the operation order for rotations from the dropdown menu which displays all the possible axial combinations.
- **translate** - set translate values for the scene. You can adjust translate values by clicking and dragging the axis in the 3D viewer.
- **rotate** - set values to rotate your camera or use Ctrl/Cmd to rotate the camera in the Viewer.
- **scale** - set scale values on individual axes, rather than uniformly or use Ctrl/Cmd+Shift to scale the camera in the Viewer.

**Note:** This control is only displayed if you've scaled the scene on individual axes.

- **uniform scale** - set a scale for your scene. You can also scale the scene using actual measurements taken on set, see Using Scene Constraints for more information.

Alternatively, you can also use the Local matrix to perform transforms on your scene.

**Note:** The World matrix is read-only and cannot be edited.

Transforming Using Channel Files

Channel files contain a set of cartesian coordinates for every frame of animation in a given shot. You can create and export them using Nuke or 3D tracking software, such as 3D-Equalizer, Maya, or Boujou.

You can transform the scene using channel files by clicking the button:

- **Import chan file** - import a channel file and transform the input object according to the transformation data in the channel file.
- **Export chan file** - export the translation parameters that you've applied to the input object as a channel file. This is a useful method of sharing setups between artists.
Transforming Using Snap To Position

The Snap To menu allows you to match the position, orientation, and scale of existing points.

Select the required points and then click the button:

- **Match selection position** - the scene is snapped to a new position depending on the points selected.
- **Match selection position, orientation** - the scene is snapped to a new position and orientation depending on the points selected.
- **Match selection position, orientation, size** - the scene is snapped to a new position, orientation, and size depending on the points selected.
Using Scene Constraints

Scene constraints enable you to use known distances, measured on set, to scale a scene more accurately. They are applicable to both Auto Tracks and User Tracks.

The unit of measure used is up to you, because Nuke equates any value as being equal to one unit in 3D space. The image shows a point cloud and camera with two User Tracks marked out. The blue square represents one unit of measure, so the smaller squares represent one tenth of that unit. So, if you measured on set in meters, one small square could be equal to a meter, centimeter, or millimeter.

To create scene constraints:

1. Select two solved points in the Viewer.
2. Right-click a highlighted point and select **scene > add scale distance**.

The image shows User Tracks, but the same workflow applies to Auto Tracks. For example, the Auto Tracks located on the window panes could easily be measured.
A connecting line, labeled dist. 1, is drawn in the Viewer.

3. In the properties panel, click the Scene tab to display the Scale Constraints table.

4. Click the distance field for dist. 1 and enter the measured distance.
   The measured distance is now taken into account when you adjust the Scene Transform controls.
Using Solve Data

Once your camera solve is correctly aligned, you can create animated or baked cameras, multi-view rigs, 3D scenes and point cloud, lens distortion, or cards from the camera data. See:

- Creating Camera Nodes.
- Creating Scenes.
- Creating Point Clouds.
- Creating Cards.
Creating Camera Nodes

Using the tracking and solve data, CameraTracker can create linked or baked cameras that emulate the original camera track on set. You can create a single camera or a camera rig for multi-view projects, depending on your script.

1. Select Camera from the Export dropdown menu.
2. Enable or disable the Link output control to determine whether the Camera node is expression linked or baked:
   • When enabled, CameraTracker creates an expression linked camera so that any adjustments made in the properties panel Output > Camera controls update the camera.
   • When disabled, any adjustments made are ignored by the camera.
3. Click Create.
A Camera node is added to the Node Graph. Expression linked cameras are linked to the CameraTracker node with a green expression arrow.

Camera node data can be used in various Nuke workflows, such as Creating Dense Point Clouds and Generating Depth Maps.

Creating a Camera Rig

CameraTracker creates as many cameras as you have views in your script, but the most common use of rigs is for stereoscopic scripts with left and right views.

1. Select Camera rig from the Export dropdown menu.
2. Enable or disable the Link output control to determine whether the Camera nodes are expression linked or baked:
   • When enabled, CameraTracker creates expression linked cameras so that any adjustments made in the properties panel Output > Camera controls update the cameras.
   • When disabled, any adjustments made are ignored by the cameras.
3. Click **Create**.

**Note:** If you try to create a rig with only one view, an error is displayed.

A Camera node is added to the Node Graph for each view present in the script. Expression linked cameras are linked to the CameraTracker node with a green expression arrow.

![Camera node example](image)

Camera node data can be used in various Nuke workflows, such as **Creating Dense Point Clouds** and **Generating Depth Maps**.

### Creating a Camera Set

CameraTracker can automatically create a camera for each solved frame.

1. Select **Camera set** from the **Export** dropdown menu.
2. Enable or disable the **Link output** control to determine whether the cameras are expression linked or baked:
   - When enabled, CameraTracker creates expression linked cameras so that any adjustments made in the properties panel **Output > Camera** controls update the cameras.
   - When disabled, any adjustments made are ignored by the cameras.
3. Click **Create**.
   - If you're processing a large amount of frames, a confirmation dialog displays.
4. Click **Yes** to continue or **No** to cancel the export.
   - CameraTracker adds a Group node to the Node Graph containing Camera nodes for every frame specified connected to a Scene node.
5. Double click the Group to open the properties panel, then click the **S** above the panel to open the group in the Node Graph.
You could then write the camera data to an `.fbx` file ready for import into Modo or Maya for projection painting work.
Creating Scenes

CameraTracker can create a ready-to-use 3D scene containing a point cloud, camera, and Scene node from the track and solve data. The Scene+ option adds LensDistortion and ScanlineRender nodes in addition to the standard scene nodes.

1. Select Scene from the Export dropdown menu.
2. Enable or disable the Link output control to determine whether the scene's Camera node is expression linked or baked:
   - When enabled, CameraTracker creates an expression-linked camera so that any adjustments made in the properties panel's Output > Camera controls update the camera.
   - When disabled, any adjustments made are ignored by the camera.
3. Click Create.

CameraTracker adds Camera, CameraTrackerPointCloud, and Scene nodes to the Node Graph. Expression linked cameras are linked to the CameraTracker node with a green expression arrow.

Creating a 3D Scene with ScanlineRender and LensDistortion

1. Select Scene+ from the Export dropdown menu.
2. Enable or disable the Link output control to determine whether the scene's Camera node is expression linked or baked:
   - When enabled, CameraTracker creates an expression-linked camera so that any adjustments made in the properties panel's Output > Camera controls update the camera.
   - When disabled, any adjustments made are ignored by the camera.
3. Click Create.
CameraTracker adds Camera, CameraTrackerPointCloud, Scene, ScanlineRender, and LensDistortion nodes to the Node Graph. The LensDistortion node is set to undistort the 2D footage in CameraTracker's **Source** input and use the result as the background for the 3D scene. Expression-linked cameras are linked to the CameraTracker node with a green expression arrow.
Creating Point Clouds

CameraTracker can create a baked point cloud, that is, points that don’t update when you make changes to the CameraTracker properties panel controls.

1. Select Point cloud from the Export dropdown menu.
2. Click Create.
   CameraTracker adds a CameraTrackerPointCloud node to the Node Graph.
3. Double-click the CameraTrackerPointCloud node to open the properties panel.
4. Set how the points should appear, both in the Viewer and when rendered out, using the display and render dropdowns:
   • off - hides the 3D points.
   • wireframe - displays only the points.
   • solid - displays all points using the color of the pixel they represent.
   • solid+wireframe - displays all points using the color of the pixel they represent and the points themselves.
   • textured - displays only the surface texture.
   • textured+wireframe - displays the surface texture and the points.
5. Enable or disable shadow casting and receiving when the point cloud is rendered to 2D using the checkboxes.
   See Casting Shadows for more information on lighting and casting shadows.
6. Set the size of the rendered points using the Point Size control.
Creating Cards

CameraTracker can automatically create a 3D card for each solved frame using the camera to project the image at that frame onto the card.

1. Select Cards from the Export dropdown menu.

2. Enable or disable the Link output control to determine whether the cameras are expression linked or baked:
   - When enabled, CameraTracker creates expression linked cameras so that any adjustments made in the properties panel Output > Camera controls update the cameras.
   - When disabled, any adjustments made are ignored by the cameras.

3. Click Create.
   If you're processing a large amount of frames, a confirmation dialog displays.

4. Click Yes to continue or No to cancel the export.
   CameraTracker adds a Group node to the Node Graph containing Card, FrameHold, and Camera nodes for every frame specified connected to a Scene node.

5. Double click the Group to open the properties panel, then click the S above the panel to open the group in the Node Graph.

6. Use the Group node's z slider to control the card distance from the camera. You can use this setting to create a pan and tile dome at this distance from the camera.
   The Cards are scaled automatically using the camera settings.
The z depth set to 1, the default.  
The z depth set to 3.
Combining Solves

Sometimes, you might have multiple footage sources of the same scene or content available. For example, you might have footage from witness cameras, or someone has taken detailed still photos of the scene. CameraTracker provides a way to solve each of your sources and then register them all in the same world. This allows you to leverage a high-quality camera track from a secondary source that is good to solve from, and use it in another source that's difficult to solve. You can also use this technique to tie detailed close-up stills to wide-angle shots.

The key idea is that you pick one of your sources as the 'master' solve. This should be footage that has good parallax, for example a set of wide-angle stills taken on set. You also need some geometry created from the solve data. The geometry is to establish 3D survey points, which are then used to tie together the 'master' and the satellite camera solves.

The workflow is something like this:
1. Pick one of your sources as the 'master' solve and use CameraTracker to track and solve the camera.
2. Create a linked camera using CameraTracker's Export menu. See Creating Camera Nodes.
3. Use the camera data to create some reference geometry. Either:
   • connect the Camera straight into the ModelBuilder node to create the geometry, or
   • export the camera data, create the geometry in an external application, then import the geometry back into Nuke.
   See Using ModelBuilder or Exporting Geometry, Cameras, Lights, Axes, or Point Clouds.
4. Set some 3D survey points on the geometry in the 'master' solve. See Assigning 3D Survey Points.
5. Create a separate CameraTracker for each of your satellite cameras, and track the footage as you see fit.
6. Create User Tracks in the satellite cameras, corresponding to the 3D Survey points in the 'master' solve, making sure to also mark them as survey points. See Assigning 3D Survey Points.
7. Solve the satellite cameras to align them consistently with the 'master' solve.

Tip: You can copy the X, Y, Z position from a track in one CameraTracker node to another by right-clicking on a track and selecting copy > translate.
Placing Objects in the Scene

You can use the camera and point cloud to add geometry to the scene. You can add objects manually, but placing them inside the camera’s field of vision at the desired position can be time consuming. CameraTracker provides an automatic creation function to help you achieve the results you need.

1. In the 2D view, select points on the plane to hold the geometry. For example, you might place a card on a vertical or horizontal plane.

   Tip: You might find that swapping between the 2D plate and 3D point cloud helps to locate potential points.

2. Drag a marquee over the required points or hold down the Shift key and click individual points.
3. Right-click a selected point in the 2D Viewer and choose the create menu to display the available geometry.
4. Select the required shape to place it in the scene using the average position of all selected points. The following example shows two cards placed in the scene using points from the vertical and horizontal planes.
Accounting for Lens Distortion

By default, any CG elements you add to your 3D scene do not have lens distortion applied to them. As a result, when you combine them with your 2D footage, they can look like they weren't shot with the same camera. To fix this, you can:

• Use CameraTracker to calculate the lens distortion on your 2D footage and generate a LensDistortion node that applies the same distortion to your CG elements. See Distorting Your CG Elements to Match Your 2D Footage.

• Use CameraTracker to calculate the lens distortion on your 2D footage and generate a LensDistortion node that undistorts the 2D footage to match your CG elements. See Undistorting Your 2D Footage to Match Your CG Elements (Using LensDistortion).

• Use CameraTracker to both calculate the lens distortion on your 2D footage and undistort the footage to match your CG elements. See Undistorting Your 2D Footage to Match Your CG Elements (Using CameraTracker).

Note: The above assumes you set LensDistortion to Unknown Lens before creating the solve. If you set Lens Distortion to No Lens Distortion and used a separate LensDistortion node to remove lens distortion from your footage before camera tracking, you can use the same LensDistortion node to either apply the distortion to your CG elements or to undistort your 2D footage. See Working with Lens Distortion for more information on using the LensDistortion node in Nuke.

Distorting Your CG Elements to Match Your 2D Footage

1. In the CameraTracker properties, set the Export menu to Distortion and click Create.
   CameraTracker creates a LensDistortion node that is set to apply the lens distortion that exists in your 2D footage.
2. Connect the LensDistortion node to the output of your ScanlineRender node.
   The LensDistortion node distorts the CG elements in your 3D scene to match your 2D footage.
3. Composite your CG elements and 2D footage together as necessary.

Undistorting Your 2D Footage to Match Your CG Elements (Using LensDistortion)

1. In the CameraTracker properties, set the Export menu to Undistortion and click Create.
CameraTracker creates a LensDistortion node that is set to remove the lens distortion that exists in your 2D footage.

2. Connect the LensDistortion node to the output of your 2D footage.
   The LensDistortion node removes the lens distortion in your 2D footage to match your CG elements.

3. Connect the LensDistortion node to the bg input of ScanlineRender.
   The undistorted 2D footage is used as a background image for the 3D scene.

Undistorting Your 2D Footage to Match Your CG Elements (Using CameraTracker)

1. In the CameraTracker properties, enable Undistort Input.
   CameraTracker removes the lens distortion that exists in its input footage.

2. Connect the CameraTracker node to the bg input of ScanlineRender.
   The undistorted 2D footage is used as a background image for the 3D scene.
About the Lens Model

Normally, you do not need to set the **Lens** parameters on the **Output** tab, as CameraTracker does this for you, but it may be helpful to know a little about the equations CameraTracker uses to account for lens distortion. There are two modes, depending on the lens type detected:

- **Spherical** - compensates for different spherical lenses. This is the simpler of the two lens corrections and uses the following equation:

\[ X_u = \frac{X_d}{1 + d_1 r^2 + d_2 r^4} \]

In normalized coordinates from the distortion center, \( X_u \) and \( X_d \) are equal to the same point in an undistorted plate and a distorted plate respectively.

- **Anamorphic** - compensates for anamorphic lenses. Anamorphic correction uses three additional parameters (\( A_x, A_y, \) and \( A_s \)) and requires two equations because, unlike with spherical lenses, the amount of distortion parallel to the x-axis is not the same as that parallel to the y-axis.

\[ x_u = \frac{x_d}{1 + (d_1) r^2 + (d_2) r^4 + (A_x) y_d^2} \]
\[ y_u = \frac{y_d}{1 + d_1 r^2 + d_2 r^4 + A_x x_d^2} \]

In normalized coordinates from the distortion center, \( (x_u, y_u) \) and \( (x_d, y_d) \) are equal to the same point in an undistorted plate and a distorted plate respectively.

- \( d_1 \) is equal to Radial Distortion 1.
- \( d_2 \) is equal to Radial Distortion 2.
- \( r_2 \) is equal to the distance of the point from the Distortion Center.
- \( r_4 \) is equal to the square of \( r_2 \).
- **Ax** is equal to Asymmetric Distortion X.
Ay is equal to Asymmetric Distortion Y.

Asq is equal to Anamorphic Squeeze.
Using MatchGrade

This chapter teaches you how to use MatchGrade to automatically calculate a grade to match the colors in the **Source** input to the colors in the **Target** input. You can use MatchGrade to:

- extract a baked-in grade if the **Target** clip that you want to match is simply a graded version of the **Source** clip. See Extracting a Baked-In Grade.

- match the grade between two different clips to create the same look. See Matching a Grade Between Two Different Clips.

In both cases, you can also mask the grade to only match certain elements between the **Source** and **Target** clips, and export LUT or CDL files to re-use the calculated grade elsewhere.

Special thanks to The Mill for use of the above footage, used throughout this chapter.
Extracting a Baked-In Grade

1. Select **Color > MatchGrade** to insert a MatchGrade node into your script.
2. Do the following:
   - Connect MatchGrade’s **Source** input to the clip to which you want to apply the color transform.
   - Connect the **Target** input to the clip you want to match. This should be a graded version of the **Source**.
   - Connect a Viewer to the output of MatchGrade.

3. In the MatchGrade properties, make sure **Task** is set to **Match Graded Source**.
4. To extract a baked-in grade, MatchGrade requires pixel-to-pixel correspondence between the **Source** and **Target** clips. If the clips are not aligned in time and space, such as when the target clip has been reformatted, click **Align Target to Source** to add a Transform node and a Reformat node upstream of the MatchGrade node.
Tip: If the target contains a region that isn't present in the source, for example a black border, you can enable Crop Target. Clicking Align Target to Source when Crop Target is enabled generates a rectangular crop for the Target input.

5. If you want MatchGrade to calculate a single, global grade from selected reference frames that cover the characteristic colors in the sequence and apply that grade to every frame of the Source sequence, proceed to Analyzing Reference Frames on page 1626. In this mode, you can export the grade to an OCIOFileTransform or OCIOCDLTransform node to re-use it elsewhere in the script. If you'd rather have MatchGrade calculate a local grade frame-by-frame, so that the color transform updates according to the current frame, proceed to Auto-Analyzing Per Frame on page 1629. In this mode, you cannot export the grade to re-use it elsewhere.

Analyzing Reference Frames

1. In the MatchGrade properties, make sure Analysis is set to Analyze Reference Frames.
2. If you only want to color match certain areas of the image, make sure there is a mask in the Source input's alpha channel. Then, set Mask to either:
   - Alpha - to use the alpha channel as a mask. The grade is limited to the non-black areas of the alpha channel.
   - Inverted Alpha - to invert the alpha channel and use that as a mask. The grade is limited to the non-white areas of the alpha channel.

Note: When the Mask control is set to anything other than None, an additional Apply Grade to Masked Region control is displayed.

This control is enabled by default, but you can disable it to apply the grade to the whole image instead, allowing you to compute the grade from a selected region and apply it to the whole image without having to export the LUT.

3. Set Output to Target.
   MatchGrade displays the Target footage in the Viewer.
4. To select reference frames for MatchGrade to use in the analysis, play through the sequence.
   When you find frames that cover the characteristic colors in the sequence, click on the button to set the current frame as a reference frame. If necessary, you can also use the button to remove a frame from the set of reference frames.
   Typically, the more reference frames you set, the better MatchGrade is able to match the colors.
5. When you're done setting reference frames, use the Transform dropdown menu to choose how to calculate the grade:
   - **3D LUT** - Calculate the grade as a 3D look-up table (LUT). This allows you to export the grade to a *csp* format, which you can use with the OCIOFileTransform node.
   - **CDL** - Calculate the grade as a color decision list (CDL). This allows you to export the grade to an OCIOCDLTransform node.

   **Tip:** You can also set **Target** to **Matched** and **Analysis** to **Auto-analyze Per Frame** to preview where to set reference frames.

   **Note:** The CDL transform is limited and cannot model all types of color transformation. In most cases, selecting **3D LUT** gives the best results.

6. Set **Output** back to **Matched** and click **Analyze Reference Frames**.
   MatchGrade calculates the transform needed to match the **Source** image to the **Target** image.

7. To evaluate the results, select the MatchGrade node and press **D** repeatedly to toggle between the original **Source** image and the graded output.
The color-matched result.

8. If you're not happy with the results, try one or more of the following:
   • Set more reference frames.
   • Use the Pre LUT dropdown menu to specify a 1D shaper LUT to use for the analysis: Linear or Logarithmic.
   • Increase the LUT Resolution value. This is the resolution of the look-up table (LUT) in which MatchGrade stores the color transformation. Higher values can improve the results, but also increase processing time. The maximum value is 64.

To update the results, click Analyze Reference Frames again.

9. If you set Transform to CDL in step 5, you can fine-tune the grade manually by adjusting slope, offset, power, and saturation. The lock controls on the right prevent the values from being recalculated when you click Recalculate CDL, allowing you to set some values manually and estimate others automatically. For example, you can adjust and lock slope and offset and then click Recalculate CDL to automatically recalculate power and saturation.

10. If necessary, you can export the calculated grade and apply it to other nodes in your script. How you do this depends on whether you chose to calculate the grade as a 3D LUT or a CDL:

    • If you set Transform to 3D LUT, you can write the LUT into a .csp file and create an OCIOFileTransform node that you can use elsewhere in the script to apply the same grade. To do so, enter a file path and name for the .csp file in the LUT output file field and click Write to export the LUT. Then, click Create OCIOFileTransform to create an OCIOFileTransform node that applies the color transform from the .csp file.
If you set Transform to CDL, you can click Create OCIOCDLTransform to create an OCIOCDLTransform node that uses the values in the CDL Output section of the MatchGrade properties.

Auto-Analyzing Per Frame

1. In the MatchGrade properties, set Analysis to Auto-analyze Per Frame. MatchGrade calculates the transform needed to match the Source image to the Target image.

2. If you only want to color match certain areas of the image, make sure there is a mask in the Source input's alpha channel. Then, set Mask to either:
   - Alpha - to use the alpha channel as a mask. The grade is limited to the non-black areas of the alpha channel.
   - Inverted Alpha - to invert the alpha channel and use that as a mask. The grade is limited to the non-white areas of the alpha channel.

3. To evaluate the results, select the MatchGrade node and press D repeatedly to toggle between the original Source image and the graded output.

   Alternatively, you can use the Output dropdown menu in the MatchGrade properties to choose what to display in the Viewer:
   - Matched - View the color matched result.
   - Source - View the Source input.
• **Target** - View the **Target** input.

The **Source** image.  

The **Target** image.  

The color-matched result.

4. If you're not happy with the results, try increasing the **LUT Resolution** value. This is the resolution of the look-up table (LUT) in which MatchGrade stores the color transformation. Higher values can improve the results, but also increase processing time. The maximum value is 64.

5. If necessary, you can also use the **Pre LUT** dropdown menu to specify a 1D shaper LUT to use for the analysis:
   - **Auto Detect** - Automatically detect the best pre-LUT to use.
   - **Linear** - Use a linear pre-LUT.
   - **Logarithmic** - Use a logarithmic pre-LUT.
Matching a Grade Between Two Different Clips

1. Select **Color > MatchGrade** to insert a MatchGrade node into your script.
2. Do the following:
   - Connect MatchGrade's **Source** input to the clip to which you want to apply the color transform.
   - Connect the **Target** input to the clip you want to match. This should be a completely different clip, a stereo pair for the **Source** clip, or the **Source** clip in a different format.
   - Connect a Viewer to the output of MatchGrade.

3. In the MatchGrade properties, set **Task** to **Match Different Clip**.
4. Set **Output** to **Source** and then **Target** to compare the **Source** and **Target** images.
5. When you find frames that cover similar content in the **Source** and **Target** clips, click on the buttons next to **Source** and **Target** to set the current frame as a reference frame. Only frames set as reference frames are used in the MatchGrade analysis.

For example, if you want grass in the **Source** clip to have the same color as grass in the **Target** clip, you need to choose frames where the relative amount of grass pixels is roughly the same. The more similar the content, the better MatchGrade is able to match the colors.

**Note:** You do not need to use the same number of **Source** and **Target** reference frames or set the same frames as reference frames for both.
If necessary, you can also use the button to remove a frame from the set of reference frames.

6. If you only want to match certain elements between the Source and Target clips, supply a mask in both the Source and Target inputs' alpha channels and set Mask to either:
   • Alpha - to use the alpha channel from both inputs as a mask. Only the non-black areas of the Source and Target inputs' alpha channels are used in the MatchGrade analysis, and the grade is limited to the non-black areas of the Source alpha.
   • Inverted Alpha - to use the inverted alpha channel from both inputs as a mask. Only the non-white areas of the Source and Target inputs' alpha channels are used in the MatchGrade analysis, and the grade is limited to the non-white areas of the Source alpha.

Masks can also be useful if you cannot find frames that cover approximately the same amount of similar content. For example, if the Source and Target inputs have a different amount of grass and sky, you can use a Roto node in both inputs to create masks that cover the same amount of grass and sky. Bear in mind that if you do so, the grade is only applied to the areas indicated by the mask in the Source input. To apply it to the entire frame, you need to export the grade first. For more information, see step 12.

7. When you’re done setting reference frames, use the Transform dropdown menu to choose how to calculate the grade:
   • 3D LUT - Calculate the grade as a 3D look-up table (LUT). This allows you to export the grade to a .csp format, which you can use with the OCIOFileTransform node.
   • CDL - Calculate the grade as a color decision list (CDL). This allows you to export the grade to an OCIOCDLTransform node.
8. Set **Output** back to **Matched** and click **Analyze Reference Frames**.
   MatchGrade calculates the transform needed to match the **Source** image to the **Target** image.

9. To evaluate the results, select the MatchGrade node and press **D** repeatedly to toggle between the original **Source** image and the graded output.

![The Source image.](image1)
![The Target image.](image2)

The color-matched result.

10. If you're not happy with the results, try one or more of the following:
    - Look for better reference frames or use a mask in the **Target** input to make sure both inputs cover approximately the same amount of similar content.
    - Use the **Pre LUT** dropdown menu to specify a 1D shaper LUT to use for the analysis: **Linear** or **Logarithmic**.
    - Increase the **LUT Resolution** value. This is the resolution of the look-up table (LUT) in which MatchGrade stores the color transformation. Higher values can improve the results, but also increase processing time. The maximum value is 64.

---

**Note:** The **CDL** transform is limited and cannot model all types of color transformation. In most cases, selecting **3D LUT** gives the best results.
Matching a Grade Between Two Different Clips | Auto-Analyzing Per Frame

- Set **Colorspace** to a different value. The correct setting to use depends on the nature of the transformation. Try each option to see which works best with your footage.

- Increase the **Iterations** value. This determines the number of refinement passes. Higher values can produce a better color match, but also take longer to process.

To update the results, click **Analyze Reference Frames** again.

11. If you set **Transform** to **CDL** in step 8, you can fine-tune the grade manually by adjusting **slope**, **offset**, **power**, and **saturation**. The **lock** controls on the right prevent the values from being recalculated when you click **Recalculate CDL**, allowing you to set some values manually and estimate others automatically. For example, you can adjust and lock **slope** and **offset** and then click **Recalculate CDL** to automatically recalculate **power** and **saturation**.

12. If necessary, you can export the calculated grade and apply it to other nodes in your script. How you do this depends on whether you chose to calculate the grade as a 3D LUT or a CDL:

   - If you set **Transform** to **3D LUT**, you can write the LUT into a **.csp** file and create an **OCIOFileTransform** node that you can use elsewhere in the script to apply the same grade. To do so, enter a file path and name for the **.csp** file in the **LUT output file** field and click **Write** to export the LUT. Then, click **Create OCIOFileTransform** to create an **OCIOFileTransform** node that applies the color transform from the **.csp** file.

   • If you set **Transform** to **CDL**, you can click **Create OCIOCDLTransform** to create an **OCIOCDLTransform** node that uses the values in the **CDL Output** section of the MatchGrade properties.
Tip: You can also apply the grade to a different source by simply connecting a new node to MatchGrade's Source input.
Using the Smart Vector Toolset

The Smart Vector toolset allows you to work on one frame in a sequence and then use motion vector information to accurately propagate work throughout the rest of the sequence. The vectors are generated in the SmartVector node and then piped into the VectorDistort node to warp the work. That way, you only need to render the motion vectors once.

Note: All images in this chapter are courtesy of Hollywood Camera Work. See www.hollywoodcamerawork.com for more information.

Quick Start

The toolset workflow is outlined below:

1. Connect the SmartVector node to the source sequence to generate the required motion vectors. You can also write them to file using the .exr format. See Generating Motion Vectors.
2. Find a suitable reference frame and add the required paint corrections or an image to the frame. See Adding Paint to the Source or Adding an Image to the Source.
3. Connect the VectorDistort node to the SmartVector node (or a Read node referencing the .exr files rendered from the SmartVector node) and the source sequence.
4. Premultiply the VectorDistort output and merge it over the source sequence to complete the comp. See Applying Motion Vectors to the Source.
Generating Motion Vectors

The SmartVector node generates motion vectors for use in the VectorDistort node. You can connect a SmartVector node directly to the VectorDistort or VectorCornerPin nodes or write motion vectors to the .exr format, which are then used to drive the warp nodes to reduce overheads later on.

To generate motion vectors:
1. Read in your source sequence and then connect a SmartVector node to the Read node.
2. Double-click the SmartVector node to open its Properties panel, if it's not already open.
3. Set the Vector Detail control to the required detail level. The default value of 0.3 is sufficient for sequences with low detail and movement, but you may want to increase the detail to improve the vector quality in some cases.
   For example, if the area you’re working on is relatively small, the default value of 0.3 might not capture movement properly. Try increasing the control to 1.0 to capture more detail.

   **Note:** High detail vectors take longer to render, but can improve the results you get from the VectorDistort node.

4. Set the Strength control to force pixel matching between frames. Higher values allow you to accurately match similar pixels in one image to another, concentrating on detail matching even if the resulting motion field is jagged. Lower values may miss local detail, but are less likely to provide you with the odd spurious vector, producing smoother results.

   **Note:** The default value works well for most sequences.

5. If there is a lot of movement in the foreground, you might want to add a mask. See Masking Foreground and Background Areas for more information.

   **Tip:** You can examine the vectors produced by connecting a Viewer to the SmartVector node and switching the channels control above the Viewer to smartvector_<value>.

6. If you want to write the vectors to disk, click Export Write to automatically add a Write node to the script. The Write node's controls are set to channels > all and .exr output automatically.
   You can only write motion vectors to the .exr format. Don't forget to add frame padding in the form of hashes or printf notation, depending on your Preferences > Behaviors > File Handling settings.
7. Enter a **file** path in the Write node’s controls and then click **Render**.

**Note:** SmartVector does not currently output **motion**, **forward**, and **backward** channels by default. If you require these channels, add a VectorToMotion node after the SmartVector node. VectorToMotion converts the vectors to motion that can be used with VectorBlur to create motion blur, without using a VectorGenerator.

8. Enable **Advanced > Flicker Compensation** to reduce variations in luminance and overall flickering, such as light reflecting from shiny surfaces, which can cause problems with your vector output.

9. **Tolerances** in the **Advanced** section allow you to tune the weight of each color channel when calculating the image luminance.

   These parameters rarely need tuning, but you may wish to increase the weighting on a particular color to allow the algorithm to concentrate on getting the motion for a certain object correct, at the cost of the rest of the items in a shot.

10. Proceed to **Adding Paint to the Source** or **Adding an Image to the Source**.

### Masking Foreground and Background Areas

If your sequence is composed of a foreground object moving over a background, the motion estimation is likely to get confused at the edge between the two. To reduce artifacts, you can add an alpha mask to the **Matte** input.

To apply a mask to the sequence:

1. Add a Roto node downstream of the source.
2. Draw a matte around the area you want to mark as foreground or background.
3. Set the **Matte Channel** to **Matte Alpha** and the **Output** control to either **Foreground** or **Background**.

![Foreground vectors](image1.png) ![Background vectors](image2.png)

4. If you're working with **Background** vectors, you can enable **Inpaint Matte Region** to automatically fill in the missing foreground vectors using the nearest available background vectors. The **Matte Dilation** slider controls amount of dilation applied to the matte before inpainting is applied.

![Background vectors](image3.png) ![Inpainted foreground](image4.png)

You can use inpainting to effectively ignore the foreground, allowing you to work with background vectors in the masked area.

5. Generate the motion vectors as described in **Generating Motion Vectors**.
Adding Paint to the Source

The Smart Vector toolset minimizes the paint work needed to create your final comp by propagating paint through a sequence from a single source frame.

To add paint to your sequence:
1. Scrub through your sequence to find a good reference frame. Good reference frames should:
   • be at the point of least motion, also known as the motion apex,
   • contain minimal motion blur, and
   • avoid areas of occlusion.
2. Add a RotoPaint node and apply your paint corrections to the selected reference frame. Ensure that all your paint is sourced from the background image by navigating to Stroke > source and selecting background once a stroke has been added.
3. Proceed to Applying Motion Vectors to the Source.
Adding an Image to the Source

The Smart Vector toolset can also propagate an image through a sequence from a single source frame.

To add an image to your sequence:
1. Scrub through your sequence to find a good reference frame. Good reference frames should:
   • be at the point of least motion, also known as the motion apex,
   • contain minimal motion blur, and
   • avoid areas of occlusion.
2. Merge your image into the node tree at the selected reference frame.

3. Proceed to Applying Motion Vectors to the Source.
Applying Motion Vectors to the Source

Once you've generated your motion vectors and added your paint to the source sequence, the VectorDistort node takes the paint from the reference frame and propagates it through the rest of the sequence using the motion vectors from the SmartVector node.

Warping the Vectors

After generating your vectors, applying your corrections, and masking out areas of the sequence you don't want to process, use VectorDistort to push your work through the rest of the sequence.

1. Add a VectorDistort node to the Node Graph and connect the **Source** input to your sequence, downstream of the paint or image you intend to propagate.
2. Connect the **SmartVector** input to either the SmartVector node or a Read node referencing the .exr files containing the motion vector data.
3. Double-click the VectorDistort node to open its **Properties** panel, if it's not already open.
4. Scrub to the frame containing your paint or image and click **set to current frame** to set the VectorDistort reference frame.

**Tip:** If you know what frame your paint or image is applied to, you can type the frame number into the **Reference Frame** control manually.

Scrubbing in the Viewer at this stage introduces undesirable warping. As you move farther from the reference frame, the warping increases.
Warp close to the reference frame.  

Warp far from the reference frame.

5. Set the maximum **Frame Distance** over which vectors are calculated. Sequences with rapid motion typically require values closer to **1 frame**, whereas sequences with slower motion typically require values closer to **64 frames**.

See **Improving Warps** for more information on the **Frame Distance** control.

6. To apply the warped paint or image to the original sequence, premultiply the output from the VectorDistort nodes and then merge the result over the source.

For short sequences, or sequences with minimal movement and detail, your corrections are propagated nicely as shown.
If you see that your corrections warp incorrectly over time, you can use VectorDistort's properties to massage the results (see Improving Warps) or use the VectorCornerPin node or multiple VectorDistort nodes to apply warps for several reference frames (see Warping Images Using VectorCornerPin and Warping Multiple Reference Frames).
Improving Warps

For short sequences, or sequences with minimal movement and detail, your corrections are propagated nicely. If you see that your corrections warp incorrectly over time, you can use VectorDistort's **Frame Distance** control to massage the results.

**Tip:** Warping relies on good vectors from SmartVector and a good reference frame in VectorDistort. Before doing anything else, try increasing the **Vector Detail** control in the SmartVector node and re-rendering the vectors. You can also try selecting a different **Reference Frame** in the VectorDistort node's controls.

See [Generating Motion Vectors](#) or [Applying Motion Vectors to the Source](#) for more information.

At **Frame Distance** > 1 frame, VectorDistort calculates warp for every frame in the sequence, that is, frames 1-2, 2-3, 3-4, 4-5, and so on. For frames close to the **Reference Frame** with lots of movement between frames, this is a good thing as the warp needs to change significantly from frame to frame. However, for frames farther away from the **Reference Frame** with little movement between frames, this isn't required as the warp doesn't change significantly from frame to frame.

Increasing the **Frame Distance** essentially reduces the number of warps calculated between the **Reference Frame** and the current frame. In the example sequence, the **Reference Frame** is set to 57, and the warp at frame 230 has started to slip on the subject's forehead with **Frame Distance** set to 1 frame. Increasing the **Frame Distance** works well up to around 2 frames, but increasing the distance distorts the warp too much.
For longer sequences with local distortion, you can try increasing the **Blur Size** control to blur the internally calculated STMap. Increasing the **Blur Size** can remove local distortions in the warped result, particularly in longer sequences.

In some cases, no amount of adjustment is going to improve the warp over a large number of frames - particularly if there's a lot of movement or detail in the sequence. See [Warping Images Using VectorCornerPin](#) and [Warping Multiple Reference Frames](#) for more information on how to use VectorCornerPin and multiple VectorDistort nodes to minimize manual paint work.
Warping Images Using VectorCornerPin

Some sequences involve movement and detail that can't be propagated from a single frame correction. In these cases, you can use the VectorCornerPin node to minimize the amount of correction work needed. The to1-4 Viewer widgets allow you to add keyframes, which then use vector information from SmartVector to drive the warp.

The final result should be an image with minimal slip and distortion.

1. Set up your image as described in Adding an Image to the Source.
2. Connect the VectorCornerPin's SmartVector input to either a SmartVector node or a Read node referencing the .exr files containing the motion vector data.
Tip: If the image you're warping displays at the wrong resolution, switch to the VectorCornerPin's From tab, click Set To Input, and then switch back to the VectorCornerPin tab and click Copy ‘from’ to resize it automatically.

3. Scrub to the reference frame in your sequence and place the image as required using the to1-4 Viewer widgets.
4. In the VectorCornerPin Properties panel, click add keyframe \[+\] to set a keyframe at the current frame.

**Note:** You can remove the keyframe at the current frame using \[-\] or remove all keyframes from the sequence using \[\text{Ctrl}+\text{R}\].

5. Play through the sequence until the image begins to slip or warp.

**Tip:** After adding a keyframe, you can click Bake Corners to calculate the positions of the pins at each frame in a range using the vectors from the SmartVector input.
6. Adjust the image using the **to1-4** Viewer widgets. A new keyframe is added at the current frame. Frames on which no keyframe is present are interpolated automatically, so you may not need to adjust each frame individually.

7. You can smooth distortion on interpolated frames using the **Frame Distance** and **Blur Size** controls on the **SmartVector** tab.

Frame Distance **1 frame** and Blur Size **0**.
Frame Distance 4 frames and Blur Size 10.
Warping Multiple Reference Frames

Some sequences involve movement and detail that can't be propagated from a single frame correction. In these cases, you can use multiple VectorDistort nodes with different reference frames to minimize the amount of correction work needed.

1. Set up your first paint correction as described in Adding Paint to the Source and Applying Motion Vectors to the Source.
2. In the VectorDistort node's properties, disable the Hold Frame control.
3. Add a FrameHold node before the RotoPaint node containing your corrections.
4. Set the first frame control to the reference frame specified in the VectorDistort node.
5. Set the lifetime of your paint strokes in the RotoPaint node Lifetime tab to the same frame range as the VectorDistort node.

6. Repeat the process for the required number of reference frames to complete your corrections. An example node tree might appear as follows:
Warping Multiple Reference Frames | Warping the Vectors

Diagram showing the workflow of warping multiple reference frames using Nuke's vector distortion node.
Generating Depth Maps

You can use the DepthGenerator node in NukeX to generate a depth map from your footage. The node uses information from a tracked camera to create a channel that displays variations in depth. A depth map is an image that uses the brightness of each pixel to specify the distance between the 3D scene point and the virtual camera used to capture the scene.

DepthGenerator also allows you to output depth as normals and position passes, and create a Card node positioned in 3D space and displaced according to the depth channel.

Note that DepthGenerator can only create a depth map from a sequence, so you can't use a still image.

Quick Start

Here's a quick overview of the workflow:

1. Add DepthGenerator to your NukeX script. See Connecting DepthGenerator.
2. Select the type of depth map you want to output, and whether to also output normals and position passes. See Selecting What to Output.
3. Analyze the depth values for your footage. See Analyzing Depth.
4. Review and refine the results. See Refining the Results.
5. Use the results with other Nuke and NukeX nodes. See Using the Results.
Connecting DepthGenerator

To connect DepthGenerator:

1. To use the DepthGenerator node, you need a tracked camera that matches your footage. If you don’t already have one, you can create one with the CameraTracker node (see Camera Tracking).
2. Create a DepthGenerator node by clicking 3D > DepthGenerator.
3. Attach a Viewer to the output of DepthGenerator.
4. Connect the Source input to your footage and the Camera input to your Camera node.

![A simple DepthGenerator node tree.](image)

5. Depth can only be calculated where the real world 3D position of objects doesn’t change. If there are moving objects in your Source footage, DepthGenerator is likely to struggle to create an accurate depth map for those regions. To prevent this, you can exclude moving foreground regions from the depth calculation by connecting a matte to the Mask input and setting Ignore Mask in the DepthGenerator properties to the channel that contains the matte. Note that DepthGenerator expects values of either 0 (for regions to use) or 1 (for regions to ignore).

![A Source image.](image) ![An ignore mask.](image)

The Mask input only appears once you’ve connected the other two inputs.
6. Proceed to Selecting What to Output.
Selecting What to Output

To select what to output:

1. In the DepthGenerator properties, use **Depth Output** to select the type of depth map you want to generate:
   - **Depth (1/Z)** - output 1/Z where Z is the distance along the Z axis for the camera. This matches the depth output of the ScanlineRender node.
     This mode also allows you to later create a Group node with a Card positioned in 3D space and displaced according to the depth channel.
   - **Distance** - output the distance along the ray from the camera center to the 3D surface point.
     In the figure below, each pixel forms a ray, and AB measures the physical distance from the camera to the 3D point, whereas AC measures the distance along the Z axis for the camera.

![Diagram showing depth calculation](image)

Calculating the depth.

2. If you also want to output depth as a position pass, set **Surface Point** to the channels where you want to store this (for example, you could create a new layer called `ppass`, which contains the channels `X`, `Y`, and `Z`).
   The position pass includes the X, Y, and Z coordinates for each pixel in the image. You can use it with the Relight node, or with the PositionToPoints node to visualize depth as a point cloud.

![Position pass](image)

A position pass.
3. Similarly, to output depth as a normals pass, set **Surface Normal** to the channels where you want to store the normals pass (for example, create a new layer called **npass**, which contains the channels **X**, **Y**, and **Z**).

The normals pass contains three vectors of information for each pixel in the image: X direction, Y direction, and Z direction. In other words, it stores the direction in which each point in the image is facing. You can use a normals pass with the Relight node.

![A normals pass.](image)

4. Proceed to **Analyzing Depth** below.
Analyzing Depth

To analyze the depth:

1. Do one of the following:
   • Use the **Frame Separation** control to select the offset between the current frame and the frame against which to calculate depth for your footage. For example, if your current frame is 100, and your frame separation is 2, DepthGenerator uses frame 98 and frame 102 to generate the depth map.
   
   Ideally, you want frames close together so that the images are similar and include the same bits of the world. However, you get more accurate depth when the frames are further apart and the baseline between the cameras is bigger. So, for fast camera moves, you need a small frame separation, and for slow camera moves, you can use a larger frame separation. To change the separation for fast and slow movements, you can animate the **Frame Separation** control.
   
   Note that this control does NOT affect the number of frames used in the calculation, as that is always 2.
   
   • To have DepthGenerator attempt to calculate the best **Frame Separation** automatically, click **Analyze Sequence**. For this to work, the camera in the **Camera** input must be defined for all frames within the frame range.
   
   DepthGenerator goes through the entire sequence, attempts to work out the correct frame separation, and animates the **Frame Separation** value.
   
   • Alternatively, you can also click **Analyze Frame** to have DepthGenerator automatically calculate the frame separation for the current frame. This gives you more control than **Analyze Sequence**, as you can work through the timeline, analyze particular frames, and if necessary tweak the **Frame Separation** value manually.

2. Scrub through the timeline and keep an eye on the **Calculated Accuracy** value. This displays the depth accuracy calculated when analyzing frame separation. Values closer to 1 are considered accurate, and values closer to 0 inaccurate.

   You can use the frames that produce accurate values later when placing elements in 3D (for example, by clicking **Create Card** or using a PositionToPoints node).

   If you’re not getting accurate depth values, try adjusting **Frame Separation** or using an **Ignore Mask**.

3. View the depth channel in 2D by selecting it from the list of channels in the Viewer. To see more details in the depth map, it can be a good idea to temporarily adjust the gain and gamma controls in the Viewer.
4. If you have set **Surface Point** or **Surface Normal** to generate position and normals passes, you can also view those by selecting them from the list of channels in the Viewer.

5. Proceed to **Refining the Results**.
Refining the Results

To refine the results:

1. Adjust **Depth Detail** to vary the resolution of the images used to calculate the depth map. The default value of 0.5 equals half the image resolution. Lower values speed up processing and deliver a smoother result. Higher values pick up finer details, but also increase processing time.

   ![Depth Detail comparison](image1.png)

   Depth Detail = 0.15  
   Depth Detail = 0.85

2. If you find that the depth map is noisy in flat image regions, try increasing the **Noise** value in the **Depth Generation** parameter group. This sets the amount of noise DepthGenerator should ignore in the input footage when calculating the depth map. The higher the value, the smoother the depth map.

   ![Noise comparison](image2.png)

   Noise = 0  
   Noise = 0.15

3. If you think the depth map should still be smoother, you can also increase **Smoothness**. Rather than affecting the depth calculation, this applies an intelligent blur on the result.

   A high smoothness can miss lots of local detail, but is less likely to produce noisy depth values. A low smoothness concentrates on detail matching, even if the resulting depth map is jagged.

4. The **Strength** parameter defines the strength of pixel matching between frames. You can usually leave it at the default value. However, if the depth map does not appear to fit some edges of the image, you can increase **Strength** to force matches where fine details are missed. You can also reduce **Strength** to smooth the depth map, but generally it's a good idea to leave it at a relatively high value to make sure the algorithm is matching the images and calculating the depth correctly.
5. If you aren't happy with the object boundaries in your depth map, adjust **Sharpness**. Use a high value to produce distinct boundaries between the objects, or a low value to smooth the boundaries.

6. If necessary, use **Near Clip Plane** and **Far Clip Plane** to set the minimum and maximum values allowed in the depth map. All depth values that are outside this range are clipped to these values.

7. To see if there are any regions where the depth calculation is ambiguous, check **Mark Bad Regions**.
   Any ambiguous regions are marked as very large values in the depth map. Such regions might occur if CameraTracker hasn't been able to calculate the depth of all pixels using the camera data, due to certain camera movements for instance.

8. Once you've set the parameters so that you're happy with the depth map, view the normals pass (if you decided to generate one earlier). Normals are calculated from depth, and the normal direction is defined by the change in depth between pixels. The normals pass can be noisy, as
small changes in depth lead to large changes in the normals. To smooth out the normal map by calculating normals using a lower resolution depth map, reduce Normal Detail. This integrates the change in depth across a wider range of pixels, reducing noise in the normals pass. The default value of 0.25 causes the normals pass to be calculated at a quarter resolution.

Normal Detail = 0.2
Normal Detail = 0.8

9. Proceed to Using the Results.
Using the Results

Once you are happy with the results from DepthGenerator, there are several ways you can use them in Nuke and NukeX:

- If you set **Depth Output** to **Depth 1/Z**, you can click **Create Card** in the DepthGenerator controls to create a Group node with a Card positioned in 3D space and displaced according to the depth channel. This allows you to visualize depth as a surface in 3D space, making it easier to place elements (such as Cards) at the correct depth in a live-action shot.

  ![A Card node displaced according to the depth channel.](image)

- If you set DepthGenerator to create a position pass, you can supply the position pass to the PositionToPoints node (3D > Geometry > PositionToPoints) to create a 3D point cloud. This is another good way to visualize depth in 3D space.

  For more information, see **Creating a Dense Point Cloud Using the PositionToPoints Node**.

- If you used DepthGenerator to create both a position pass and a normals pass, you can supply those to the Relight node (3D > Lights > Relight). This node applies a shader to its input image based on the position and normals passes, creating realistic lighting.

  For more information, see **Relighting a 2D Image Using 3D Lights**.

- You may also be able to use the depth map if you want to introduce fog and depth-of-field effects into a shot. In Nuke, the ZDefocus node requires a depth map in its input.

  For more information, see **Simulating Depth-of-Field Blurring**.
Creating Dense Point Clouds

Dense point clouds are a useful starting point for 3D modeling and can be helpful in positioning 3D objects into a scene. Using the PointCloudGenerator node, you can create a dense point cloud based on the information generated by CameraTracker and use the points to create a 3D mesh of your 2D footage.

Quick Start

Here's a quick overview of the workflow:

1. Make sure your PointCloudGenerator node is connected to the appropriate footage and Camera node. For more information, see Connecting the PointCloudGenerator Node.
2. Set keyframes in the footage to determine which frames are tracked to create the point cloud. For more information, see Setting Keyframes in a Sequence.
3. Track the footage to create a dense point cloud. For more information, see Tracking a Dense Point Cloud.

![A 2D source image.](image1) ![The resulting 3D point cloud.](image2)

4. Filter points from the result to adjust it and eliminate bad points. For more information, see Filtering Your Point Cloud.
5. Create groups within the cloud to aid visualization or bake out grouped sections. For more information, see Grouping, Labeling, and Baking Points.
6. Then if you need to, you can move on to creating meshes. For more information, see Creating a Mesh from a Point Cloud.
7. Alternatively, use the PoissonMesh node to create a mesh. For more information, see Using the PoissonMesh Node.
Connecting the PointCloudGenerator Node

To create a dense point cloud, PointCloudGenerator needs a Camera node with a solved camera path, either from CameraTracker or from a third party 3D application, and the tracked 2D footage.

To connect the node, do the following:
1. Click 3D > Geometry > PointCloudGenerator.
2. Connect the necessary nodes to the inputs:
   - Source - the 2D footage from which the camera track was solved.
   - Camera - the solved Camera node.
3. If you want to mask part of the scene to avoid tracking moving objects or reflections, connect a matte to the Mask input. For more information about masking, see Masking Out Regions of the Image.
4. Click Image > Viewer to insert a Viewer node and connect it to the PointCloudGenerator node.
5. Proceed to Setting Keyframes in a Sequence.
Masking Out Regions of the Image

If you don't want to track all regions of your image, for example, if there are moving objects or reflections in the image, you can attach a matte to the Mask input to define image regions that should not be tracked. You can also use the source input's alpha channel as a matte.

1. If you want to use a separate matte for masking, connect it to the Mask input in the Node Graph.
2. In the properties panel, set Ignore Mask to the component you want to use as a mask:
   - None - Track features in the whole sequence.
   - Source Alpha - Use the alpha channel of the source clip to define which areas to ignore.
   - Source Inverted Alpha - Use the inverted alpha channel of the source clip to define which areas to ignore.
   - Mask Luminance - Use the luminance of the mask input to define which areas to ignore.
   - Mask Inverted Luminance - Use the inverted luminance of the mask input to define which areas to ignore.
   - Mask Alpha - Use the mask input alpha channel to define which areas to ignore.
   - Mask Inverted Alpha - Use the inverted mask input alpha channel to define which areas to ignore.

Note: PointCloudGenerator can track multiple ranges, adding the new points to the existing cloud, so feel free to mask and re-track as much as necessary.
Setting Keyframes in a Sequence

Keyframes represent the best frames in a sequence to perform a particular function, in this case tracking. PointCloudGenerator can analyze the sequence for you, automatically picking frames with good parallax, or you can set keyframes manually by marking the frames yourself.

Setting Keyframes Automatically

1. Click Analyze Sequence.
   PointCloudGenerator walks the sequence, adding keyframes in suitable areas.
2. Once the process is complete, scrub the playhead and use the Calculated Accuracy field to evaluate the keyframes - values closer to 1 indicate a high accuracy.

   **Note:** You can also add keyframes to the sequence manually. See below for more information.

3. Proceed to Tracking a Dense Point Cloud.

Setting Keyframes Manually

1. Scrub the playhead to the frame you want to mark as a keyframe and click Add,
   OR
   Set the Frame Spacing field to the number of frames between keyframes throughout the sequence and click Add All.
   For example, if you set this value to 2, keyframes are set at every other frame throughout your sequence.
2. If necessary, click Delete to remove the keyframe at the current playhead position or click Delete All to remove all keyframes.

   **Note:** The keyframes generated automatically using Analyze Sequence are not deleted.

3. Proceed to Tracking a Dense Point Cloud.
Tracking a Dense Point Cloud

The next step towards creating a dense point cloud is to track your footage for more 3D feature points using the information from keyframes in the sequence and the solved Camera.

1. Click **Track Points** to track the sequence and create a dense point cloud.

   An example 3D point cloud.

   The default settings work well on most sequences, but you can adjust the **Dense Tracking** controls to alter the appearance of the cloud:

2. Adjust the **Point Separation** control to set the separation value, in pixels, between points in the point cloud.

   As you can see, lower values produce denser clouds, but at the expense of processing time.

   **Point Separation 0.** **Point Separation 10.**

3. Set the **Track Threshold** to compare the similarity of features over a number of frames and reject them if they exceed the threshold. You can adjust this value to test whether a track is reliable.

   **Low threshold.** **High threshold.**
Note: If the threshold is set too high, you may find that you don't get any reliable tracks at all. The red pixels in the above image are rejected tracks.

4. Proceed to Filtering Your Point Cloud.
Filtering Your Point Cloud

You can filter your point cloud by adjusting the number and quality of points it includes. NukeX filters the tracked point cloud directly, without recalculating, streamlining your workflow.

1. Ensure that **Display rejected points** is enabled and adjust the threshold controls to update the 3D Viewer dynamically. Rejected tracks are highlighted in red.

2. Adjust the **Angle Threshold** to set the minimum acceptable angle to triangulate 3D points (in degrees). Points with a large triangulation angle tend to be more accurate.

   Set a threshold of 0 to triangulate all points or increase the threshold to highlight the least accurate points.

   **Tip:** As a rule of thumb, anything below 5 degrees is likely to be incorrect.

3. Adjust the **Density Threshold** to set the minimum acceptable point density. Isolated points tend to be less accurate.

   Set a threshold of 0 to output all points or increase the threshold to highlight the most isolated, less accurate points.

4. The Viewer **wipe** tool can help you locate isolated points by comparing the 2D source footage to the point cloud.

   Set Input A to the PointCloudGenerator node and Input B to the Read node for the source image.

   See **Using the Viewer Controls** for more information.

5. Proceed to **Removing Rejected Points**.
Removing Rejected Points

Once you've identified and highlighted the less accurate points:

1. Click **Delete Rejected Points** to automatically remove all highlighted points.
2. You can then be a little more selective by switching the Viewer to **Vertex selection** mode.

3. Use the **Output** controls to assist you when visualizing your point cloud:
   - **Point Size** - set the pixel size of points in your cloud.
   - **Output points per frame** - by default, all tracked points in your sequence are displayed in the Viewer. Enabling this control allows you to view only the points generated at the current frame.

   **Note:** If you’re having trouble viewing points individually, try reducing the **3D handle size** in the **Preferences > Panels > Viewer Handles** tab.

4. Manually select points in the cloud using the 3D Viewer by:
   - Dragging a marquee over the required points in the Viewer, or
   - Holding **Shift**, and dragging a marquee over several selection areas in the Viewer to select all points at once.
Note: You can remove points from a selection by holding Shift+Alt and re-selecting highlighted points.

5. Press Delete/Backspace or right-click in the Viewer and select delete selected to remove the points.
6. Proceed to Grouping, Labeling, and Baking Points.
Grouping, Labeling, and Baking Points

Point clouds can be awkward to interpret, especially if the Point Separation control is set to a relatively high value. Grouping points in the cloud can help to visualize the scene, particularly when sensibly labeled and colored in the Viewer.

Groups can also be baked out as separate point clouds or converted into meshes. See Creating a Mesh from a Point Cloud for more information.

To create a group from a point cloud:
1. Manually select group points in the 3D Viewer by:
   - Dragging a marquee over the required points in the Viewer, or
   - Holding Shift, and dragging a marquee over several selection areas in the Viewer to select all points at once.

   **Note:** You can remove points from a selection by holding Alt + Shift and re-selecting highlighted points.

2. Right-click on a highlighted point and select create group.

   ![Image of create group option]

   The new group is added to the Groups tab.

   **Note:** If groups already exist, the option to add the selected points to a group is enabled.

3. Add more groups as required to build up the visualization of the scene.
4. In the properties panel, select the Groups tab to display the list of existing groups. Use the Groups controls to determine the appearance of your groups:
- **Display groups in overlay** - when enabled, groups are highlighted in the Viewer using their associated RGB color.
- **Create Group** - click to add the selected points in the Viewer to a new group.
- **Delete Selected Groups** - click to delete all the selected groups in the list.

5. Double-click the table columns to edit group names or colors, and toggle visibility.
6. Enable **Output visible groups only** to display only selected groups in the Viewer.

![Image of point cloud with highlighted groups]

7. After creating the required groups, you can split them off into individual point clouds, using **Bake Selected Groups**, or create group meshes.
8. Proceed to **Creating a Mesh from a Point Cloud**
Creating a Mesh from a Point Cloud

PointCloudGenerator can create meshes from grouped points in the point cloud that you can use as stand-alone 3D objects, for example, in 3D modeling. You can also use these meshes to quickly project the 2D sequence onto the mesh using the Project3D node.

PointCloudGenerator meshes are based on the Poisson Surface Reconstruction calculation method. The original source code and paper were created by Michael Kazhdan, Matthew Bolitho, and Hugues Hoppe (see http://www.cs.jhu.edu/~misha/Code/PoissonRecon/ for more information).

To create a mesh from a group:

1. On the Groups tab, select the required group(s) from the Groups list.
   You can mesh a single group or all the groups available in the list.
2. Click Bake Selected Groups to Mesh.
   A mesh is created automatically using the information from the PointCloudGenerator node.

   **Note:** You can use the resulting group mesh node in Nuke as well, though you can’t edit the geometry.

3. Proceed with Adding Texture to a Mesh.

Many factors influence the quality of any mesh, chief among which are the contents of the scene and the camera track obtained from the sequence. If you consider a simple scene, such as that shown in the example images, the resulting mesh can be quite accurate.
Adding Texture to a Mesh

You can quickly add texture to your point cloud mesh using the Project3D node. Do the following:

1. Click 3D > Shader > Project3D.
2. Connect the solved Camera node to the Project3D cam input.
3. Connect the other input of the Project3D node into the sequence.
4. Connect the Group1_Mesh node img input to the Project 3D node.

The mesh is now textured with your original 2D sequence.

**Tip:** You can export your camera, point cloud, and meshes as Alembic (.abc) objects to other applications. For more information about exporting .abc files, see Exporting Geometry, Cameras, Lights, Axes, or Point Clouds.
Using the PoissonMesh Node

The PoissonMesh node uses information from a dense point cloud to generate a mesh that you can further use as a 3D object, in 3D modeling for instance. The PoissonMesh node is based on the Poisson Surface Reconstruction calculation method. The original source code and paper were created by Michael Kazhdan, Matthew Bolitho and Hugues Hoppe (see http://www.cs.jhu.edu/~misha/Code/PoissonRecon/ for more information).

Before using the PoissonMesh node, you need a dense point cloud with normals such as the one created by the PointCloudGenerator node. For more information on creating a dense point cloud, see Tracking a Dense Point Cloud on page 1670. When you've got one, you're ready to create a mesh:

1. Connect the PoissonMesh node to a PointCloudGenerator node.
2. A mesh is created automatically using the information from the PointCloudGenerator node and you can view it in the 3D view.
3. If you want to, you can tell PoissonMesh to use the selected points only in the point cloud to create a mesh. To do this, check the Use Selection box in the properties panel. You can also use the Filtering box to use the PointCloudGenerator filtering to refine your mesh.
4. You can also adjust other properties for your mesh on the PoissonMesh tab:
   • **Depth** - set the maximum depth (in integers) for the calculation tree used to create your mesh. You can enter values up to 10, but increasing this value increases the solver's memory usage, so it's a good idea to keep this as low as possible.
   • **Scale** - set the ratio between the 3D cube used to create the mesh and the bounding box of the footage it's created from.
   • **Solver Divide** - set the depth at which the solver equation is used in the creation of your mesh. This can help you reduce your memory usage when generating complex meshes.
   • **Iso Divide** - set the depth at which the iso-surface extractor is used in extraction. This can help you reduce your memory usage when generating complex meshes.
   • **Samples per Node** - set the minimum number of sample points used to create your mesh. Use larger values if your original footage is very noisy.
   • **Confidence** - check to use the size of the normals as confidence information when generating the mesh. This may take longer, but gives you better results since your point cloud point creation gets double-checked.

**Note:** You can only use the PoissonMesh node in NukeX as the information used to create the mesh is not stored in the node and therefore cannot be recreated for viewing in Nuke.
Adding Texture to the PoissonMesh

You can add texture to your point cloud mesh using the Project3D and ApplyMaterial nodes. Do the following:

1. Connect your texture node to the Project3D node.
2. Then connect the Project3D node to the mat input of the ApplyMaterial.
3. Connect the other input of the ApplyMaterial node into the PoissonMesh node. The mesh is now textured with your texture.

Tip: You can export both your point cloud and the mesh as FBX objects to other applications, but don't forget to export the Camera node too. For more information about exporting FBX files, see Exporting Geometry, Cameras, Lights, Axes, or Point Clouds.
Using ModelBuilder

The ModelBuilder node provides an easy way to create 3D models for 2D shots. You can build a model by creating shapes and then editing them, and align models over your 2D footage by dragging vertices to their corresponding 2D location.

Prerequisites

To be able to align models, ModelBuilder needs a tracked camera and an input image for visual reference. You can also use other 3D geometry and point clouds as a reference if you already have these for your scene.

**Note:** You can use ModelBuilder without a camera, image sequence, or reference geometry. In this case, all the Edit mode features work just fine; it just means you aren't able to do anything in Align mode.

Creating and editing models using ModelBuilder requires a NukeX license, but the resulting geometry can also be used in Nuke.

A 2D source image with shapes created using ModelBuilder.

Special thanks to belowthetradar TV for use of the above footage, used throughout this chapter.
Quick Start

Here's a quick overview of the workflow:

1. Connect the ModelBuilder node to your footage and a changing Camera. For more information, see Connecting the ModelBuilder Node.
2. As a starting point for your model, create one or more of the built-in 3D shapes using the toolbar on the left hand side of the Viewer.
   See Creating Shapes.
3. If necessary, adjust the display characteristics of your shapes so you can better see them in the Viewer. You can do this at any point during the process.
   See Editing Shapes’ Display Characteristics.
4. Position the shape using your 2D footage or existing 3D geometry as a reference.
   See Positioning Shapes.
5. Edit the shape until you're happy with your model.
   See Editing Shapes.
6. Once you’re happy with your model, you're ready to texture it.
   See Applying Textures.
7. If necessary, you can also create a separate geometry node for the selected shapes and use that as you would use any other geometry node in Nuke.
   See Exporting Shapes to Separate Geometry Nodes.
Connecting the ModelBuilder Node

To connect the ModelBuilder node:

1. If you don’t already have a tracked camera that matches your footage, create one using the CameraTracker node (see Camera Tracking).
   ModelBuilder needs a changing Camera (with a change of more than 5 degrees) in order to calculate 3D points from your 2D footage.

2. Select 3D > Geometry > ModelBuilder to create a ModelBuilder node.

3. Connect your Camera node to ModelBuilder’s cam input, and your 2D footage to its src input.
   Note that the objects in your src footage that you want to model should not be moving, as ModelBuilder works best with static objects.

4. If you have 3D geometry (such as a shape or a point cloud) that you want to use as a reference when positioning new shapes, connect it to ModelBuilder’s geo input.
   For example, you can use the PointCloudGenerator node (see Creating Dense Point Clouds) to create a 3D point cloud for your shot, and connect that to the geo input. When creating new shapes, you can then select vertices on the point cloud to automatically align the shapes with the point cloud. This gives you an approximate initial position for your shapes.

5. If you have a 2D texture image that you want to display as a background in the UV preview window and on the model in the 3D Viewer, connect it to ModelBuilder’s tex input.

6. Attach a Viewer to the output of the ModelBuilder node.

7. Proceed to Creating Shapes below.
Creating Shapes

To create shapes:

1. Double-click on the ModelBuilder node to open its properties.

2. Use the shape creation menu in the ModelBuilder toolbar on the left hand side of the Viewer to select the basic 3D shape that best matches the object you are trying to model. You can select between Point, Card, Cube, Sphere, Cone, Cylinder, and Polygon (when creating a polygon, click on the Viewer to create vertices where you need them, and when you’re done, press Return). A colored border appears around the Viewer window to indicate that the action of creating a shape is in progress.

   ![ModelBuilder interface with a yellow circle under the cursor.]

   **Tip:** If you want to model a simple wall, a Card might be a good starting point. For an entire building, you may want to use a Cube instead.

You can adjust the shapes (for example, the number of rows or columns in them) on the Shape Defaults tab. These controls only apply when creating a shape - you cannot adjust them after the fact.

3. Hover over the Viewer. You should see a yellow circle appear under the cursor. You can now:
• click in the Viewer to set where you want the new shape created. If you click over empty space, the shape is created at a distance specified by the **Distance** control. There are no specific units for this value; low values position the shape close to the camera, and higher values mean further away.

• hold down **Shift** and click to snap to the surface of any geometry under the cursor.

• drag the mouse left or right to scale the shape.

The shape you selected appears in the Viewer and in the Scene list in the ModelBuilder properties.

4. If the shape was created too close to the camera or too far away, Activate **Edit Mode** in the ModelBuilder toolbar and make sure the selection mode menu is set to **Select objects**. Then, select the shape in the Viewer, and drag the transform handles to reposition it.

5. If you have more than one object that you want to model in your src footage, you can repeat the above steps to create several shapes using a single ModelBuilder node.

6. Proceed to **Editing Shapes’ Display Characteristics** below.

**Tip:** You can also use the Create buttons on the Shape Defaults tab to create shapes. This is exactly the same as using the shape creation menu in the Viewer.

**Tip:** If you want to use 3D geometry as a visual reference and connected it to the geo input, before creating a shape, you need to:

1. Activate **Edit Mode** in the ModelBuilder toolbar on the left hand side of the Viewer and set the selection mode menu to **Select vertices**.

2. Drag in the Viewer to select vertices on the object that you want to model, bearing in mind that when you create a new shape, it’s automatically aligned with the selected vertices. If you cannot see the object in the geo input, make sure **Pass Through Geo** is enabled in the ModelBuilder properties.

3. Use the shape creation menu in the ModelBuilder toolbar to select the basic 3D
shape that best matches the object you are trying to model.

ModelBuilder creates the shape to align with the selected vertices.
Editing Shapes’ Display Characteristics

To edit shapes' display characteristics:

1. If you created a lot of shapes, you may want to organize them into groups. Click the + button in the ModelBuilder properties to create a Group item and drag shapes inside it to move them into the group.

2. To rename shapes or groups, click on them in the Scene list and enter a new name.

3. If the Viewer seems too cluttered, you can toggle the visibility of both shapes or groups by clicking in the Scene list.

   Note that when an item is hidden, it doesn't appear in the Viewer or renders.

4. If you have connected geometry to the geo input but don't currently need it as a reference, you can also uncheck Pass Through Geo. This tells ModelBuilder to not output that geometry.

   Note that when Pass Through Geo is enabled, the geometry appears both in the Viewer and renders.

5. By default, the src image is displayed in the 3D Viewer whenever the Viewer is locked to the input camera. If you don't want this, uncheck Show Source Image.

   ![Show Source Image enabled. Show Source Image disabled.](image)

6. Use the display and render controls to select how you want your shapes to appear in the Viewer and renders. These controls affect all the existing shapes.

   For example, you may want to set display to wireframe initially in order to better see what you are doing when positioning your shapes, but change it to textured to review the final result.

   For more information on the available options, see Object Display Properties.

7. If you set display to textured or textured+wireframe, you can also use the Texture control to select the frame used to texture the shapes:

   • Current frame - Project the current frame onto the shape.
• **Locked frame** - Project the frame specified in the field on the right onto the shape. This can help you line up the shape against your source footage.

• **Frame difference** - Subtract the frame specified in the field on the right from the current frame and project the resulting frame onto the shape. This can help you line up the shape against your source footage.

8. If you want to disable shape selection in the Viewer, uncheck **selectable**. This only has an effect when the selection mode menu is set to **Select nodes** in the ModelBuilder toolbar.

9. If you don't want the shapes to cast or receive shadows, uncheck **cast shadow** or **receive shadow**.

   For more information on shadows, see **Casting Shadows**.

10. Proceed to **Positioning Shapes** below.
Positioning Shapes

To position shapes:

1. If the **Align Mode** isn't already active, click   in the ModelBuilder toolbar to activate it.
2. The Viewer should also automatically be locked to your input Camera node. If this isn't the case, select the Camera from the dropdown menu in the top right corner of the Viewer and click   . The positioning handles only appear when the Viewer is locked to the input camera.
3. Use the **Align** dropdown menu on the top of the Viewer to choose whether you want to position the shape by transforming the entire shape or individual vertices:
   - **Object transform** - ModelBuilder tries to transform the whole object. This guarantees that flat polygons stay flat and the angle between polygons doesn't change, but it's less flexible because you're always changing the whole object at once.

![The original card.](image1) ![The card after transforming the entire object.](image2)

- **Vertex position** - ModelBuilder transforms each vertex separately. While this can be a very convenient way to make the geometry match up exactly, it can also lead to non-flat polygons and other problems.

![The original card.](image3) ![The card after transforming a vertex position.](image4)
4. Move a few vertices so that they match the shape of the object you want to model. For example, if you are modeling a building using a cube, drag the corners of the cube so that they line up with the corners of the building. When you click on a vertex, you'll notice a zoom window in the top left of the Viewer. This allows you to accurately position vertices without zooming the entire Viewer. To adjust the size and magnification factor for the zoom window, use the **Zoom** dropdown menus on top of the Viewer.

As you do this, ModelBuilder transforms the shape to fit your adjusted vertex positions. It also sets keys on the 2D positions of the vertices. Any vertices that have keys set on them turn purple in the Viewer.

5. Go to a different frame and move one of the blue vertices to match its new location in the **src** footage. When you do so, a purple line appears. The correct location for the vertex on the current frame should lie somewhere on this line; however, you can drag the vertex away from the line if necessary.

Positioning just one vertex is usually enough to get the object position right, but not the rotation or scaling.

6. In order to lock off the rotation and scaling of the object, move a few more vertices to their correct location.

7. Play back through the sequence. The object should stay roughly in the right place relative to the **src** footage. If you see any vertices starting to drift, move them to their correct location.

8. Proceed to **Editing Shapes**.
Editing Shapes

When editing ModelBuilder shapes, there are two types of editing actions:

- **Single-step actions** that happen instantly, such as extruding and merging faces.
- **Multi-step actions** (such as beveling and subdividing faces) that have extra parameters you can set.

All multi-step actions behave in the same way:

- When you start the action, you can see a preview in the Viewer even though the action has not been applied yet. A colored border appears around the Viewer window to indicate that the action is in progress.

![Viewer window with a colored border around it](image)

**Tip:** You can change the color of the border in the Preferences. Press `Shift+S` on the Node Graph to launch the Preferences dialog, go to the Appearance tab, and use the Highlight button to select a new color. Note that the brightness of the color is adjusted to prevent it from clashing with the Viewer background.

- The parameters for the action appear at the top of the Viewer. As you adjust them, the preview in the Viewer is updated.
- To cancel the action, you can press `Esc` on the Viewer.
- To accept the current parameter values and complete the action, you can press `Return` (or change the selection mode, start a new action, press the `+, -`, or `duplicate` buttons, add a new shape, change into Align mode, or close the ModelBuilder properties).

To edit your shapes:

1. You can edit vertices, edges, faces, and entire objects. For more information, see:
   - Editing Vertices

![Viewer window with a colored border around it](image)
• Editing Edges and Edge Loops
• Editing Faces, and
• Editing Objects.

2. When you're happy with your edits, proceed to Exporting Shapes to Separate Geometry Nodes.
Editing Vertices

To edit vertices:

1. Activate **Edit Mode** by selecting 🔄 from the ModelBuilder toolbar.

2. Set the selection mode menu to **Select vertices** 📣 in the ModelBuilder toolbar and select one or more vertices on the object.

3. Edit your selection as necessary:
   - To translate, rotate, or scale the selected vertices, drag the transform handles that appear on them. To move the pivot point for the transform handles, press **Ctrl/Cmd+Alt** while dragging on the center of the transformation overlay.

   **Tip:** If necessary, you can also use the controls at the top of the Viewer to set the initial position and alignment of the transform handles (that is, the position used whenever you change the selection). For more information, see *Setting the Initial Transform Action Center*.

   - To carve the selected vertices, right-click on them and select **carve**. A colored border appears around the Viewer window to indicate that the action is in progress. Click on an edge or vertex to begin a carve. All of the surrounding faces are highlighted in red. Click anywhere inside a highlighted face, or on an edge or vertex of a highlighted face, to carve an edge between the previous vertex and the place you just clicked.

     ![A highlighted face.](image1.png) ![A new edge carved inside the highlighted face.](image2.png)

     To carve out a freehand edge, hold down **Ctrl/Cmd+Shift** and drag.

   - To extrude the selected vertices, select **extrude** from the right-click menu and drag. This stretches the selected vertices into three-dimensional polygons. ModelBuilder creates faces to complete the object, for example a rectangular pyramid from a vertex.
• To bevel the selected vertices, select **bevel** from the right-click menu.

A colored border appears around the Viewer window to indicate that the action is in progress.

At the top of the Viewer, use **relative inset** to define how far back along the surrounding edges to start the bevel from: **0.0** means no distance and **1.0** means at the opposite end of the edge. The default is **0.1**, meaning the beveling starts 1/10th of the way back along each edge.

Set **round level** to the number of times to repeat the initial bevel, effectively rounding the edges. A value of **0** just does the initial bevel, a value of **1** bevels the output of the initial bevel; a value of **2** bevels the bevel of the initial bevel, and so on.

When you’re happy with the values you’ve entered, press **Return** to apply the bevel.

Beveling is like extruding the selected vertices, except that the resulting polygons have smooth edges and corners. You may want to use beveling to add realism to your model, as real-world objects rarely have perfectly sharp corners.

• To delete the selected vertices, select **delete vertices** from the right-click menu (or press **Delete**).

**Note:** You can only delete simple vertices that have no more than two edges connected to them.
Editing Edges and Edge Loops

To edit edges and edge loops:

1. Activate **Edit Mode** by selecting 🔄 from the ModelBuilder toolbar.

2. Set the selection mode menu to either **Select edges** 🔄 or **Select edge loops** 🔄 in the ModelBuilder toolbar and select one or more edges or edge loops on the object.

   **Tip:** An edge loop is a string of connected edges across the model. Often, the last edge meets again with the first edge, forming a loop. In ModelBuilder, the loop follows the middle edge in every intersection that has an even number of edges. The loop ends when it encounters an intersection with an odd number of edges.

   Edge loops can help you produce more natural deformations for organic models that are animated. They work best when you've got a mesh that's mostly made up of four-sided polygons, known as *quads*.

3. Edit your selection as necessary:
   • To translate, rotate, or scale the selected edges or edge loops, drag the transform handles that appear on them. To move the pivot point for the transform handles, press **Ctrl/Cmd+Alt** while dragging on the center of the transformation overlay.

Selected edges.  Translating the selected edges.
Tip: If necessary, you can also use the controls at the top of the Viewer to set the initial position and alignment of the transform handles (that is, the position used whenever you change the selection). For more information, see Setting the Initial Transform Action Center.

• To carve the selected edges, right-click on them and select **carve**. A colored border appears around the Viewer window to indicate that the action is in progress. Click on an edge or vertex to begin a carve. All of the surrounding faces are highlighted in red. Click anywhere inside a highlighted face, or on an edge or vertex of a highlighted face, to carve an edge between the previous vertex and the place you just clicked.

![Highlighted faces. Carving a new edge inside the highlighted faces.](image)

To carve out a freehand edge, hold down **Ctrl/Cmd+Shift** and drag.

• To extend your current edge selection along loops, select **select edge loop** from the right-click menu. This is the same as activating **Select edge loops** and clicking on an edge.

![Original selection. Select edge loop.](image)

• To extend your current selection to form a ring around the face or the entire shape, choose **select edge ring** from the right-click menu.

![Original selection. Select edge ring.](image)
If the starting edge is part of the boundary for an entire shape, this selects all edges along that boundary.

![Original selection. Select edge ring.](image)

Otherwise, it selects all edges around the face that the starting edge belongs to.

![Original selection. Select edge ring.](image)

- To extrude the selected edges or edge loops, select **extrude** from the right-click menu and drag. This stretches the selected edges or edge loops into a three-dimensional polygon.

![A selected edge. Extruding the selected edge.](image)

- To bevel the selected edges or edge loops, select **bevel** from the right-click menu. A colored border appears around the Viewer window to indicate that the action is in progress. At the top of the Viewer, use **relative inset** to define how far back along the surrounding edges to start the bevel from: **0.0** means no distance and **1.0** means at the opposite end of the edge. The default is **0.1**, meaning the beveling starts 1/10th of the way back along each edge.

Set **round level** to the number of times to repeat the initial bevel, effectively rounding the edges. A value of **0** just does the initial bevel, a value of **1** bevels the output of the initial bevel; a value of **2** bevels the bevel of the initial bevel, and so on.

When you're happy with the values you've entered, press **Return** to apply the bevel.
Beveling is like extruding the selected edges, except that the resulting polygons have smooth edges and corners. You may want to use beveling to add realism to your model, as real-world objects rarely have perfectly sharp corners.

- To divide the selected edges or edge loops into segments of equal length, select **subdivide**. A colored border appears around the Viewer window to indicate that the action is in progress. Use the **segments** control on top of the Viewer to set the number of segments, and press **Return** to complete the action.

- To delete the selected edges or edge loops, select **delete edges** from the right-click menu (or press **Delete**).
Editing Faces

To edit faces:

1. Activate **Edit Mode** by selecting 🔄 from the ModelBuilder toolbar.

2. Set the selection mode menu to **Select faces** 📋 in the ModelBuilder toolbar and select one or more faces on the object.

3. Edit your selection as necessary:
   - To translate, rotate, or scale the selected faces, drag the transform handles that appear on them. To move the pivot point for the transform handles, press Ctrl/Cmd+Alt while dragging on the center of the transformation overlay.

   **Tip:** If necessary, you can also use the controls at the top of the Viewer to set the initial position and alignment of the transform handles (that is, the position used whenever you change the selection). For more information, see Setting the Initial Transform Action Center.

   - To carve the selected faces, right-click on them and select **carve**. A colored border appears around the Viewer window to indicate that the action is in progress. Click on an edge or vertex to begin a carve. All of the surrounding faces are highlighted in red. Click anywhere inside a highlighted face, or on an edge or vertex of a highlighted face, to carve an edge between the previous vertex and the place you just clicked.

     To carve out a freehand edge, hold down Ctrl/Cmd+Shift and drag.

     - To extend your current selection along loops, choose **select face loop** from the right-click menu. A face loop is a string of connected faces across the model. Often, the last face meets again with the first face, forming a loop.

     To create a face loop, you need to select at least two neighboring faces, so that ModelBuilder knows which direction you want to expand the selection in. If the selected faces are above and
below each other, ModelBuilder expands the selection vertically. If the selected faces are to the left and right of each other, ModelBuilder expands the selection horizontally. If you have multiple groups of selected faces, they all get expanded out.

Face loops only work with four-sided polygons, known as quads. If a face loop reaches a face with any other number of sides, the loop ends there.

- To extrude the selected faces, select extrude from the right-click menu (or press Return) and drag. This stretches the selected faces into a three-dimensional polygon. ModelBuilder creates faces to complete the object.

If you select extrude from the right-click menu again and extrude the same faces, ModelBuilder creates another three-dimensional polygon next to your first one.

- To merge adjacent faces, select merge from the right-click menu.

- To bevel the selected faces, select bevel from the right-click menu.

A colored border appears around the Viewer window to indicate that the action is in progress. At the top of the Viewer, use relative inset to define how far back along the surrounding edges to start the bevel from: 0.0 means no distance and 1.0 means at the opposite end of the edge. The default is 0.1, meaning the beveling starts 1/10th of the way back along each edge.

Set round level to the number of times to repeat the initial bevel, effectively rounding the edges. A value of 0 just does the initial bevel, a value of 1 bevels the output of the initial bevel; a value of 2 bevels the bevel of the initial bevel, and so on.

When you’re happy with the values you’ve entered, press Return to apply the bevel.
Beveling is like extruding the selected faces, except that the resulting polygons have smooth edges and corners. You may want to use beveling to add realism to your model, as real-world objects rarely have perfectly sharp corners.

- To invert the selected face’s normals, select **flip face normals** from the right-click menu. This can be useful, for example, if editing operations (such as a sequence of extrusions) have resulted in normals that point the wrong way.

**Tip:** To see the normals, press **S** on the Viewer to display the Viewer Settings, go to the **3D** tab, and activate the **show primitive normals** button.

- To turn the selected face into a triangle fan, select **tessellate > triangle fan** from the right-click menu. This adds a new vertex at the center of the face and joins each of the vertices around the perimeter of the face to it. It can be useful if you want to quickly add more detail to your mesh.
a tessellated triangle fan.

- To delete the selected faces, select **delete faces** from the right-click menu (or press **Delete**).
Editing Objects

To edit objects:

1. Activate **Edit Mode** by selecting from the ModelBuilder toolbar.

2. Click the **Select object** button in the ModelBuilder toolbar and select an object.

3. Edit the object as necessary:
   - To translate, rotate, or scale the selected object, drag the transform handles that appear on it. To move the pivot point for the transform handles, press Ctrl/Cmd+Alt while dragging on the center of the transformation overlay.

   **Tip:** If necessary, you can also use the controls at the top of the Viewer to set the initial position and alignment of the transform handles (that is, the position used whenever you change the selection). For more information, see *Setting the Initial Transform Action Center*.

- • To carve the selected objects, right-click on them and select **carve**. A colored border appears around the Viewer window to indicate that the action is in progress. Click on an edge or vertex to begin a carve. All of the surrounding faces are highlighted in red. Click anywhere inside a highlighted face, or on an edge or vertex of a highlighted face, to carve an edge between the previous vertex and the place you just clicked.

  ![Highlighted faces.](image1.png) ![Carving a new edge inside the highlighted faces.](image2.png)

  To carve out a freehand edge, hold down **Ctrl/Cmd+Shift** and drag.

- • To mirror the selected object along its x axis, right-click on the object and select **mirror x**.

- • To mirror the selected object along its y axis, right-click on the object and select **mirror y**.

- • To mirror the selected object along its z axis, right-click on the object and select **mirror z**.

- • To invert the selected object's normals, right-click on the object and select **flip face normals**. This can be useful, for example, if your camera is located inside the selected object.
**Tip:** To see the normals, press S on the Viewer to display the Viewer Settings, go to the 3D tab, and activate the *show primitive normals* button.

Before **flip face normals**.  After **flip face normals**.

• To delete the selected object, select **delete objects** from the right-click menu (or press **Delete**).
Setting the Initial Transform Action Center

When you translate, rotate, or scale vertices, edges, faces, or objects, you can change the initial position from where the action originates (that is, where the transform handles appear whenever you change the selection). To do so:

1. Make sure **Edit Mode** is active in the ModelBuilder toolbar.
2. To change the default position of the transform handles relative to your current selection, set **handle pos** at the top of the Viewer to:
   - **selection average** - The transform handles appear at the average location of all selected items.
   - **object center** - The transform handles appear at the center of the bounding box for the object that the selected items are part of.
   - **object surface** - The transform handles are snapped to the surface of the object, as close as possible to the average position in the selection. This is the default behavior.
Note: The object center and object surface options only apply when you have selected items on a single object. If you have selected items on more than one object, selection average is always used.

3. To change the orientation of the transform handles relative to your current selection, set handle align at the top of the Viewer to:
   • world axes - The transform handles are oriented so that the x, y, and z arrows align with the world x, y, and z axes.
   • object axes - The transform handles are oriented so that the x, y, and z arrows align with the object's local x, y, and z axes.
• **surface normal** - The transform handles are oriented so that the y arrow aligns with the object's surface normal and the x and z axes are at an arbitrary orientation. This is the default behavior.

![Image of transform handles aligning with surface normal](image)

**Note:** The **object axes** and **surface normal** options only apply when you have selected items on a single object. If you have selected items on more than one object, **world axes** is always used.

4. To prevent the position and alignment of the transform handles from being changed when you change your selection, enable **locked** at the top of the Viewer.

This makes it easier to rotate a face relative to one of its edges, for example: instead of having to **Ctrl/Cmd+Alt+drag** the transform handles into the right place by eye, you can set **handle pos** to **selection average**, select the edge, and enable **locked**. When you then select the face, the transform handles stay where they are.

![Images showing edge selection and face rotation](images)

- Selecting an edge and enabling **locked**.
- Rotating a face around the previously selected edge.
Applying Textures

There are a couple of ways to texture your ModelBuilder models in Nuke:

• If your 3D model closely matches the original 2D footage, you can project the 2D footage onto the geometry. See Projecting Textures onto Your Shapes.

• If you have added new objects over the top of your 2D footage (for example, added a new window to a building or extra bins alongside a street), you can't project your 2D footage over those objects because they were never in the original footage. Instead, you need to supply a texture from somewhere else and map it over the surface of your 3D object. See UV Unwrapping.
Texturing an added cube object separately.
Projecting Textures onto Your Shapes

If your ModelBuilder model closely matches the original 2D footage, you can simply project the 2D footage onto the geometry. To do so:

1. At the bottom of the ModelBuilder properties panel, set the bake menu to **Projection** and click **Bake**.
   This creates a projection at the current texture frame: a Project3D node with a FrameHold node set to lock the input image and camera to the texture frame. The Project3D node is also connected to the **mat** input of an ApplyMaterial node.

2. To apply the projection to the geometry, connect the ModelBuilder node (or a geometry node created by baking **Selected geometry**) to the ApplyMaterial node's unnamed input and view the result.
**Tip:** To texture your geometry on more than one frame, you can bake Projection on several frames and use Roto nodes to mask things out before combining the Project3D nodes using a MergeMat node. Then, connect the MergeMat node to the ApplyMaterial node's mat input and ModelBuilder to its unnamed input. Connect a Viewer to the output of ApplyMaterial and view the results.

**Tip:** The options in the bake menu are implemented in Python, and you can also use Python to add your own entries to the menu. To see how the built-in options have been implemented, have a look at the modelbuilder.py module in the nukescipts folder of your installation (for more information on the location of this file, see Viewing More Examples). Then, to create your own menu options, edit the modelbuilder.py file and use the populateBakeMenu function to add entries where it says "# Add your own entries here, if desired".
UV Unwrapping

If you've used ModelBuilder to add new objects over the top of your 2D footage (for example, added a new window to a building or extra bins alongside a street), you can't project your 2D footage over those objects because they were never in the original footage. Instead, you need to supply a texture from somewhere else and map it over the surface of your 3D object.

An extra cube has been added to the scene. It cannot be textured using the original footage. Here, the cube has been textured separately.

To map a texture image over the surface of your 3D object, you first need to flatten your 3D object out into 2D space through a process called UV unwrapping. This generates UV coordinates for each vertex in the object, allowing you to map points on the surface of your 3D object to pixels in your texture image.

A 3D cube flattened out into 2D space.
Tip: UVs are simply 2D coordinates that tell 3D applications how to apply a texture to a model. The letters U and V were chosen because X, Y, and Z were already used to denote the axes of objects in 3D space. The U coordinate represents the horizontal axis of the 2D texture, and the V coordinate the vertical axis.

Quick Start

The process of UV unwrapping is roughly the following:

1. Define where to cut the model in order to flatten it out into 2D space. See Creating Seams on Your Model.
2. Unwrap the model and preview the results. See Unwrapping Your Model and Previewing Its UVs.
3. Edit the generated UVs as necessary. See Editing UVs.
4. Apply a texture to your model. See Applying Textures.
Creating Seams on Your Model

To prepare your model for UV unwrapping, you need to mark some seams. The seams tell ModelBuilder where it's allowed to cut the model in order to flatten it out into 2D space.

**Tip:** If you don't have enough seams marked, the unwrapping will be very poor with lots of overlapping faces as well as stretching and distortion. On the other hand, too many seams can make it difficult to texture the object, as the gaps in the seams can easily become noticeable on the textured object. If possible, it's a good idea to hide seams in areas where they will not be seen.

To Create Seams

1. If you have created several objects using the same ModelBuilder node, click ▪ in the node properties to hide all the other objects except the one you want to generate UVs for.

2. In the ModelBuilder toolbar, click ▪ to activate UV mode.

3. Repeat one or more of the following until you've got a good set of seams:
   - To mark an edge as a seam, right-click on the Viewer and choose **edge selection**. Select an edge on your model, right-click, and choose **mark as seam**.

   ![Image of marking an edge as a seam]

   • To mark an edge loop as a seam, right-click on the Viewer and choose **edge loop selection**. Select an edge loop on your model, right-click, and choose **mark as seam**.

   ![Image of marking an edge loop as a seam]
• To mark all edges around a face as seams, right-click on the Viewer and choose face selection. Select a face on your model, right-click, and choose mark as seam.

![Image of face selection and mark as seam](image)

**Tip:** You can also select a face loop and mark all edges around it as seams. To do so, select at least two faces on your model, right-click, and choose select face loop. Then, right-click on the Viewer again and choose mark as seams.

• To remove an edge, edge loop, or face from the set of seams, select it in the Viewer, right-click, and choose unmark as seam.

• To clear all seams, right-click on the object and choose clear all seams. Any edges that have been marked as seams are displayed in red in the Viewer.

4. Proceed to Unwrapping Your Model and Previewing Its UVs below.

**Tip:** The easiest way to create a good set of seams for buildings (or any other roughly cubic or cylindrical object) is to 1) select the top faces and mark them as seams, 2) select the bottom faces and mark them as seams, and 3) select edges along the side of the model that connect the top to the bottom and mark them as seams.

Unwrapping Your Model and Previewing Its UVs

Once you're happy with your set of seams, you are ready to unwrap the model.

To Unwrap a Model and Preview Its UVs

1. Right-click on the Viewer and choose object selection.
2. Right-click on your model and choose unwrap.
   A colored border appears around the Viewer window to indicate that the action is in progress.
3. Press Return to complete the unwrap.
ModelBuilder generates UV values for each of the vertices on your model. The unwrapped model appears in the UV preview window in the top left of the Viewer.

4. To display a 2D texture image as a background in the UV preview window, connect the image to ModelBuilder’s `tex` input and make sure the `Preview` dropdown in the ModelBuilder properties (or on top of the Viewer) is set to `tex input`. To also see the `tex` input on the model in the 3D Viewer, set `display` to either `textured` or `textured+wireframe` in the ModelBuilder properties.

5. To zoom in or out of the UV preview window, press `Ctrl/Cmd+Alt` and drag.
6. To pan the UV preview window, press `Ctrl/Cmd` and drag.
7. To change the size of the UV preview window, click and drag on its lower right corner.
8. To hide the UV preview window, disable `uv window` at the top of the Viewer.
9. Proceed to Editing UVs below.

**Tip:** If you are not happy with the generated UVs, you can right-click on your model and select `clear existing uvs`. If necessary, you can also change the seams and unwrap your model again. Each time you do, ModelBuilder replaces the UVs you had with the newly generated UVs.
Editing UVs

You can tweak the controls at the top of the Viewer or edit the UVs manually in the UV preview window.

1. Adjust the following controls at the top of the Viewer:

   • **iterations** - The number of times the unwrapper applies its rules about how to move the UV coordinates towards good locations. This gives you a speed vs. quality trade-off: more iterations give a better result but take longer to unwrap. Note that the results can only be improved up to a point, and where that point is depends on the complexity of your model.

   • **threshold** - The unwrapper stops before reaching the maximum number of iterations if it sees that the improvement from iteration to iteration is negligible. The threshold control tells it how much change is considered negligible. If the amount of change between iterations is below the threshold, the unwrapper stops. This is another way to trade off quality for speed. You generally use this if you want a higher quality result: if you've increased iterations but the unwrapper is stopping before reaching the maximum, you can lower the threshold to make it keep running.

   • **separation** - The number of pixels to leave between each patch in the unwrapping. If you don't have enough pixels between your patches, you can get color bleeding through from neighboring patches when you use the texture. A higher value means more widely spaced patches, but also more wasted space in your texture image.

   As you edit these controls, the UV preview window updates to allow you to see the effect of your changes.

2. If necessary, you can also edit the UVs manually in the UV preview window:

   • To edit a single vertex, drag it to a new location.

   • To edit several vertices together, select them and use the transform jack that appears to translate, rotate, or scale the selection. The transform jack is the same one used elsewhere in Nuke, so all the controls work the same way. For example, you can click and drag on the handles to scale out from the center, or Ctrl/Cmd+click and drag to scale out from the opposite edge.
3. Proceed to Applying Textures.

Applying Textures

At this point, it may be that you've got several objects in the scene that you still want to project your camera footage onto (see Projecting Textures onto Your Shapes), but there's one object that you want to texture differently. There are two ways to do this:

• You can export your model as a separate geometry node and use an ApplyMaterial node to texture it. This can be a convenient way to work if your geometry is finished and locked off, so you never need to go back and change it. But what you lose with this is the "live" view: if you do need to go back and edit the object, you have to export it again. See Method 1.
• Alternatively, you can add an ApplyMaterial node directly after your ModelBuilder node and tell ApplyMaterial to ignore all the geometry that doesn't match the filter you give it. See Method 2.

Method 1

1. In the ModelBuilder properties, select the object that you want to texture differently.
2. At the bottom of the properties panel, set the bake menu to Selected geometry and click Bake.

ModelBuilder creates a geometry node for the selected object.
3. From the toolbar, select 3D > Shader > ApplyMaterial to create an ApplyMaterial node.
4. Connect the geometry node you created in step 2 to the unnamed input of the ApplyMaterial node.

5. Then, connect your 2D texture image to the **mat** input of the ApplyMaterial node. The ApplyMaterial node applies the texture from the **mat** input onto your 3D geometry object. (To be able to see this in the 3D Viewer, you may have to hide the object in the ModelBuilder properties.)

6. Connect the ApplyMaterial node to your Scene node and use a ScanlineRender node to render all the objects connected to that scene.
Method 2

1. From the toolbar, select **3D > Shader > ApplyMaterial** to create an ApplyMaterial node.

2. Connect the unnamed input of the ApplyMaterial node to your ModelBuilder node and the **mat** input to your 2D texture image.
   By default, ApplyMaterial applies the texture from the **mat** input onto all objects in your ModelBuilder node.

3. To only apply the texture onto a particular object, open the ApplyMaterial properties and set **filter** to **name**. This allows you to tell ApplyMaterial to ignore any geometry that doesn't match the filter.

4. To set the filter, click the **choose** button. In the dialog that opens, select the object you want to apply the texture to and click **OK**.

   **Tip:** You can also **Ctrl/Cmd+click** or **Shift+click** to select multiple objects.
For more information on ApplyMaterial, see Applying a Material Using the ApplyMaterial Node.

5. Connect the ApplyMaterial node to your Scene node and use a ScanlineRender node to render all the objects connected to that scene.
Exporting Shapes to Separate Geometry Nodes

If necessary, you can export shapes on your ModelBuilder model to separate geometry nodes. This allows you to operate on one part of the scene separately from the rest. You can use the geometry nodes in the same way as any other geometry nodes in Nuke.

To Export Shapes to Separate Geometry Nodes:

1. In the ModelBuilder properties, select any shapes you want to export to a separate geometry node.
2. Make sure the bake menu under Export is set to Selected geometry and click Bake. ModelBuilder creates a geometry node for the selected items in the scene.

Tip: The options in the bake menu are implemented in Python, and you can also use Python to add your own entries to the menu. To see how the built-in options have been implemented, have a look at the modelbuilder.py module in the nukescripts folder of your installation (for more information on the location of this file, see Viewing More Examples). Then, to create your own menu options, edit the modelbuilder.py file and use the populateBakeMenu function to add entries where it says "# Add your own entries here, if desired".
Creating 3D Particles

Nuke's Particle node set is a solution for creating particles in a 3D environment. You can create things like smoke, fog, falling snow, explosions, and bubbles - the possibilities are endless. You can use the various Particle nodes for emitting, manipulating, and displaying limitless types of particles in your 3D scene.

Quick Start

Here's a quick overview of the workflow:

1. Create a ParticleEmitter node, and connect it to a Viewer.
2. Connect a source of emission and a particle representation to the `emit` and `particle` inputs of the ParticleEmitter. For more information, see Connecting Particle Nodes.
3. Modify your particles’ lifetime, velocity and other basic properties in the ParticleEmitter properties panel. For more information, Emitting Particles.
4. Connect other particle nodes to the ParticleEmitter's output. See Adjusting the Speed and Direction of Particles, Modifying the Particles' Movement and Adjusting Particle Simulation Settings.
5. If necessary, cache your particle simulation in order to read it back in without the need for recalculation. For more information, see Caching Particles.

Rain created using particles. Snow created using particles
Connecting Particle Nodes

In order to create particles, the minimum setup you need is the ParticleEmitter node. To connect your Particle nodes:

1. Click the **Particles** menu in the Toolbar and select the ParticleEmitter node.
2. Connect it to a Viewer node.
3. Connect a 3D geometry object, or PositionToPoints point cloud, to the **emit** input of the ParticleEmitter. A 3D object from which to emit the particles is optional: if you don't use one, the particles are emitted along the y axis from a point of origin.
4. To specify the appearance of your particles, connect an image or a geometry in the **particle** input of the ParticleEmitter. This image or geometry is then multiplied and used as representations of each of your particles. If you want to use more than one representation, connect further image or geometries to the other numbered **particle** inputs. ParticleEmitter picks one of these at random for each particle.
5. If you have another particle system that you'd like to connect to your new ParticleEmitter, you can connect it to the **merge** input. You can also merge particle systems with the ParticleMerge node (see Merging Particle Streams).
6. Now you're ready to start modifying your particles to look the way you want. To do this, you can pick any of the various particle nodes in the Toolbar's **Particle** menu and connect them to the ParticleEmitter node's output, or to other particle nodes. Each of them has its own effect on the particles and a set of controls you can make adjustments with.

![Particles emitted from Sphere geometry.](image)
Emitting Particles

The ParticleEmitter node is the first and only required node in a minimum particle setup. Once you've created a ParticleEmitter, connected it to a Viewer and clicked **play** on the timeline, you'll see the default set of particles emitting (from a 3D geometry or point cloud, if you've connected one). You can then adjust the ParticleEmitter controls to change the way the particles appear:

1. Set the **channels** in which particles are emitted. Channels **a** and **b** are arbitrary names for channels which are useful if you want different particle force nodes to have an effect on separate channels.
2. Use the **start at** field to pre-roll or delay the point at which the first particles are emitted. For example, particles could imitate snow that has fallen already, instead of displaying the first flakes falling down, by using a negative **start at** frame.
3. Select the emission order and rate for the particles. Set:
   - **emit from** - select **points**, **edges**, **faces** or **bbox** (bounding box) to specify from which part of the 3D object, or point cloud, particles are emitted.

   - **emit order** - select the order that the particles are emitted:
     - **randomly** - produces a random emission order.
     - **uniformly** - emits all particles at the same time.
• **in order** - emit particles as a multiple of the **emission rate**. For example, an **emission rate** of 2 could emit from two points, edges, or faces at a time.

![](image1.png)  
Particles emitted in order.  Particles emitted uniformly.

• **randomize type** - select whether or not particles are emitted randomly. Select **no random direction** for a specific direction based on the emit object's normals, **randomized direction** for a randomly selected initial direction, or **randomize outwards** for a randomly selected direction (dependent on the **emit from** selection):

• **bbox** - particles move randomly away from the center.

• **points, edges, and faces** - particles move randomly away from the origin, but at no more than 90 degrees from the nearest normal.

• **emission rate** - select the number of particles emitted per frame. This is an exact number, and it is affected by the **rate channel**. If the channel is at a varying value, the emission rate also increases or decreases.

• **only emit from selected points** - specify whether the particles are emitted from the object, or selected vertices of the object using a GeoSelect node. You can also emit from selected points in a PositionToPoints point cloud using any normals information present.

  See **3D Selection Tools** for more information.

![](image2.png)  
Vertices selected using an upstream GeoSelect node.  Emitting particles only from the selected vertices.

• **rate variation** - specify the range of variation for emitting particles. If you set this to zero, particles are emitted at a constant even level.

• **rate channel** - select a specific channel to which you want to emit particles. Unchecking this is the same as selecting none, and in that case particles are emitted to all the channels. For
example, if you are emitting from a Card node that has a Ramp texture, it emits particles from the light parts of the ramp at a higher rate (values closer to 1) than from the dark parts (values closer to 0).

4. Set the particles’ color and channels. Enter:

- **color** - select a color for your particles. Use this if you’re not using the particle input.
- **color from texture** - check to tint the particles with the colors from the geometry’s texture. If you leave this unchecked, the particles get color only from their own texture.
- **channels** - select the channels you want to apply particles to. By default, particles are emitted to channel a. Channels a and b are arbitrary names for channels which are useful if you want different ParticleEmitter nodes or other particle force nodes to have an effect on separate channels. An example of this might be if you want two ParticleEmitter nodes emitting particles, one to a channel, the other to b channel, so further down in your particle stream you can apply an effect to a specific set of particles.

5. Select how long you want the particles to exist. Set:

- **max lifetime** - specify the maximum life span for a particle in frames.
- **max lifetime range** - adjust the range within which your particles’ lifetime varies. If you set this to 0, all particles have the same lifetime.
- **lifetime channel** - select a channel of the input geometry texture that you want to use to modulate the lifetime. For example, if you are emitting particles from a Card node with a Ramp texture in the lifetime channel, particles emitted from the lighter parts of the ramp (values closer to 1) would have a lifetime value closer to that set in the **max lifetime** than particles emitted from the dark parts (values closer to 0).
- **halflife** - select the number of frames over which the number of the particles emitted is halved.

6. Adjust the velocity and rotation for your particles. Set:

- **velocity** - specify the initial speed at which you want your particles to move.
- **velocity range** - adjust the range within which you want your particles’ velocity to vary. If you set this to 0, the velocity doesn’t vary between particles.
- **velocity channel** - select a channel of the input geometry texture that you want to use to modulate the velocity. For example, if you are emitting from a Card node that has a Ramp texture, it emits particles from the light parts of the ramp at a higher velocity (values closer to 1) than from the dark parts (values closer to 0).
- **rotation velocity** - adjust the initial speed at which each particle rotates around its individual Y-axis. The Y-axis points to the direction the particles were initially emitted, and then stays unchanged (unless you use the ParticleMotionAlign node to change its direction). Rotational velocity is most useful when you’re emitting particles from a geometry object.
- **rotation velocity range** - adjust the scale of a random variation in the **rotation velocity** value. Value of 0 means the rotation velocity value is as set, value of 1 means it's very random.
• **rotation velocity channel** - Select a channel of the input geometry texture that you want to use to modulate the speed of rotation. For example, if you are emitting from a Card which has a Ramp texture, the rotation velocity of the particles emitted from the light part of the Ramp (values closer to 1) is greater than that of those emitted from the dark parts (values closer to 0).

• **transfer velocity** - adjust this to transfer the velocity of the initial emitter to the particles. If you set this to 0, no velocity is transferred to the spawned particles. At value 1, full velocity is transferred from the originating particle.

• **transfer window** - adjust the time, in frames, to look forward or backward in order to determine the velocity that should be transferred to the particles.

7. Modify the size and mass of the particles. Set:

• **size** - specify the initial size of each particle.

• **size range** - specify the range within which your particle size varies.

• **size channel** - select a channel of the geometry texture that you want to use to modulate the size of the particles. For example, if you are emitting particles from a Card node which has a Ramp texture in the size channel, the size of the particles emitted from the lighter part of the Ramp (values closer to 1) is greater than that of those emitted from the darker parts (values closer to 0).

• **mass** - adjust the initial mass of each of your particles. The mass of the particles only becomes relevant when you apply a force to your particles, such as one generated by the ParticleDirectionalForce node.

• **mass range** - adjust to produce a random variation in the mass value. Simply put, 0 value means the particles' mass is the value specified by the mass control, whereas a value of 1 means it's very random.

• **mass channel** - select a channel of the input geometry texture that you want to use to modulate the mass. For example, if you are emitting from a Card which has a Ramp texture in the mass channel, the particles from the light part of the Ramp (values closer to 1) has a higher mass value than from the dark parts (values closer to 0). With this control, you can emit particles with different masses from different areas based on the input geometry's texture.

• **spread** - adjust the extent to which you want your particles to spread in different directions during their lifetime. By default, this forms a cone around the direction of emission. If you set this to zero, each particle has a straight trajectory.

8. Modify the way your particles are affected by the ParticleEmitter's inputs. Adjust:

• **input order** - if you're using more than one particle input, you can select which particle input Nuke should select when creating particles. Select **Randomly** to pick one of the inputs randomly, or **in order** to rotate the inputs in numerical order.

• **start at** - select which frame of the particle input should be the representation of each new particle. Select **first** to pick the first frame of the particle input for each new particle. Select **in order** to pick consecutive frames from the input for each new particle. Select **current** to pick
the frame where the particle was emitted. Select random to pick a random input frame for each new particle

- **limit to range** - limit the particle output to the representation's frame rate, looping the frame range when in order or current is selected.
- **advance** - use this to determine if a particle should animate after being emitted. Select constant to keep the same representation throughout the particle's lifetime. Select in steps - to animate the representation frame by frame. Select randomly to animate the representation one random frame after another.

9. Vary the results of your range controls with the random seed field. Enter the random number used in the range controls (such as max lifetime range) to achieve slightly different effects.

**Tip:** When you set large values for any of the particle controls it might take Nuke a few moments to calculate the result. In such a case a progress bar appears with a Cancel button you can use to cancel the calculation if necessary.
Spawning Particles with ParticleSpawn

If you're looking to have your existing particles emit even more particles, you should turn to ParticleSpawn.

1. Connect the ParticleSpawn node to your particle stream (the ParticleEmitter output, for example). All the particles emitted now start spawning more particles.

2. Adjust the ParticleSpawn controls. Most of the ParticleSpawn controls are identical to those in the ParticleEmitter node (see Emitting Particles), with only a few exceptions:
   - **transfer velocity** - adjust this to transfer the velocity of the initial emitter to the particles. If you set this to 0, no velocity is transferred to the spawned particles. At value 1, full velocity is transferred from the originating particle.
   - **conservation of mass** - check if you want the mass of spawned particles to be removed from the mass of the original particle. If the mass of a particle is zero at the end of a frame, it gets deleted.
   - **conservation of momentum** - check this to subtract the momentum of the spawned particles from the original particle, in correspondence with Newton’s third law of motion.
   - **align velocity to direction of motion** - check to align velocity with the direction of the particles’ motion.
   - **inherit color** - check to take the particle color from the originating particle. Otherwise the color is determined by the color control.

![Spawned particles.](image)
Adjusting the Speed and Direction of Particles

Applying Gravity to Particles

When applying gravity to particles, as opposed to our familiar gravity, Nuke doesn't restrict you to a certain direction but works in any or all of the x, y and z directions. You can add gravity either by:

- Using the ParticleDirectionalForce to apply a directional force. Just connect it to your particle stream, and adjust the strength of the force in the x, y, and z directions by entering x, y, and z values in the strength fields.

OR

- Using the ParticleGravity node. When you connect the ParticleGravity node to your particle stream, an arrow appears in the Viewer, which you can then use to determine the direction and velocity of the gravity. The bigger and longer the arrow, the stronger the effect. Instead of adjusting the arrow, you can also use the controls in the properties panel:
  - from - enter the point of origin for the gravity effect on the x, y, and z axis. This determines from which direction the force appears to come, indicated by the base of the arrow in the Viewer.
  - to - enter direction for the gravity effect on the x, y, and z axis. This is indicated by the point of the arrow in the Viewer.

Aligning Particles

To align your particles’ motion, direction, and orientation, you can use two nodes:

- You can add the ParticleMotionAlign node in your particle stream to realign all the particles along their direction of motion. This is useful if your particles seem too rigid in their movement.
- Add the ParticleLookAt node to determine a 3D point that all the particles are looking toward. To specify this point, adjust the position control. The x, y and z coordinates specify the point that the particles are looking at.

Controlling Particle Speeds

The ParticleSpeedLimit node limits the particles to a specified minimum and maximum speed.
1. Connect it to your particle stream.
2. In the properties panel, adjust:
   - **minimum** - the minimum speed at which each particle can travel.
   - **maximum** - the maximum speed at which each particle can travel.

### Attracting Particles to a Specific Point

With the ParticlePointForce, you can attract particles to or repel them from a certain point in the 3D space.

1. Connect the node to your particle stream.
2. Adjust the ParticlePointForce controls:
   - **strength** - set the strength of the force attracting or repelling particles. Negative values cause attraction, positive values repulsion.
   - **falloff** - choose how quickly strength of the attraction falls off with respect to the distance by selecting the type of falloff, **none** for no falloff, **inverse** for inverse falloff, or **inverse square** for inverse falloff squared.
   - **radius** - set the radius of influence. Outside of this radius, no particles are affected by point force.
   - **position** - set the position of the point that attracts or repels particles. You can use an animated or a still Axis node expression-linked to these fields, or you can just enter a position value manually.
Modifying the Particles’ Movement

Bouncing Particles off Objects

With the ParticleBounce node, you can make your particles bounce off the shape of a 3D object instead of traveling through it. Connect this node to your other particle nodes, and adjust the ParticleBounce controls:

1. To set your particles to bounce off the outside of the bounce object (specified by the object control), select a mode in the external bounce mode dropdown: none to apply no external bounce effect, bounce to bounce the particles, kill to end the life of the particles as they bounce.

2. Select a channel where a particle should be assigned to when a bounce is detected in the new channels dropdown. Setting this to none doesn’t perform any channel assignment.

3. In the bounce field, specify the strength of the external bounce effect, and adjust friction to control the amount of friction for external bounce effect.

4. To set your particles to bounce off the inside of the bounce object (specified by the object control), use the internal bounce mode dropdown:
   - none - to apply no internal bounce effect.
   - bounce - to bounce the particles.
   - kill - to end the life of the particles as they bounce.

5. Select a channel where a particle should be assigned to when a bounce is detected in the new channels dropdown. Setting this to none doesn’t perform any channel assignment.

6. In the bounce field, specify the strength of the internal bounce effect, and adjust friction to control the amount of friction for the internal bounce effect.

7. Select the object you want to use for the bounce effect in the object dropdown:
   - plane, sphere or cylinder - add a Nuke standard primitive to use as the bounce surface for the particle system. You can use these shapes to quickly test your particle effects.
   - input - add custom geometry attached to the geometry input.

8. Use the transform controls to fine-tune your geometry's position to achieve the desired bounce result.
Adding Drag to Particles

With the ParticleDrag node you can apply drag on your particles. This gradually slows them down over time.

1. Connect the ParticleDrag node to your particle stream.
2. Adjust the ParticleDrag controls:
   - **drag** - increase to add a drag effect to your particles, slowing and stopping their movement as they are making distance from the particle center.
   - **rotational drag** - increase to add a drag effect to your particles’ rotation, slowing and stopping their rotation.

![Particles bouncing off a plane.](image)

**Tip:** You can also enter negative values to apply a reverse drag effect and speed the particles up.

Adding Turbulence Motion on Particles

The ParticleTurbulence node applies Perlin noise to the particle movement in the \(x, y\) and/or \(z\) directions.

1. Connect a ParticleTurbulence node to your particle stream.
2. Adjust the ParticleTurbulence controls:
• **strength** - set the strength for the turbulence effect on the \( x \), \( y \) and \( z \) axes.
• **scale** - set the scale of the effect, or the size of the area affected, on the \( x \), \( y \) and \( z \) axes.
• **offset** - set the offset applied to the effect, or the offset of the area affected, on the \( x \), \( y \) and \( z \) axes.

### Adding Spiral Motion to Particles

The ParticleVortex node applies a circular force to the particles and attracts them to an imaginary line, thus creating a whirlpool of particles.

1. Connect the ParticleVortex node to your particle stream.
2. An arrow appears in the Viewer, which you can drag to determine direction and velocity of the vortex effect. The bigger and longer the arrow, the stronger the effect. Alternatively, you can use the **from** and **to** controls in the properties panel.
3. In the ParticleVortex properties panel modify the parallel effect with the **parallel** control. This accelerates the particles in the direction of the imaginary vortex center line. If you set this to 0, no parallel force is applied, and positive and negative values determine the direction of the force. Adjust the **parallel falloff** to choose how quickly strength of the parallel force falls off with respect to the distance by selecting the type of falloff, **none** for no falloff, **inverse** for inverse falloff, or **inverse square** for inverse falloff squared.
4. With the **tangential** slider you can force the particles to circulate the vortex center line. Adjust **tangential falloff** to choose how quickly strength of the tangential force falls off with respect to the distance by selecting the type of falloff, **none** for no falloff, **inverse** for inverse falloff, or **inverse square** for inverse falloff squared.
5. Use the **radial** slider to adjust the force that attracts the particles to (positive values), or repels (negative values), them from the center line. Adjust **radial falloff** to choose how quickly strength of the radial force falls off with respect to the distance by selecting the type of falloff, **none** for no falloff, **inverse** for inverse falloff, or **inverse square** for inverse falloff squared.

**Tip:** If you want to create a helix of particles, you can turn up both the **parallel** and **tangential** values. This creates a particle vortex in a shape of a corkscrew.
Adding a Wind Effect to Particles

With the ParticleWind, you can simulate a wind blowing on your particles.

1. Connect the ParticleWind node to your particle stream. An arrow appears in the Viewer, which you can then use to determine direction and velocity of the wind. The bigger and longer the arrow, the stronger the wind effect. Alternatively, you can use the from and to controls in the properties panel.

2. In the ParticleWind controls, check air resistance to enable a drag effect on the wind.

3. Adjust the drag slider to increase or decrease the simulated air resistance.
Adjusting Controls Common to Several Particle Nodes

Many of the particle nodes share a number of controls, such as rendering and transform controls. The following covers the use of these.

Particle Rendering Controls

The first controls on the properties panels of all the particle nodes have to do with how the particles are output to the Viewer and rendered out. To read more about the display, selectable and render controls, see Object Display Properties.

Particle Transform Controls

Several of the particle nodes have a set of transformation controls in their properties panels. You can use these controls for example to translate, rotate and skew the force that the node applies on your particles. For more information on how transform controls work, see Transforming Geometry, Cameras, and Lights.

Condition and Region Controls

Various particle nodes have controls on the Conditions and Region tabs in the properties panel.

Conditions tab

On the Conditions tab, use the following controls to restrict the way in which the specific node affects your particles:

- **probability** - set the probability that this node affects your particles. If you set this to zero, the node won't affect any particles, and if the value is 1, the node affects every particle.

- **min age** - set this to limit the effect of this node only to particles above this minimum age. The age of the particle is its lifetime normalized between 0 and 1.

- **max age** - set this to limit the effect of this node only to particles below this maximum age. The age of the particle is its lifetime normalized between 0 and 1.

- **random seed** - enter an integer to change the results of generated randomness in your particles. You can achieve slightly different effects by changing this number.
• **channels** - specify which particle channels the effect of this node should be applied to. Channels a and b are arbitrary names for channels which are useful if you want different ParticleEmitter nodes or other particle force nodes to have an effect on separate channels.

**Region tab**

1. Using the **region** control you can select the region which you want to use to confine the particle effect to. For example, if you select a sphere, only particles inside that sphere-shaped region is affected by the particle effect. Select **none** to apply no confining **region**, or the appropriate shape for a region between **sphere**, **box**, **half-space** and **cylinder**.

2. You can also check **invert region** to only affect the particles outside the region specified.

3. Then move on to adjusting your region with the transform controls. For more information, see Transforming Geometry, Cameras, and Lights.
Adjusting Particle Properties Using Curves

With the ParticleCurve, you can apply a curve to particle properties (such as size or mass) to change them over time.

1. Connect the node to your particle node stream.
2. Adjust the curves in the ParticleCurve properties panel. The x axis represents the lifetime of the particles.
   - r - adjust the curve for the red channel.
   - g - adjust the curve for the green channel.
   - b - adjust the curve for the blue channel.
   - a - adjust the curve for the alpha channel.
   - size - adjust the curve for the size of the particles.
   - mass - adjust the curve for the mass of the particles.

A size curve modifying the particles to grow in size toward the end of their lifetime.

**Note:** If you're using an image or a 3D object as your ParticleEmitter’s particle input, the ParticleCurve might not alter the colors of the particles as expected.

3. If you want, you can adjust the curve for your particles’ alpha channel so that each particle fades to invisibility toward the end of its lifetime.
Adjusting Particles Using Expressions

With the ParticleExpression node, you can adjust your particles by setting expressions on their attributes. Using expressions gives you a vast variety of ways of adjusting how your particles behave. You can use a similar expression syntax as you would elsewhere in Nuke, with the exception that some functions that work in normal Nuke expressions aren't available in particle expressions and vice versa.

The main difference between Nuke's Expression node and ParticleExpression is that particle expressions can return a 3D vector instead of just a single floating point number. If a particle expression returns a single number N in a field that expects a vector (such as velocity or acceleration) it is converted into a vector with N for each of its components. For more information about the functions you can use with ParticleExpression, see Particle Expression Functions.

1. Connect the ParticleExpression node to your particle stream.

2. In the ParticleExpression controls you can use four temporary expression fields. With these, you can set up any expression on a particle attribute and then give it a temporary name. This temporary name can then be used in the following fields to refer to the corresponding temporary expression. This can be useful if you need to use a long expression in several fields. By default, the per-particle box is ticked to make the expressions affect each particle individually. Uncheck the box to apply the expression to all particles at once. For more information on expressions, see Expressions.

3. You can also set expressions on a set of attribute fields:
   - **color** - set an expression to edit the color of the particles.
   - **opacity** - set an expression to edit the opacity of the particles.
   - **size** - set an expression to edit the size of the particles.
   - **mass** - set an expression to edit the mass of the particles.
   - **accel** - set an expression to edit the acceleration of the particles.
   - **force** - set an expression to edit the force of the particles.
   - **pos** - set an expression to edit the position of the particles. For example, enter \( \sin (\text{age} \times 10) \times 5 \) to emit particles randomly on a single static line.
   - **vel** - set an expression to edit the velocity of the particles.
   - **onlyonnew** - check this next to each attribute field to make the expression only affect new particles and ignore any existing ones.
### Particle Expression Functions

Here are some functions you can use with the ParticleExpression node:

<table>
<thead>
<tr>
<th>Function</th>
<th>Purpose</th>
<th>Related Functions</th>
</tr>
</thead>
<tbody>
<tr>
<td>abs(f)</td>
<td>Returns the absolute value of f.</td>
<td>See also: fabs.</td>
</tr>
<tr>
<td>acos(f)</td>
<td>Returns the arc cosine of f. The result is the angle in radians whose cosine is f.</td>
<td>See also: cos, cosh, asin, atan2.</td>
</tr>
<tr>
<td>age</td>
<td>The age of the particle, in frames.</td>
<td>-</td>
</tr>
<tr>
<td>asin(f)</td>
<td>Returns the arc sine of an angle. The result is the angle in radians whose sine is f.</td>
<td>-</td>
</tr>
<tr>
<td>atan(f)</td>
<td>Returns the arc tangent of an angle. The result is the angle in radians whose tangent is f. Can be called with one or two arguments; if called with two arguments it's equivalent to atan2.</td>
<td>-</td>
</tr>
<tr>
<td>atan2(x,y)</td>
<td>Returns the principal value of the arc tangent of y/x, using the signs of the two arguments to determine the quadrant of the result.</td>
<td>See also: sin, cos, tan, asin, acos, atan, hypot.</td>
</tr>
<tr>
<td>ceil(f)</td>
<td>The ceiling of f, rounding any fractional part up.</td>
<td>-</td>
</tr>
<tr>
<td>color</td>
<td>The color of the particle. This is a 3D vector value, where x() is the red component, y() is green and z() is blue.</td>
<td>-</td>
</tr>
<tr>
<td>cos(f)</td>
<td>Returns the cosine of angle f. The angle is in radians.</td>
<td>See also: sin, tan, asin, acos, atan, hypot.</td>
</tr>
<tr>
<td>Function</td>
<td>Purpose</td>
<td>Related Functions</td>
</tr>
<tr>
<td>----------</td>
<td>---------</td>
<td>-------------------</td>
</tr>
<tr>
<td>cosh(f)</td>
<td>Returns the hyperbolic cosine of f.</td>
<td>-</td>
</tr>
<tr>
<td>exp(x)</td>
<td>Returns the value of e (the base of natural logarithms) raised to the power of x.</td>
<td>-</td>
</tr>
<tr>
<td>fabs(f)</td>
<td>A synonym for abs(f).</td>
<td>-</td>
</tr>
<tr>
<td>floor(f)</td>
<td>The floor of a number, rounding any fractional part down.</td>
<td>-</td>
</tr>
<tr>
<td>fmod(x, y)</td>
<td>Floating point modulus function. fmod(x, y) returns the remainder after dividing x by y.</td>
<td>-</td>
</tr>
<tr>
<td>hypot(x,y)</td>
<td>The Euclidean distance function. hypot(x, y) returns the length of the hypotenuse of a right-angled triangle where the other sides have length x and y respectively.</td>
<td>-</td>
</tr>
<tr>
<td>id</td>
<td>The index number for each particle.</td>
<td>-</td>
</tr>
<tr>
<td>int(f)</td>
<td>Convert floating point number f to an integer, discarding any fractional part.</td>
<td>See also: trunc(f).</td>
</tr>
<tr>
<td>life</td>
<td>The maximum lifetime of a particle, in frames.</td>
<td>-</td>
</tr>
<tr>
<td>log(x)</td>
<td>Returns the natural logarithm of x.</td>
<td>-</td>
</tr>
<tr>
<td>log10(x)</td>
<td>Returns the base 10 logarithm of x.</td>
<td>-</td>
</tr>
<tr>
<td>mag(v)</td>
<td>Returns the magnitude (length) of the 3D vector v.</td>
<td>-</td>
</tr>
<tr>
<td>mass</td>
<td>The mass of the particle. Used when applying a force to a particle.</td>
<td>-</td>
</tr>
<tr>
<td>new</td>
<td>Returns 1 if the particle has just been created, 0 otherwise.</td>
<td>-</td>
</tr>
<tr>
<td>Function</td>
<td>Purpose</td>
<td>Related Functions</td>
</tr>
<tr>
<td>--------------</td>
<td>-------------------------------------------------------------------------</td>
<td>-------------------</td>
</tr>
<tr>
<td>norm(v)</td>
<td>Normalise the 3D vector v to have a length of 1.0 while pointing in the same direction.</td>
<td>-</td>
</tr>
<tr>
<td>opacity</td>
<td>A number between 0.0 and 1.0, where 0.0 is fully transparent and 1.0 is fully opaque.</td>
<td>-</td>
</tr>
<tr>
<td>pos</td>
<td>The position of the particle. This is a 3D vector.</td>
<td>-</td>
</tr>
<tr>
<td>pow(x, y)</td>
<td>Returns x raised to the power of y.</td>
<td>-</td>
</tr>
<tr>
<td>pow2(f)</td>
<td>Returns the square of f, i.e. f raised to the power of 2.</td>
<td>-</td>
</tr>
<tr>
<td>random</td>
<td>Returns a random number.</td>
<td>-</td>
</tr>
<tr>
<td>randomv</td>
<td>Returns a random vector. The vector directions are uniformly distributed around the unit sphere.</td>
<td>See also: uniform samplesphere.</td>
</tr>
<tr>
<td>rint(f)</td>
<td>Round the floating point number f to an integer.</td>
<td>-</td>
</tr>
<tr>
<td>sin(f)</td>
<td>Returns the sine of the angle f. The angle is in radians.</td>
<td>-</td>
</tr>
<tr>
<td>sinh(f)</td>
<td>Returns the hyperbolic sine of f.</td>
<td>-</td>
</tr>
<tr>
<td>size</td>
<td>The size of the particle.</td>
<td>-</td>
</tr>
<tr>
<td>sqrt(f)</td>
<td>Returns the square root of f. f must be greater than or equal to zero.</td>
<td>-</td>
</tr>
<tr>
<td>tan(f)</td>
<td>Returns the tangent of angle f. The angle is in radians.</td>
<td>-</td>
</tr>
<tr>
<td>tanh(f)</td>
<td>Returns the hyperbolic tangent of f.</td>
<td>-</td>
</tr>
<tr>
<td>trunc(f)</td>
<td>A synonym for int(f).</td>
<td>-</td>
</tr>
<tr>
<td>uniform</td>
<td>A synonym for randomv.</td>
<td>-</td>
</tr>
<tr>
<td>Function</td>
<td>Purpose</td>
<td>Related Functions</td>
</tr>
<tr>
<td>----------</td>
<td>---------</td>
<td>------------------</td>
</tr>
<tr>
<td>samplesphere</td>
<td></td>
<td></td>
</tr>
<tr>
<td>v(x, y, z)</td>
<td>Create a vector from three separate numbers.</td>
<td>-</td>
</tr>
<tr>
<td>vel</td>
<td>The velocity of the particle. This is a 3D vector.</td>
<td>-</td>
</tr>
<tr>
<td>x(v)</td>
<td>Get the x component of the 3D vector v.</td>
<td>-</td>
</tr>
<tr>
<td>y(v)</td>
<td>Get the y component of the 3D vector v.</td>
<td>-</td>
</tr>
<tr>
<td>z(v)</td>
<td>Get the z component of the 3D vector v.</td>
<td>-</td>
</tr>
</tbody>
</table>
Adjusting Particle Simulation Settings

If you want to adjust how many steps of particle simulation take place per animation frame, you can use the steps per frame control in the ParticleSettings properties panel. Sometimes simulations cannot generate enough accuracy only calculating once per frame, and the resulting particle movement can appear jagged. Steps per frame forces the simulation to assess the movement of the particles multiple times per frame to enable a more analogue movement. You should enter the lowest value you can, as having an unnecessarily high value can slow down your particle calculations.
Merging Particle Streams

If you have more than one set of particle nodes, and you want to combine them into one stream, ParticleMerge is your node. You can merge as many particle streams as you need into a single ParticleMerge.

1. Connect a ParticleMerge node to other particle nodes at any point in the particle stream.
2. Attach another particle stream (or a ParticleEmitter node) to another one of ParticleMerge's numbered inputs and you're all set.
Controlling Particles by Channel

The ParticleToGeo node allows you to control particles in a simulation by channel, giving you the ability to isolate certain particles at any point in the Node Graph. For example, you might want to freeze certain particles within a simulation while allowing others to emit as normal or apply a particle effect to the particles in a single channel.

ParticleToGeo allows you to control geometry or sprite particles by channel, but in the case of sprite particles, you can also influence their alignment using the align mode control.

The following example uses a very simple particle system containing three colored particles in separate channels and adds effects to one channel using ParticlesToGeo. The other particles are unaffected.
Split off the channel you're interested in using ParticleToGeo and MergeGeo nodes. Set the ParticleToGeo node's **channels** control to the channel you want to affect. In this example, channel c with the Sphere geometry is the affected particle.

To add a new texture to the geometry, you can use the AddMaterial node to apply a material to the affected channel.
Caching Particles

The ParticleCache node allows you to store the geometry simulation for a particle system to file. It can then be read back in different sessions of Nuke or on different machines without the need for recalculation.

This allows a particle system to be produced by an artist and then used by a render farm without recalculation, speeding up render times.

**Note:** The ParticleCache node doesn't replace the particle system. It just stores the simulation to disk and still relies on the particle system being connected in the same way with the same inputs. If anything in the particle system changes, the ParticleCache node detects this and shows an error to alert you to changes that have potentially been made upstream without your knowledge.

To cache your particle simulation:
1. Once you are happy with your particle simulation, select **Particles > ParticleCache** to create a ParticleCache node.
2. Place the ParticleCache node at the bottom of a single particle system or multiple merged particle systems.
Note: You cannot place a ParticleCache node in the middle of a string of particle nodes or beneath a Scene node connected to two separate particle streams.

3. In the ParticleCache properties, click the folder icon next to the file field and navigate to the directory where you want to cache the particle simulation. After the directory path, enter a name for the cache files and include the frame number variable (for example, ####) in the name. Click Open.
   ParticleCache uses the .nkpc file extension.

Note: ParticleCache may need to render up to 100 sub-frames. To account for this, it adds decimals to the file name's frame number. For example, if the file name in the file field is particle_cache.####.nkpc, ParticleCache may generate files called particle_cache.0001.01.nkpc, particle_cache.0001.02.nkpc, and so on.

4. If you have nodes downstream that generate motion blur, the particle system may need to request frames outside the normal frame range. If this is the case, increase the padding value in the ParticleCache properties to set the number of extra frames added to the start and end of the ParticleCache render.

5. Click Render.
   ParticleCache renders your particle simulation out to file frame by frame.

6. To use the cache data, enable read from file.
   If you get a "Particle cache data not found" error, return to step 4 and increase the padding value.
PrmanRender

PrmanRender is a render node that works together with Pixar’s PhotoRealistic RenderMan® Pro Server software to give you an even better quality render result. PrmanRender is an alternative to Nuke's ScanlineRender with additional features for rendering 3D scenes.

Setting Up RenderMan Pro Server and PrmanRender

In order to use the PrmanRender node, you need to have Pixar’s PhotoRealistic RenderMan Pro Server 20, or earlier, installed and licensed on your machine (for brevity, we call it RenderMan from now on). To do this:

1. If you're using Mac OS X 10.8 (Mountain Lion) or above, make sure X11 is installed on your system. Unlike previous versions of the operating system, 10.8 and above do not have X11 installed by default. For more information, see http://support.apple.com/kb/HT5293.

2. Follow the instructions in the RenderMan installation guide to get RenderMan working on your machine. This is in most cases enough to get you going with using both RenderMan and the PrmanRender node in Nuke. Note that RenderMan specifically needs two environment variables set in order to work with Nuke:
   • RMANTREE - This needs to point to the location of your RenderMan distribution.
   • Depending on which operating system you’re using either DYLD_LIBRARY_PATH (on Mac), LD_LIBRARY_PATH (on Linux) or PATH (on Windows). This environment variable needs to point to "/%RMANTREE%/lib". For more information on setting environment variables, see Environment Variables.

3. To make sure your PrmanRender node is working in Nuke, start the application from the terminal (for more information, see the Installation and Licensing chapter).

4. Create a PrmanRender node in Nuke (3D > RenderMan > PrmanRender) and connect it to your nodes. You can try the following combination of nodes as a test: Checkerboard > Cube > PrmanRender > Viewer.
Using The PrmanRender Node

The PrmanRender node can render nearly everything that you previously used ScanlineRender for, but with PrmanRender, you have control over aspects like shadows and reflections in your render result.

Connect the PrmanRender node to your Scene node, Camera node and any optional inputs, in the same way you would connect ScanlineRender. For more information about connecting a ScanlineRender node, see Setting Up a Scene.

On the PrmanRender tab, you can select which aspects you’d like to render out by checking shadows, reflections, refractions or dof (depth of field). For more information, see Shadows, Reflections, Refractions and Depth of Field.

You can select your projection mode in the projection mode dropdown:

- **Perspective** - objects in front of the camera have the illusion of depth defined by the camera’s focal-length and aperture.
- **Orthographic** - objects are viewed using a parallel projection.

If necessary, also adjust:

- **overscan** - to set how many pixels are rendered over the right/left and top/bottom of the frame, if a subsequent operation requests this.
- **ambient** - to add global ambient lighting.
Render Quality

On the **Sampling** tab, you can adjust controls that affect your render quality. Adjust:

- **ray trace max depth** - to set the maximum depth of the view rays PrmanRender uses to trace your scene and make render calculations.
- **pixel samples** - to set the number of samples to render per pixel. Having more samples increases your render quality, but also increases render time.
- **filter** - to select a texture sampling filtering algorithm. For more information on the different options, see [Choosing a Filtering Algorithm](#).
- **antialiased filter** - to select an antialiasing filter: `box`, `triangle`, `catmull-rom`, `sinc`, `gaussian`, `mitchell`, `separable-catmull-rom` or `blackman-harris`.
- **antialiased filter size** - to select the size of the antialiasing filter.
- **shading rate** - to set the shading calculation for primitives. This value, along with pixel samples, directly affects your rendering time and the final quality of your results. A small shading rate value means your render takes more time, but the quality is very high. A large value on the other hand means your render is faster, but the final quality is not as good.
Shadows, Reflections, Refractions and Depth of Field

On the **PrmanRender** tab, you can select if you want to render shadows, reflections, refractions and depth of field, or all of them into your result. All these effects are calculated using a retracing method that is based on drawing rays from the camera to the object. Check the box for:

- **shadows** - to add shadows to your render. You can adjust the parameters for your shadows on your Light node’s **Shadows** tab. For example, if you adjust the **sample width**, the shadows are softer. For more information, see [Casting Shadows](#).

- **reflections** - to add reflections to your render. You can adjust the parameters for your reflections on your Reflection node’s properties panel. See [Using the Reflection Node](#).

- **refractions** - to add refractions to your render. You can adjust the parameters for your refractions on your Refraction node’s properties panel. See [Using the Refraction Node](#).

- **dof** - to add depth of field to your render.
Motion Blur Parameters

On the **Sampling** tab, you can adjust controls that affect motion blur. Adjust:

- **motion blur samples** - to set the number of samples to render per pixel when motion blurring.
- **shutter** - Enter the number of frames the shutter stays open when motion blurring. For example, a value of 0.5 would correspond to half a frame. Increasing the value produces more blur, and decreasing the value less.
- **shutter offset** - Select when the shutter opens and closes in relation to the current frame value when motion blurring:
  - **centered** - to center the shutter around the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 29.5 to 30.5.
  - **start** - to open the shutter at the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 30 to 31.
  - **end** - to close the shutter at the current frame. For example, if you set the shutter value to 1 and your current frame is 30, the shutter stays open from frame 29 to 30.
  - **custom** - to open the shutter at the time you specify. In the field next to the dropdown menu, enter a value (in frames) you want to add to the current frame. To open the shutter before the current frame, enter a negative value. For example, a value of -0.5 would open the shutter half a frame before the current frame.
- **randomize time** - adjust this to add randomness to the distribution of samples in time so they don't produce regularly spaced images. The larger the value, the larger the time difference between the samples.
- **shutter opening** - to set the shutter opening behavior. Select:
  - **none** - to use the default shutter opening method, resulting in instantaneous timing to open and close.
  - **linear** - to set the shutter to open and close in linear intervals.
  - **bezier** - to set the shutter to open and close more gradually, according to a Bezier curve.
Shader Parameters

On the Shader tab you can select which channels are affected by motion vectors and output vectors.

- Click motion vectors and select the type of vectors you’d like to render:
  - off - no motion vector information is rendered.
  - velocity - store the velocity of every single pixel in the motion vector channels.
  - distance - for every pixel, store the distance (in pixels) between samples in the motion vector channels.

- Adjust motion vector channels to select which channels you want the motion vectors to be output to.
- Check output vectors if you want to render output vectors.
- Select which channels to apply surface points and surface normals from the surface point and surface normal dropdowns.
RIB Parameters

With the RIB (RenderMan Interface Bytestream) parameters, you can choose to filter the information Nuke generates for RenderMan, set your arguments to it, and output your own RIB file. On the RIB tab:

• filter - check this to filter the information on your scene, generated by Nuke for RenderMan. In order to do this, Nuke calls a Python function called nukescripts.renderman.filterRIB. Filtering can make the render startup slightly slower as a temporary RIB file is created for each render.

• arguments - specify your arguments for filtering. This string is passed by Nuke's Python filter function as extra arguments to RenderMan. If you want to use your own filter, you can also replace Nuke's Python function, and have your arguments passed directly to your own Python function. For example, you could set the filter arguments to "-rif myfilter.so" to load your own RenderMan Interface filter.

Tip: For further details on filtering, have a look at your RenderMan documentation.

A RIB file is a RenderMan-compatible ASCII file with information that Nuke generates when rendering your footage. To output a RIB file:

1. In the file field under output, specify the file name and location for your RIB file.
2. Click Execute.
Using the ModifyRIB Node

You can use the ModifyRIB node to insert RIB (RenderMan Interface Bytestream) statements into your script to modify a RIB stream before it's passed to the PrmanRender node. This can be useful in situations where you might want to adjust the shading on the surface of an object, replace it, or perform a variety of transformations and adjustments to multiple objects in a scene. For example, to replace an object in your script with basic geometry from RenderMan:

1. Add a Scene node to the basic script that you created at the beginning of the chapter and connect two Light nodes and a Camera.
2. Click 3D > Modify > RenderMan > ModifyRIB and insert the node between the Cube and the Scene.
3. Next to the archive field, uncheck use. This activates the statements field.

**Note:** You can load a RIB archive by leaving use checked and clicking the Select file button to locate it.

4. In the ModifyRIB control panel select replace from the operation dropdown menu and enter the following RIB statement in the statements field:
   
   Sphere 0.25 -0.25 0.25 360

   You should now see a basic sphere in the Viewer where the cube was.
5. To change the color of the sphere and add a basic surface shader to the sphere, enter the following statements underneath:
   
   Color 0.4 0.6 0.1
   Surface "wood"

   This changes the color of the sphere to green and applies a wood-like finish to the surface.

**Note:** If you're copying statements directly from a RIB file, only copy statements from between WorldBegin and WorldEnd as copying the entire contents of the file may result in instability issues.

For more information on RIB statements, please refer to the documentation provided with RenderMan.
Using the Reflection Node

Reflection is the familiar physical phenomenon where an image of an object is cast back from a particular kind of surface, such as glass or water. Using PrmanRender node, you can replicate this effect in your render result of 3D objects, and using the Reflection node, you can adjust the controls for creating the reflection effect. PrmanRender uses raytracing to create this effect and you can use the Reflection node to adjust the result.

**Note:** Nuke's RayRender node can also use the Reflection node to control reflections.

1. Create a Reflection node by clicking **3D > Shader > RenderMan > Reflection.**
2. Connect the node to the PrmanRender node and set the controls to adjust your reflection:
   - **reflection color** - sets the color of the reflection.
   - **value** - sets the intensity of the reflection.
Using the Refraction Node

Refraction is the familiar physical phenomenon of light traveling differently through different materials and thus reflecting differently off objects behind that material. For example, if you have a glass of water with a straw in it, the part of the straw that’s not in water appears to be in a different angle to the part which is in the water. This is due to water bending the light waves. PrmanRender uses raytracing to create this effect and you can use the Refraction node to adjust the result. Without the PrmanRender node and RenderMan software though, the Refraction node has no effect.

1. Create a **Refraction node** by clicking **3D > Shader > RenderMan > Refraction**.
2. Connect the node to the PrmanRender node and set the controls to adjust your refraction:
   - **refraction index** - slide to change the type of refraction.
   - **value** - sets the intensity of the refraction.
Using F_DeFlicker2

When working in film, you sometimes have to deal with shots that have a luminance flicker. This chapter concentrates on removing flicker using F_DeFlicker2.

Introduction

F_DeFlicker2 is used to remove flicker. It is particularly suitable for removing flicker that is localized and dependent on the geometry of the scene (that is, flicker that is not present across the whole of the image), such as that caused by an unsynchronized fluorescent light in a shot. It works by calculating the gain between the current frame and each frame in a small analysis range surrounding it. It then tries to adjust the gain so that it varies smoothly over this frame range. This means it is better at reducing fast flicker than flicker which varies slowly over the image sequence, as the latter will already appear smooth over the small frame range and F_DeFlicker2 will leave it largely untouched.

The algorithm used by F_DeFlicker2 can introduce blurring in areas where there is rapid motion. This problem could be alleviated by using local motion estimation before deflickering the frames. However, this process is complicated by the fact that the presence of flicker can adversely affect the results of the motion estimation. F_DeFlicker2 therefore adopts a two-stage approach to this problem. First, the normal deflickering process is performed. Then, the motion vectors for the sequence are calculated on the resulting deflickered frames, and applied to the original frames in order to align them. The deflicker calculation is then performed on the aligned frames to give the final result. To use this approach, turn on **Use Motion** in F_DeFlicker2.

**Note:** Because F_DeFlicker2 looks at input frames outside the current frame when performing its calculation, it can be a computationally expensive plug-in. As such, using more than two instances of F_DeFlicker2 in a node tree will dramatically increase render times. It is strongly advised therefore, that you render each instance out separately.
Quick Start

To remove flicker from a sequence:

1. Select **Image > Read** to load the sequence you want to remove flicker from.
2. Select **FurnaceCore > DeFlicker2** to apply DeFlicker2. View its output.
3. If you're not happy with the results, adjust the DeFlicker2 parameters. The available parameters are described below.
Parameters

The parameters for F_DeFlicker2 are described below.

- **DeFlicker Amount** - Use this to reduce flicker without removing it entirely; smaller values mean more will be left behind.

- **Block Size** - To find where a certain pixel is located in the analysis range, the deflicker algorithm looks for a block of pixels centered around that pixel. Block size defines the width and height of these blocks (in pixels). On rare occasions, a large block size can produce data that's lacking in detail. This is because a small feature can fit into a large block, causing the motion estimation to concentrate on the background motion and ignore the small feature. A small value, instead, can produce a noisy motion field, as there aren't enough constraints in a small block to fit the motion accurately. In most cases, however, the default value is small enough so that details aren't lost, and the smoothing step of the algorithm ensures the motion field isn't too noisy. Therefore, this value very rarely needs editing.

- **Use Motion** - Turn this on to do a second deflicker pass using motion-compensated frames. This can improve results in areas where there is fast motion, where the initial deflicker pass can introduce blurring.

- **Vector Detail** - Determines the density of the motion vectors used when **Use Motion** is turned on. The maximum value of 1 will generate one vector per pixel. This will produce the most accurate vectors but will take longer to render. A value of 0.5 will generate a vector at every other pixel.

- **Analysis Range** - The number of frames searched each side of the current frame when calculating the flicker. Higher values may give better results, but can also bring in erroneous information and take longer to process.
Using F_ReGrain

This chapter looks at adding grain to sequences using F_ReGrain.

Introduction

F_ReGrain is used to add grain to a sequence. It has been designed to sample an area of grain from one image and then to generate unlimited amounts of this grain with exactly the same statistics as the original. This new grain can then be applied to another image.

The figure on the left shows an enlarged and exaggerated sample of grain from Kodak 320 film stock. F_ReGrain was used to sample the original Kodak 320 stock and synthesize a plate of grain. The result is shown in the figure on the right. Note that the grain characteristics closely match the original.

![Kodak 320. F_ReGrain.]

Similarly, below the figure on the left is a sample from Kodak 500 film stock and the figure on the right shows this replicated using F_ReGrain.

![Kodak 500. F_ReGrain.]

1766
Quick Start

You can sample grain from an image and apply it to another or select from a variety of pre-sampled, standard grain types.

Adding Sampled Grain

To add grain to a sequence, do the following:

1. Select **Image > Read** to load the sequence you want to add grain to. Then, load the image you want to sample grain from.
2. Make sure you are working at full resolution and not proxy resolution. F_ReGrain will not work at proxy resolution. (See **Proxy Resolutions**.)
3. Select **FurnaceCore > F_ReGrain**.
4. Connect the sequence that you want to have grain to F_ReGrain’s source (Src) input. Then, connect the sequence you want to sample grain from to the **Grain** input. View the output from F_ReGrain.
5. Set **Grain Type** to **From Grain Clip**.
6. Position the on-screen sample region over an area of the **Grain** sequence just containing grain and no picture detail. See the figure below.

This shows two possible selection regions that contain no edge detail and little luminance variation.

It is important to get your selection right. You should avoid any image detail or even a plain area that has luminance variations underneath the grain. The better this initial selection, the better the result will be. If you can’t find a decent sample area on the current frame, then try other frames from the same film stock. The default size of the sample area should be enough to gather
information about the grain characteristics of your image. However, you may need to change its size and shape to fit over a plain area free of image detail.

**Warning:** There is a minimum size of this sample area below which the statistical analysis of the grain will be unreliable. If the sample area you select is too small, you will see a warning message which prompts you to select a larger region. (See Proxy Resolutions.)

7. View the output of F_ReGrain to judge the results. The output will now contain the **Src** image with grain from the **Grain** image applied. Both the size and the luminance of the new grain can be manually tweaked using **Grain Size** and **Grain Amount** respectively. It helps to view the **Grain** input while editing the parameters of F_ReGrain.

The grain is sampled on a single frame which is set when you adjust the sample area (or by manual adjustment of the **Analysis Frame** parameter). Although it is sampled on only one frame, the algorithmically created grain will change from frame to frame but mirror the characteristics of the sample grain.

**Using Pre-Sampled, Standard Grain Types**

If you don't have an image to sample grain from, you can also select from a variety of pre-sampled, standard grain types. Do the following:

1. Select **Image > Read** to load the sequence you want to add grain to.
2. Make sure you are working at full resolution and not proxy resolution. F_ReGrain will not work at proxy resolution. (See Proxy Resolutions.)
3. Select **FurnaceCore > F_ReGrain**.
4. Connect the sequence that you want to have grain to F_ReGrain's source (**Src**) input.
5. Set **Grain Type** to **Preset Stock**.
6. Try the different grain types using the **Preset Stock** dropdown menu. 2K, 4K, aperture corrected, and non aperture corrected stocks are included. Individual color channels can be selected and adjusted using the **Advanced** parameters.
Response

In its default setting, F_ReGrain adds the same amount of grain over the whole image. However, the amount of grain on an image is normally a function of luminance. Various parameters in the **Grain Response** group allow you to adjust how the amount of grain added varies with luminance:

- Pressing **Sample Grain Response** will cause the variation of the amount of grain with luminance to be calculated from the **Grain** input, and switching on **Use Sampled Response** will apply these curves to the grain added to the **Src** sequence.

- To view the sampled response curves, switch on **Draw Response**; an example is shown in the figure below.

- The amount of grain added to the lowlights, midtones and highlights of the image can be adjusted using the **Low Gain**, **Mid Gain** and **High Gain** parameters. The effect of adjusting these can also be seen on the response curves.

![Grain Response Curve](image)

This shows an example of the grain response with luminance. The x axis represents luminance and the y axis the amount of grain.
Checking the Result

To test that the new grain is the same as the old grain, set **Output** to **Grain Plate**.

This generates a sheet of grain with the same luminance level as the mean of the sample region. The sample region with the original grain is also displayed. It should be impossible to differentiate between the two regions. The figure on the left shows a good selection area giving a good test plate of grain in the figure on the right.

![Good selection area](image1.png) ![...producing a good test plate of grain, free of artifacts.](image2.png)

Below, the figure on the left shows a poor selection area since it contains image detail. The figure on the right shows the resulting test plate which clearly highlights the problem.

![Bad selection area](image3.png) ![...producing a poor result.](image4.png)
Proxy Resolutions

Grain manipulation at proxy resolution should be avoided as the results are unreliable. The grain selection area may be too small at proxy resolution to give a good result, and making this area larger may drag in unwanted detail from the image. If you try to use F_ReGrain at proxy resolution, we simply pass the image through untouched and issue the following warning:

*Cannot work at proxy scale.*

We decided that this was preferable behaviour to doing poor grain replication at proxy resolution. You can, of course, crop the input clip and work with that rather than the proxy. There is a minimum size for the selection box, which is about 37x37 at base resolution. If the box you select is smaller, you will get this warning along the top of the viewer:

*Sample box is too small - please select a larger sample of the grain.*
Parameters

The parameters for this plug-in are described below:

• **Grain Type** - Selects whether the grain is sampled from the Grain image (From Grain Clip) or from a set of standard stocks.
  - **Preset Stock** - grain characteristics are sampled from a supplied film stock. 2K, 4K, aperture corrected and non aperture corrected stocks are supplied. Although standard stocks are included, it is recommended where possible that you sample from the film stock you are trying to match.
  - **From Grain Clip** - samples and reconstructs the grain characteristics from the Grain input.
• **Preset Stock** - The film stock the grain characteristics are sampled from when Grain Type has been set to Preset Stock. Common Fuji and Kodak stocks are supplied. The exposure can be under, over, or, if left blank, non aperture corrected. The size is either 2K or 4K pixels. For example, FUJIF500 2K refers to the grain characteristics sampled from a 2K plate of Fuji Film 500 film stock non aperture corrected.
• **Grain Amount** - Adjusts the brightness of the grain. Setting this to 0 means no grain is added.
• **Grain Size** - Adjusts the size of the grain granules. The larger the value, the bigger and softer the granules.
• **Output** - Sets whether to render the result or a test image.
  - **Result** - shows the Source image with the grain applied.
  - **Grain Plate** - shows a test image with the grain applied. This test image is composed from a section of the input image surrounded by a uniform solid color sampled from the image with the grain applied. If the inner area is indistinguishable from the outer area, then you have a good grain sample.

**Analyse** - This is a push button which will trigger an analysis of the input clip. Press this button if the input clip from which the grain was analyzed has changed but you do not want to move the analysis region to trigger re-analysis. Whenever the input clip changes, you will see the following warning in the Viewer:

*The clip from which the grain was analyzed has changed. Press Analyse or move the analysis region to re-analyze grain.*

• **Analysis Region** - A selection box that marks the region of image used to analyze the grain when Grain Type is set to From Grain Clip. This part of the frame must contain no image detail, only grain.
• **Analysis Region BL** - controls the position of the bottom left corner of the analysis region.
• **Analysis Region TR** - controls the position of the top right corner of the analysis region.
• **Analysis Frame** - sets the frame to sample the grain from.
**Grain Colour Space** - This tells F_ReGrain what color space the grain sample clip was in when the grain originated. Setting this correctly ensures that the grain is not exaggerated by any color space conversions prior to sampling.

- **Cineon**
- **sRGB**
- **Linear**

**Advanced** - The parameters under Advanced allow detailed adjustment of the grain.

- **Process Red** - Switch this on to process the red channel.
- **Red Amount** - sets the brightness of the grain in the red channel.
- **Red Size** - adjusts the size of the grain granules in the red channel.
- **Process Green** - Switch this on to process the green channel.
- **Green Amount** - sets the brightness of the grain in the green channel.
- **Green Size** - adjusts the size of the grain granules in the green channel.
- **Process Blue** - Switch this on to process the blue channel.
- **Blue Amount** - sets the brightness of the grain in the blue channel.
- **Blue Size** - adjusts the size of the grain granules in the blue channel.

**Grain Response** - The parameters under Grain Response allow the amount of grain added to be varied as a function of the image luminance.

**Apply Grain In** - This controls what color space the grain sample is re-applied to the image.

Generally, this should be set to Grain Colour Space to ensure the most accurate recreation.

You may want to override this though for some looks or special cases.

- **Cineon / sRGB / Linear** - The grain sample will be applied in the specified space.
- **Grain Colour Space** - The Grain sample will be applied in the color space set in the Grain Colour Space dropdown menu, in the Grain Sample section.

**Low Gain** - adjusts the gain of the grain in the lowlights.

**Mid Gain** - adjusts the gain of the grain in the midtones.

**High Gain** - adjusts the gain of the grain in the highlights.

**Use Sampled Response** - switch this on to scale the brightness of the grain as a function of the luminance of the Grain image.

**Sampled Response Mix** - this control is usually set to 1. Decreasing it reduces the effect of the response curves until, at 0, they have no effect on the output. This parameter is only available if Use Sampled Response is on.

**Sample Grain Response** - press this to update the response curves from the current frame. Multiple presses accumulate the grain response rather than resetting every time. This parameter is only available if Use Sampled Response is on.
• **Reset Grain Response** - press this to reset the grain curves to their default (flat) response. This parameter is only available if **Use Sampled Response** is on.

• **Draw Response** - overlays the response curves on the bottom left corner of the viewer. This parameter is only available if **Use Sampled Response** is on.
Color Space in FurnaceCore Plug-ins

Some of the algorithms in the FurnaceCore tool set are sensitive to the color space of the source footage. If the footage is not in the expected color space, you may get poor results from some of the plug-ins.

Like Nuke, FurnaceCore expects all footage to be in **Linear** space. Nuke converts all footage to Linear upon import, so unless you have changed the colorspace in your **Read** nodes (or in the node tree by using a **Colorspace** node before a FurnaceCore plug-in), your footage should be Linear by the time it reaches the plug-in anyway.

If you know that the input to a plug-ins isn’t Linear, you should use a Colorspace node before and after the plug-in to convert to and from Linear for processing. This should ensure optimal results.

Color Space in F_ReGrain

Because color space transformations can distort the look of grain, and by default, Nuke converts your footage to linear, if you are sampling grain from your own plate, you need to make sure you tell the plug-in what space the plate was in originally, so the sample isn’t distorted.

This is accomplished by setting the **Grain Colour Space** dropdown menu in the F_ReGrain controls to the right space. For example, if you were sampling grain from a film scan, you would want to set this to **Cineon**. If you had footage from a digital video camera, this would most likely be **sRGB**. F_ReGrain automatically sets the right color space when using one of the pre-sampled grain clips, which are in **sRGB**.

F_ReGrain also works best when applying grain in the same color space that the sampled grain originally existed. There is a dropdown menu in the **Response** section of the F_ReGrain controls which allows you to match or override this. It defaults to applying the grain in the same space as the sample.
Using F_WireRemoval

This chapter looks at the removal of wires from images using F_WireRemoval.

Introduction

Many effects movies feature complicated stunts that require an actor to be suspended by wires for their safety. These wires need to be digitally removed.

There are many ways of doing this, including painting the wires out frame by frame and replacing large parts of the background to cover the wires. The method used depends on the type of image under the wire. F_WireRemoval is particularly good at replacing the painting method but it also includes tools to assist in clean plating when new backgrounds have to be tracked into place.

Three wires. Two of the wires removed using F_WireRemoval.
Background

Clean Plates

The use of clean plates in wire removal is very common and gives good results in certain situations.

Consider a scene shot with an actor suspended from wires and then the same scene shot again without the actor. This second sequence is called the clean plate. The wires from the first shot can be painted out using pixels from the clean plate leaving the actor suspended in thin air.

Shooting a clean plate if the camera is locked off is easy. If the camera moves, then motion control rigs can be used to exactly reproduce the first pass. But it doesn't always work, particularly if the scene is shot against backgrounds that don't look the same on the second pass, such as clouds, sky, or smoke. Motion control rigs are also expensive and that makes them a rarity. Often, a clean plate is not shot during the filming and the compositor is left to create one by merging together unobstructed pixels from many frames. This single frame can then be tracked into place to cover the wires.

FurnaceCore

FurnaceCore's wire removal plug-in should make the process of wire removal much easier. It is particularly good at removing wires over heavily motion blurred backgrounds or wires over smoke, dust, or clouds. It can be used to remove each wire in a sequence or to quickly create a clean plate which can then be tracked into place.

The reconstruction of the background behind the wire can be done spatially, in other words, using only information from the current frame. Alternatively, motion estimation techniques can be used to warp frames from before and after onto the current frame so that, where available, background pixels from these frames can be used to improve the repair. Our reconstruction methods are unique in that they remove the wire without removing and reapplying the grain. They are also tuned to remove long thin objects, leaving other objects untouched. For example, if transient foreground objects cover the wire, they will be left alone.
Reconstruction Methods

Our four reconstruction methods are:

• Spatial
• Temporal With Static Scene
• Temporal With Moving Scene
• Clean Plate

The spatial method takes the background information from adjacent pixels in the current frame, and the clean plate method takes the information from a separate clean plate input. The other methods are temporal and attempt to get background information from frames either side of the current frame. Where this information is not available, for example, because the wire covers part of the same region in one or more of the other frames, the spatial method will be used for the repair.

The spatial method is fastest. It uses a slope-dependent filter that interpolates across the wire at the most likely angle, given the image behind the wire. It works well when the wire is over backgrounds such as sky, clouds, or smoke, and can also cope with some backgrounds where there is a noticeable gradient, such as the edge of a roof, building, or car. If this fails and the wire is moving relative to the background, you should try one of the temporal methods. These look at frames before and after the current one to find likely pixels with which to repair the region and use the spatial method in areas where there are none available.

Where traditional clean plates are possible or give better results than using F_WireRemoval to repair the wire on each frame, you can supply a clean plate as the fourth input to the plug-in. It will then automatically match-move it to repair each frame. If the overall luminance of the background is different to that of the clean plate or varies during the sequence, turn on Luminance Correct. The luminance of the background pixels used for the reconstruction will then be adjusted before the repair is made. Of course, various FurnaceCore tools can be used to create a clean plate, including an F_WireRemoval pass on a single frame where the repair has been successful.
Tracker

F_WireRemoval incorporates a tracker which can automatically track a moving wire through a clip. This tracker has its own control panel, which will float inside the NukeX viewer if you have checked **Show On Screen Controls** in the WireRemoval controls. Below is a screenshot that illustrates this.

![Screenshot with tracker panel.](image)

You can resize the panel by dragging its edges. For details of the rest of the controls, see the section on Tracker Controls on [Tracker Controls](#). All the controls are also available in the WireRemoval properties.
Quick Start

To remove a wire from an image sequence, do the following:

1. Select **Image > Read** to load the clip that needs a wire removed.

2. Select **FurnaceCore > F_WireRemoval**. Connect your image sequence to F_WireRemoval’s **Source** input. View the output from F_WireRemoval.

3. Make sure **Output** is set to **Source**, and use the on-screen tools to draw a region that defines the position and shape of the wire to be removed. For more information, see **Positioning the On-Screen Wire Tool**.

4. If the wire you want to remove is moving, start the wire tracker by clicking on the **Track Forwards** button, so that it follows the wire during the clip. For more information, see **Tracker Controls**.

5. From the **Repair** dropdown menu, choose a repair method that removes the wire. If you choose to use the **Clean Plate** method, connect a clean plate image to F_WireRemoval’s **CleanPlate** input. For more information on the available methods, see **Reconstruction Methods**.

6. To see the output, set **Output** to **Repair**.

7. If necessary, increase **Overall Width** to expand the width of the repair region until the wire disappears.
Positioning the On-Screen Wire Tool

To position the on-screen wire tool, do the following:

1. In the F_WireRemoval controls, set **Output** to **Source**. This way, you will be able to see the wire you're trying to remove but won't have to wait for F_WireRemoval to repair the wire every time you change wire tool's position.

2. Choose the number of points that are needed to describe the wire: two for straight lines, three for simple curves, or five for more complex shapes. This can be done using the **Type** dropdown in the properties panel, or by toggling the Number of Points button in the on-screen tracker panel.

   **Tip:** To show the on-screen tracker panel, check **Show On Screen Controls** in the WireRemoval properties.

3. Position the on-screen wire tool so that it roughly fits the wire you want to remove. To move the entire tool, drag its central line to a new location. To move an individual wire point on the tool, drag its center. To move an individual point in either a horizontal or vertical direction only, drag the horizontal or vertical line on the cross that marks the point.

4. Press the Snap To button in the WireRemoval properties or the on-screen tracker panel. This will find the edges of the wire and adjust the removal region to fit it more closely. It does this by locating the areas of highest gradient, which should correspond to the edges of the wire.

5. In practice, the edges of the wire will be slightly soft, so the resulting region might not cover the whole width of the wire. To correct for this, you can adjust the **Overall Width** parameter. This expands the region outwards, over all key frames, to ensure the entire width of the wire is covered.
Tracking

F_WireRemoval incorporates a tracker that tracks the region to be repaired through the image sequence.

To use the tracker, do the following:
1. Drag the on-screen wire tool to position the repair region on top of the wire on one frame. This sets a user keyframe for the current frame, enabling tracking.

2. To start the tracker, press the Track Forwards button.
   - If you haven't set a user keyframe (step 1), the tracker cannot be started and F_WireRemoval displays the following message:
     **Please select a user key frame before tracking.**

3. When the track is ready, check that the wire has been correctly tracked across the other frames and adjust the track if necessary. The tracking controls in the tracker panel are outlined under **Tracker Controls**. Any points initially positioned off the image will remain off the image. This way, the tracker will not lose the wire at the edges of the image.

4. If after tracking the wire you adjust the position of the on-screen wire tool manually on one frame, you may want to create a smooth transition between the user keyframe you created and the track keyframes around it. The easiest way to do this is to go on the previous and next frame on the timeline and press the Delete track keyframe button. This deletes the track keyframes around the frame you adjusted and forces NukeX to interpolate smoothly around the adjustment. Similarly, if there is too much jitter in the track between some frames, you can delete the track keyframes between these frames. This way, NukeX will interpolate smoothly from one frame to another.

If it is not possible to track the wire automatically, the wire can be manually keyframed. Adjusting the on-screen wire tool on each frame will automatically set user keyframes. Alternatively, they can be set by pressing the Create User Keyframe button in the tracker panel.
User Keyframes and Track Keyframes

User keyframes are keyframes you set manually. When a user keyframe has been set on the current frame, this is indicated by a red key that appears on the on-screen wire tool. Track keyframes are keyframes set by F_WireRemoval's tracker.

User keyframes override track keyframes on the same frame. This has the following consequences:
• Once you set a user keyframe, the track keyframe on that frame will be lost.
• When you track a wire, existing user keyframes will not be replaced by track keyframes.
Indicators on the On-Screen Wire Tool

You may see the following indicators appear on the on-screen wire tool:

<table>
<thead>
<tr>
<th>Control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="true" alt="✓" /></td>
<td>The current frame is inside the range of frames to track the wire over.</td>
</tr>
<tr>
<td><img src="false" alt="✗" /></td>
<td>The current frame is not inside the range of frames to track the wire over.</td>
</tr>
<tr>
<td><img src="up" alt="▲" /></td>
<td>The current frame is the first frame in the range of frames to track the wire over.</td>
</tr>
<tr>
<td><img src="down" alt="▼" /></td>
<td>The current frame is the last frame in the range of frames to track the wire over.</td>
</tr>
<tr>
<td><img src="key" alt=" setUserKeyFrame" /></td>
<td>A user key frame has been set on the current frame.</td>
</tr>
</tbody>
</table>
Tracker Controls

The following controls appear in the on-screen tracker panel that you can display in the Viewer by checking **Show On Screen Controls** in the WireRemoval properties.

<table>
<thead>
<tr>
<th>Control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Image] Toggle display mode</td>
<td>Toggle display mode - Toggle the display mode for the on-screen wire tool: show the points and lines, show just the points, or hide both points and lines.</td>
</tr>
<tr>
<td>![Image] Number of points</td>
<td>Number of points - Changes the number of points used to describe the wire.</td>
</tr>
<tr>
<td>![Image] Create user key frame</td>
<td>Create user key frame - Creates a user keyframe.</td>
</tr>
<tr>
<td>![Image] Delete user key frame</td>
<td>Delete user key frame - Deletes a user keyframe.</td>
</tr>
<tr>
<td>![Image] Snap to wire</td>
<td>Snap to wire - Finds the edges of the wire and snaps the edges of the region onto them.</td>
</tr>
<tr>
<td>![Image] Track backwards</td>
<td>Track backwards - Plays backwards through the sequence tracking from frame to frame.</td>
</tr>
<tr>
<td>![Image] Step backward</td>
<td>Step backward - Tracks backwards one frame.</td>
</tr>
<tr>
<td>![Image] Step forward</td>
<td>Step forward - Tracks forward one frame.</td>
</tr>
<tr>
<td>![Image] Track forwards</td>
<td>Track forwards - Plays forwards through the sequence tracking from frame to frame.</td>
</tr>
<tr>
<td>![Image] Smart track</td>
<td>Smart track - Tracks from beginning to end of frame range in an intelligent order.</td>
</tr>
<tr>
<td>Control</td>
<td>Description</td>
</tr>
<tr>
<td>---------</td>
<td>-------------</td>
</tr>
<tr>
<td><img src="image" alt="Delete Track Key Frames Backwards" /></td>
<td><strong>Delete track key frames backwards</strong> - Deletes track keyframes backwards through the sequence until either a user keyframe or the beginning of the sequence is reached.</td>
</tr>
<tr>
<td><img src="image" alt="Delete Track Key Frame and Step Backward" /></td>
<td><strong>Delete track key frame and step backward</strong> - Deletes a track keyframe and steps backwards one frame.</td>
</tr>
<tr>
<td><img src="image" alt="Delete Track Key Frame" /></td>
<td><strong>Delete track key frame</strong> - Delete the current track keyframe.</td>
</tr>
<tr>
<td><img src="image" alt="Delete Track Key Frame and Step Forwards" /></td>
<td><strong>Delete track key frame and step forwards</strong> - Deletes a track keyframe and steps forwards one frame.</td>
</tr>
<tr>
<td><img src="image" alt="Delete Track Key Frames Forwards" /></td>
<td><strong>Delete track key frames forwards</strong> - Deletes track keyframes forwards through the sequence until either a user keyframe or the end of the sequence is reached.</td>
</tr>
<tr>
<td><img src="image" alt="Delete All Track Key Frames" /></td>
<td><strong>Delete all track key frames</strong> - Deletes all track keyframes from the sequence.</td>
</tr>
<tr>
<td><img src="image" alt="Delete All Track and User Key Frames" /></td>
<td><strong>Delete all track and user key frames</strong> - Deletes both track keyframes and user keyframes.</td>
</tr>
</tbody>
</table>

The corresponding controls are also available in the WireRemoval properties.
Parameters

The parameters for this plug-in are described below.

Tracker buttons - most of the on-screen tracker panel buttons also appear here. For more information, see Tracker Controls.

**Type** - this parameter controls the number of points on the on-screen wire tool. Choose the number of points needed to describe the wire you wish to remove.
- **Two Points** - choose this if your wire is straight.
- **Three Points** - choose this if your wire is a simple curve.
- **Five Points** - choose this if your wire has an s-shaped curve.

**On-Screen Wire** - the display mode for the on-screen wire tool.
- **Show** - show both points and lines.
- **Hide** - hide both points and lines.
- **Points only** - only show the points.

**Show On Screen Controls** - use this to show or hide the tracker panel in the Viewer.

**Output** - sets the output mode for the plug-in.
- **Source** - output the untouched source image. Use this output mode to position the on-screen wire tool over the wire you wish to remove.
- **Repair** - output the repaired source image, with the wire removed from under the on-screen tool.
- **Wire Matte** - renders a matte for the wire. This may be useful if the wire has been tracked but cannot be repaired using F_WireRemoval and other techniques have to be used.
- **Repair Matted** - output the repaired source image and a matte in the alpha channel. If you want, you can adjust your image further manually using the matte.

**Track Range** - the range of frames to track the wire over.
- **Specified Range** - use the **Track Start** and **Track End** parameters to specify the range over which to track the wire.
- **Source Clip Range** - track the wire over the entire range of the **Source** clip.

**Track Start** - use this to specify the start of the tracking range when **Track Range** has been set to **Specified Range**.

**Track End** - use this to specify the end of the tracking range when **Track Range** has been set to **Specified Range**.

**Repair** - sets the algorithm used to remove the wire from under the grain.
• **Spatial** - this method uses a slope dependent filter that interpolates across the wire at the most likely angle, given the image behind the wire. It uses information from the current frame only.

• **Temporal With Static Scene** - this method uses local motion estimation to align frames from before and after onto the current frame. If the wire is not in the same place with respect to the background in these two frames, it can then use background information from them to fill in the background behind the wire in the current frame. Where no such information is available, for example, if the wire covers part of the same region in all three frames, the spatial repair method above will be used. This is useful for sequences where the wire is moving and where the motion in the rest of the scene is non-uniform, for example, if the camera has been locked to place but there are objects moving in the area surrounding the wire.

• **Temporal With Moving Scene** - also aligns frames from before and after onto the current frame, but uses global motion estimation. Again, it gets background information from these two frames where it can and uses the spatial repair method to fill in the rest. This is useful for sequences where the wire is moving and the motion in the rest of the scene is fairly uniform, for example, if the entire scene is moving in the same direction as a result of a camera pan.

• **Clean Plate** - choose this method if you have a clean plate you wish to use for the repair, or if F_WireRemoval does not do a good job of removing the wire from each frame. Connect the clean plate or single frame with the wire removed to the **CleanPlate** input. F_WireRemoval will then align this frame to the frame to be repaired in order to reconstruct the background behind the wire.

**Filter Size** - this is a trade off between the amount of grain removed and the blurring of image along the direction of the wire. If the wire you are trying to remove has detail within it (for example, a steel wire in which the twisted threads are reflecting light), then the algorithm may leave these alone, thinking that they are grain. In this situation, you should decrease the filter size. A value of zero will definitely remove the artifacts but also the grain, which would then have to be put back using FurnaceCore's F_ReGrain.

**Temporal Offset** - the time offset of the additional frames to use for the **Temporal With Static Scene** or **Temporal With Moving Scene** methods. This determines which two frames before and after the current frame are used to fill in the background behind the wire. For example, if this is set to 2 and the current frame is 30, frames 28 and 32 are used.

**Luminance Correct** - turn this on for methods other than Spatial repair where there are global luminance shifts between one frame of the sequence and the next, or between a frame of the sequence and a clean plate you are using for the repair. With **Luminance Correct** turned on, F_WireRemoval will adjust the luminance of the background pixels it uses to the correct level before doing the repair.

**Lum Block Size** - change this value if luminance correction is not working as well as it should. The luminance correction uses overlapping blocks and matches the luminance within those blocks; changing the size of the blocks will change the regions covered by each block and could result in a better match.
**Points** - Parameters to set the position of the points used to define the wire.

**Point 1** - the position of the start point on the wire.

**Point 2** - the position of the point on the wire between the start point and the mid point (this is only active if **Type** is **Five Points**).

**Point 3** - the position of the mid point on the wire.

**Point 4** - the position of the point on the wire between the mid point and the end point (this is only active if **Type** is **Five Points**).

**Point 5** - the position of the end point on the wire.

**Start Width** - the width of the wire at point 1 of the on-screen wire tool.

**End Width** - the width of the wire at point 5 of the on-screen wire tool. This allows you to make your repair region wider at one end than the other, which is useful for cases where there is motion blur on the wire.

**Overall Width** - increase this parameter to expand the width of the repair region along its entire length, and for all key frames.
Global Motion Estimation

FurnaceCore has three effects based on global motion estimation (GME).

Introduction

FurnaceCore effect based on global motion estimation calculate a four-corner pin, which finds the best fit of one image onto another, and then apply that pin. These effects differ in how that corner pin is calculated and to which image that pin is applied. This chapter describes the general idea behind these plug-ins; for more detailed information on each plug-in, please read their individual chapters.

These effects are:

• **F_Align** - which lines up two shots of the same scene, by finding a corner pin from each source clip frame to the corresponding reference clip frame. For example, you can use this to align two separate but similar steady cam shots of the same scene.

• **F_RigRemoval** - which removes unwanted objects (rigs) from image sequences without accurate rotoscoping or keying to produce a clean plate.

• **F_Steadiness** - which removes motion from a single clip, by calculating a pin that either locks all frames in the clip to a single reference frame, or smooths that motion out over a window of frames. For example, you can use this to remove camera shake.
What is Global Motion Estimation?

Global motion estimation (GME) is a technique that attempts to map one image onto another with a simple four-corner pin. This differs from local motion estimation (LME), which attempts to find where each individual pixel in the image is in the other image. GME is much cheaper to compute than LME, but gives you less information about the image. Nevertheless, it is still very powerful for a variety of applications.

Using the plug-ins’ parameters, you can tell the GME engine what type of motion to expect. This can be a combination of any of:

1. **translation** - which allows the four corners to translate by the same amount,
2. **rotation** - which allows the corners to rotate about their center,
3. **scale** - which allows the size of the area defined by the corners to change,
4. **perspective** - which allows the angles at the corners to change, so that the area defined by them is no longer a rectangle.

The more types of motion you allow, the more expensive the motion estimation becomes. For many scenes, rotation and translation are sufficient.

The GME effects have an accuracy control, which controls the amount of work Foundry’s GME engine does to calculate the global motion. Typically, the higher this is, the better the estimation, but the more expensive it is.

Limitations of GME

As stated above, global motion estimation simply calculates a four-corner pin to transform one image onto another. This means that GME can't be used to match two images where there is heavy parallax, very complicated foreground motion, changing objects, and so on.

The best way to think of what GME can do is that if you can do it with a four-corner pin, it can; if you can’t, it can’t. However, GME will take the pain out of hand matching pins frame by frame.
Global Motion Estimation Effects

F_Align and F_Steadiness work in a similar way, which is distinct from the way F_RigRemoval works. These two effects calculate a four-corner pin for each frame and save it into the corner pin parameters. These pins are then used during the render to move the source image.

Using Them

These effects analyze images over a range of frames to figure out their four-corner pins. This is done in response to the user pressing the Analyse button in the effects control panel. During analysis, the effect will run through a range of frames adding keys to the corner pin parameters. These corner pins are then applied to the source clip to render a frame.

F_Steadiness has a separate analysis pass that happens interactively; it then uses the previously computed and key-framed corner pins during render. This speeds up its operation, as the analysis step only has to be done once, and once it has been done this plug-in is very quick to render. However, F_Align, which only ever needs the two current frames from each clip, can compute the corner pin on the fly (but not keyframe it!). This leads to a slightly different mode of operation for the following effects:

• F_Steadiness
  • needs to have an analysis run before it renders useful output,
  • will always use the value in the corner pin parameters when rendering the output image.
• F_Align
  • no need to have the analysis run to render output, but doing so will give you a key-framed corner pin,
  • during render, it will use the value of the corner pin parameters only if there is a keyframe there, otherwise it will analyse on the fly during render. This means that analysis will speed up later renders, as just rendering the corner pin is much cheaper than calculating it.

Some parameters to the effect control how the effect performs GME analysis, and some only affect the rendering. If you ever modify one of these parameters, then any analysis you may have performed will be out of date. To let you know, an overlay warning will be posted whenever this happens. You don’t have to re-analyze and your renders will still look at the keyed corner pins.

If you have not modified a parameter that affects analysis (the warning overlay will let you know), pressing Analyse will only re-analyse a frame if there is no key on the corner pin at that time. This avoids redundant re-analysis if you have interrupted the analysis or extended the analysis range. However, if you want to force re-analysis, press Clear Analysis and all keys will be deleted.
F_Steadiness and F_Align have an analysis region rectangle parameter which is used to specify which area of the reference image should be analysed during GME. So, for example, with F_Steadiness set to **Lock Mode**, this is the area inside the lock frame that a match will be sought for. The documentation for each plug-in describes exactly how to use the analysis region.
Controls

The controls common to all GME plug-ins are described below. They are grouped into two sections:

- Ones that determine how analysis is carried out. See Parameters That Affect Analysis.
- Ones that control the rendering of the output. See Parameters That Affect Rendering.

Parameters That Affect Analysis

The following parameters affect the analysis of the four-corner pin.

**Analyse** - This is a push button which triggers an analysis of the input clips and calculate a corner pin. Interrupting the analysis does not delete the corner pin keys that have already been calculated.

**Render During Analysis** - If set, this toggle causes the effect to update the time line and render a freshly analyzed frame so you can see the progress of the effect. Doing so slows down the analysis somewhat, so toggle this off to speed up the general analysis.

**Clear Analysis** - Pressing this push button deletes all keyframes from the corner pin parameters, allowing you to force a re-analysis if you feel the need to.

**Analysis Range** - This controls the range of frames any analysis will be run over. It can be one of:

- **Specified Range** - which looks at the parameters **Analysis Start** and **Analysis Stop** for the range of frames to analyze,
- **Source Clip Range** - which automatically determines the range of frames to analyze from the length of the input clip.

**Analysis Start** - The first frame to analyze from if **Analysis Range** is set to Specified Range.

**Analysis Stop** - The last frame to analyze from if **Analysis Range** is set to Specified Range.

**Scale** - A toggle that indicates whether the calculated corner pin can include a scaling factor.

**Rotate** - A toggle that indicates whether the calculated corner pin can include rotations.

**Translate** - A toggle that indicates whether the calculated corner pin can include translations in x and y.

**Perspective** - A toggle that indicates whether the calculated corner pin can include perspective transforms.
**Analysis Region** - This is the region analyzed to calculate the four-corner pin. This is especially useful when doing any form of frame locking, in which case, go to the lock frame, look at the reference clip and position the box over the area you want locked.

- **Analysis Region BL** - controls the position of the bottom-left corner of the analysis region.
- **Analysis Region TR** - controls the position of the top-right corner of the analysis region.

**Accuracy** - This controls the time/accuracy trade off in the GME engine. The higher this is, the slower it goes, but you have a better likelihood of a good result.

### Parameters That Affect Rendering

These following parameters control how a GME effect renders the four-corner pin. Some of them are set during the analysis pass.

**Filtering** - This controls the quality of the rendering.
- **Low** - uses nearest neighbor filtering. This gives low quality but is quick to render.
- **Medium** - uses a bilinear filter. This gives good results and is quicker to render than high filtering.
- **High** - uses a sinc filter to interpolate pixels giving a sharper repair. This gives the best results but takes longer to process.

**Invert** - If set, then the inverse of the calculated four-corner pin is used during render.

**Four Corner Pin** - The corner pins calculated during the analysis pass:
- **Bottom Left** - the lower left corner pin.
- **Bottom Right** - the lower right corner pin.
- **Top Left** - the upper left corner pin.
- **Top Right** - the upper right corner pin.
Widgets

All the Analyzing GME effects have two on-screen widgets: one to provide feedback and one to set up the analysis region.

**Analysis Region Widget** - This is a rectangle widget which you use to set the analysis region over the reference image.

**Four Corner Widget** - This is a widget that shows the state of the four-corner pin that has been calculated. You can change it by grabbing any of the corners and tweaking the shape of the pin. To give you more feedback as to what frames have been analyzed, it will be drawn solid if there is a key in the corner pin at the frame being displayed; otherwise, it will be drawn dashed.
Using F_Align

This chapter looks at how to use F_Align to register (line up) two shots that are of the same scene, but have slightly different camera motion and foreground objects. This can be useful, for example, for doubling up the crowd size by lining up and comping together two shots of the same scene, or locking your freshly generated clean plate to the original.

Introduction

F_Align takes two sequences that were shot of the same scene and lines them up spatially. It uses Global Motion Estimation (GME) to calculate a four-corner pin so that each frame in one shot (the source input) will be aligned with the corresponding frame in a second shot (the reference input). The result is the source image which has been repositioned to line up with the reference image.

To be able to align the sequences, F_Align analyzes them for global motion. This analysis can be triggered for the complete sequence, specified frame range, or a single frame when you press the Analyse button in the F_Align controls. Alternatively, it can be done on the fly for a single frame when
you move to a new frame on the timeline. The advantage of pressing Analyse is that during the analysis, F_Align stores the calculated four-corner pin as key-framed parameters. When you then render the output of the plug-in later, F_Align can use these key frames without having to calculate them again.

If you analyze on the fly, you won’t have direct access to the calculated corner pins. Any re-rendering will also be significantly slower, as the ‘on the fly’ calculations will have been lost and F_Align will have to analyze again.

If at any stage you modify the effect in such a way to invalidate the key-framed analysis (for example by changing the Accuracy parameter), a warning will be posted and the effect will analyze on-the-fly during render, ignoring the keyed analysis.

The on-screen widget and the Analysis Region parameters are used to control which section of the Reference frame is being matched to each Source frame. Typically, leaving the region at its default is good enough. However, a heavy mismatch in foreground detail may make it necessary to change the region to a section that is shared between shots.

The transformation in F_Align is concatenated with other NukeX transform nodes. This means that if you add a row of F_Align and NukeX transform nodes to a tree, their functions are combined. Because the image is only re-sampled once, there is no loss of image quality and processing time is decreased. However, as in NukeX, certain nodes, including color correction nodes, can break the concatenation.

For more of an overview of Global Motion Effects and a description of the common way of working many of these effects have, please see Global Motion Estimation.
Quick Start

This section gives a very brief outline of how to use the plug-in. It covers both analyzing using the Analyse button and analyzing on the fly.

Analyzing with the Analyse button

To align two shots and store the results as keyframed parameters, do the following:

1. Find two shots that are of the same scene, but have slightly different camera motion and foreground objects. Select Image > Read to load both these shots.

2. Select FurnaceCore > F_Align. Connect the shot you want to reposition to the source (Src) input of F_Align and the shot you want to match to the reference (Ref) input. View the output from F_Align.

   The source will be immediately repositioned so that it aligns to the reference shot without any need for analysis.

   You will see the following banner in the overlay:

   No valid key framed analysis found. Analyzing during render.

3. Depending on the exact difference between the two shots, you may need to enable the Scale and/or the Perspective toggles to get a decent alignment.

4. You may also need to reposition the Analysis Region depending on the differences in foreground detail. However, leaving it at the default position works well for most shots.

5. Click on the Analyse button.

   F_Align will now start analyzing each frame in the shot, figuring out the four-corner pin and writing it as key frames to the corner pin parameters, Bottom Left, Bottom Right, Top Left, and Top Right. You will find these parameters under Advanced > Four Corner Pin.

   F_Align will update the timeline at each frame, and you will see the aligned image render in the output. If you don't want this to happen, uncheck Render During Analysis.

   If you interrupt the analysis, the pins it has keyed until that point will be retained.

6. Play or scrub through the aligned frames. The rendering will be faster as F_Align will no longer need to analyze on the fly. However, if you scrub to a frame where a corner pin has not been keyed, F_Align will re-analyze that frame on the fly.

7. To see how closely the two clips have aligned, press M on the Node Graph to insert a Merge node. Connect the Merge node’s A input to F_Align and the B input to the reference image. View the output from the Merge node. Then, adjust the mix slider in the Merge controls to dissolve between F_Align’s output and the reference clip you wanted to match.
Analyzing On The Fly

To align two shots and calculate the alignment on the fly, do the following:

1. Find two shots that are of the same scene, but have slightly different camera motion and foreground objects. Select Image > Read to load both these shots.
2. Select FurnaceCore > F_Algm.
3. Connect one of the shots to the source (Src) input of F_Algm and the other to the reference (Ref) input. View the output from F_Algm.
   
   The Src will be immediately repositioned so that it aligns to the Reference shot without any need for analysis.
   
   You will see the following banner in the overlay:
   
   *No valid key framed analysis found. Analyzing during render.*

4. Depending on the exact difference between the two shots, you may need to enable the Scale and/or the Perspective toggles to get a decent alignment.
5. You may also need to reposition the Analysis Region depending on the differences in foreground detail. However, leaving it at the default position works well for most shots.
6. To see how closely the two clips have aligned, press M on the Node Graph to insert a Merge node. Connect the Merge node's A input to F_Algm and the B input to the reference image. View the output from the Merge node. Then, adjust the mix slider in the Merge controls to dissolve between F_Algm's output and the reference clip you wanted to match.
Parameters

The parameters for this plug-in are described below.

**Analyse** - This is a push button which will trigger an analysis of the input clips and calculate a corner pin. Interrupting the analysis will not delete the corner pin keys that have already been calculated.

**Render During Analysis** - If set, this toggle will cause the effect to update the time line and render a freshly analyzed frame in the Viewer so you can see the progress of the effect. Doing so will slow down the analysis somewhat, so toggle this off to speed up the general analysis.

**Clear Analysis** - Pressing this push button will delete all keyframes from the corner pin parameters, allowing you to force a re-analysis if you feel the need to.

**Analysis Range** - This controls the range of frames any analysis will be run over. It can be one of:

- **Specified Range** - which will look at the parameters **Analysis Start** and **Analysis Stop** for the range of frames to analyze,
- **Source Clip Range** - which will automatically determine the range of frames to analyze from the length of the input clip.
- **Current Frame** - the analysis occurs only on the current frame. This is useful for correcting any errors that may have occurred while analyzing the entire clip.

**Analysis Start** - The first frame to analyze from if **Analysis Range** is set to **Specified Range**.

**Analysis Stop** - The last frame to analyze from if **Analysis Range** is set to **Specified Range**.

**Scale** - A toggle that indicates whether the calculated corner pin can include a scaling factor.

**Rotate** - A toggle that indicates whether the calculated corner pin can include rotations.

**Translate** - A toggle that indicates whether the calculated corner pin can include translations in x and y.

**Perspective** - A toggle that indicates whether the calculated corner pin can include perspective transforms.

**Analysis Region** - This is the region analyzed to calculate the four-corner pin. This is especially useful when doing any form of frame locking, in which case, go to the lock frame, look at the reference clip and position the box over the area you want locked.

- **Analysis Region BL** - controls the position of the bottom left corner of the analysis region.
- **Analysis Region TR** - controls the position of the top right corner of the analysis region.
**Advanced** - The lesser used refinement controls.

**Accuracy** - This controls the time/accuracy trade off. The higher this is, the slower the analysis, but you have a better likelihood of a good result.

**Filtering** - Sets the filtering quality.
- **Low** - low quality but quick to render.
- **Medium** - uses a bilinear filter. This gives good results and is quicker to render than high filtering.
- **High** - uses a sinc filter to interpolate pixels giving a sharper repair. This gives the best results but takes longer to process.

**Invert** - if set, then the inverse of the calculated four-corner pin is used during render.

**Four Corner Pin** - The corner pins calculated during the analysis pass:
- **Bottom Left** - the lower left corner pin.
- **Bottom Right** - the lower right corner pin.
- **Top Left** - the upper left corner pin.
- **Top Right** - the upper right corner pin.
Using F_RigRemoval

This chapter looks at the removal of unwanted objects (rigs) from image sequences without accurate rotoscoping or keying to produce a clean plate.

Introduction

In this context, we define a rig as a foreground element in a sequence that moves over a background element. The plug-in will only work satisfactorily if it is possible to model the background motion by a global 3D transform. For example, if the background contains multiple objects moving in different directions, the results will be poor. Typically, good results will only be achieved in situations where a skilled artist could generate, and track in, a clean plate in order to repair the sequence. However, this plug-in should make the process quicker and easier. The rig removal algorithm works by estimating the background motion between successive frames, ignoring the foreground object, and then using the motion information to look forward and backward in the sequence in order to find the correct piece of background to fill in the missing region. The size of the missing region and the speed of the background dictate how far away from the current frame it is necessary to search to find the correct information.

Before with bird.  After applying F_RigRemoval.

Before with taxi.  After applying F_RigRemoval.
Quick Start

To remove an unwanted object from an image sequence, do the following:

1. Select **Image > Read** to load the sequence with an unwanted object.
2. Select **FurnaceCore > F_RigRemoval** and connect your image sequence to F_RigRemoval’s **Src** input. View the output of F_RigRemoval.
3. On each frame, define the area that will be repaired. You can do this in the following three ways:
   - If the image sequence has an embedded alpha channel, then you can use that alpha to define the area to be repaired. To do so, set **Rig Region** to **Src Alpha**.
   - Using F_RigRemoval’s **RigMask** input, you can feed in a matte sequence to define the area to be repaired. To use this matte, set **Rig Region** to **Rig Mask Alpha** (or one of the other **Rig Mask** options).
   - You can use the on-screen rectangle to define the area to be repaired. To do so, set **Rig Region** to **Box**. Position the on-screen rectangle on top of the unwanted object. To key-frame the position to follow the object, select **Set key** from the animation menus next to **Rig Region BL** and **Rig Region TR** in the **Rig Region Box** parameter group. Move to a new frame and reposition the on-screen rectangle. A new key frame is set automatically. Repeat as necessary until the rectangle covers the object on every frame you want to remove the object from.

Whichever method you choose, the region does not need to be the exact foreground region, but just a rough outline. However, you should avoid making it unnecessarily large as this will increase rendering time.

4. Having defined the region to repair throughout the clip, set **Frame Range** to the number of frames that the plug-in needs to analyze forwards and backwards to find enough data to repair the sequence. On the first frame, this will be quite time consuming as the algorithm needs to estimate the motion between each pair of frames. Subsequent frames will be much quicker.

5. If it has not been possible to replace all the foreground pixels, either because **Frame Range** was set too low or the background information does not exist anywhere within the sequence, the pixels will be displayed in red. Try to adjust **Frame Range** until no red pixels are visible and then render the sequence.

Below, in the figure on the left, we are using a box to define the pixels to replace, and **Frame Range** is set to zero. Increasing this value, as shown in the figure on the right, gathers pixels from other frames and improves the result. To completely remove the red pixels, you’d need a **Frame Range** value of 5.
6. View the results.

Tip: F_RigRemoval is fairly slow to process, which can make the initial keyframing of the rectangular region frustrating.

Sometimes, the easiest way to adjust the region is to load up F_RigRemoval, and to view the source so the effect is not processed, but the parameters are visible. Then, animate the rectangular region over the foreground object you’re trying to remove throughout the sequence. When you’re happy with the region position, click back onto F_RigRemoval’s output and wait for it to update. Slowly increase the Frame Range parameter until the whole region is repaired, then check the output on other frames.

Note: F_RigRemoval uses frames from a wide range around the current frame. Feeding the output of one F_RigRemoval node into another will greatly increase the memory over head, as the second F_RigRemoval node will require the first F_RigRemoval node to calculate all its frames before passing the result on. Therefore, it is strongly recommended that you process out the first result and then use the processed result with the second F_RigRemoval node.
Occlusions

The algorithm used in F_RigRemoval is unable to differentiate between multiple foreground objects. If there is another foreground object in the sequence that moves through the background region that is being used in the repair, this second foreground object will also be cut up and used, resulting in an incorrect repair. To try and assist in these situations, it is possible to mark regions of the image as not to be used for repair by setting their alpha value to mid gray. This will ensure that spurious bits of other foreground objects do not appear in the repair.

In the figure below, we are trying to remove the woman in the center of the screen as she walks from left to right down the street. At this frame, a man walks in the opposite direction and her feet and his head overlap.

![Original shot.](image)

Below, the figure on the left shows the normal matte for the woman, and the figure on the right shows the result of using this in F_RigRemoval. Note that the man’s head interferes with the repair and the reconstruction of the pavement is particularly bad, probably due to the man becoming the dominant motion source.

![The normal matte.](image) ![The output.](image)

To fix this, we can adapt the matte to include a mid gray area over the man. This tells the rig removal algorithm to ignore that area in the repair. This matte is shown below in the figure on the left, and the result is shown in the figure on the right. Note that the repair on the pavement is improved, and the man is simply clipped rather than being used in the repair.
The matte with a mid-gray area.  

The output.
Parameters

The parameters for this plug-in are described below.

**Rig Region** - Defines the area to repair.

- **Box** - repair the area inside a rectangular box, controlled by the box parameters below or the on-screen box.
- **Src Alpha** - repair the region defined by the alpha of the source input.
- **Src Inverted Alpha** - repair the region defined by the inverted alpha of the source input.
- **RigMask Luminance** - repair the region defined by the luminance of the **Rig Mask** input.
- **RigMask Inverted Luminance** - repair the region defined by the inverted luminance of the **Rig Mask** input.
- **RigMask Alpha** - repair the region defined by the alpha of the **Rig Mask** input.
- **RigMask Inverted Alpha** - repair the region defined by the inverted alpha of the **Rig Mask** input.

**Frames Searched** - Sets whether to search forwards, backwards, or in both directions to find missing data.

- **Forward and Backward** - searches before and after the current frame.
- **Forward** - searches frames after the current frame.
- **Backward** - searches frames before the current frame.

**Frame Range** - Sets the number of frames the algorithm should look forwards and backwards in the sequence to find the missing data. If you are getting red pixels, then increase this value. See **Quick Start**.

**Frames Used in Range** - If **Frame Range** has to be set to a large number to make an effective repair, the rendering time can be prohibitive. **Frames Used in Range** can speed up the repair by not using every frame to fill the foreground region, effectively skipping frames. However, this may reduce the quality of the result.

- **All Frames** - use every frame in the searched frame range to construct the repair.
- **Half of Frames** - use every other frame in the searched frame range to construct the repair.
- **Quarter of Frames** - use every fourth frame in the searched frame range to construct the repair.
- **10% of Frames** - use every tenth frame in the searched frame range to construct the repair.
- **Max 25 Frames** - use no more than 25 frames from the searched frame range to construct the repair. This option can be useful if **Frame Range** has been set to a very large number.

**Max Rig Movement** - To avoid perspective changes, F_RigRemoval searches for the missing data inside an area immediately around the rig region. **Max Rig Movement** defines the width of this area (in pixels). Fast movement in the **Src** footage requires a higher value than slow movement. However,
because the area used for the repair may be from a different part of the image, high values can introduce perspective problems.

**Rig Region Box** - The rectangular area used to define the repair when **Rig Region** is set to **Box**.

**Rig Region BL** - controls the position of the bottom left corner of the rig region.

**Rig Region TR** - controls the position of the top right corner of the rig region.

**Advanced** - The lesser used refinement controls.

**Filtering** - Sets the filtering quality.

  - **Low** - low quality but quick to render.
  - **Medium** - uses a bilinear filter. This gives good results and is quicker to render than high filtering.
  - **High** - uses a sinc filter to interpolate pixels giving a sharper repair. This gives the best results but takes longer to process.

**Luminance Correct** - Switch this on to correct for luminance changes from information taken from other frames. This is particularly important if the lighting changes throughout the sequence.

**Perspective Correct** - Switch this on to correct for minor perspective changes.

**Overlap Correct** - The repair is built up using slices of information from other frames in the sequence. These slices can be overlapped and blended to give a more natural looking repair. This parameter controls how much the regions overlap. Increasing this parameter too much will degrade image sharpness.

**Repair Fail Marker Opacity** - Sets the level of transparency of the red pixels used to show where the repair has failed.

**Preserve Alpha** - Switch this on to preserve the original alpha channel. By default, this is switched off, and the alpha channel is set to white where the repair has failed and black everywhere else.
Using F_Steadiness

This chapter looks at how to stabilize a shot using F_Steadiness.

Introduction

F_Steadiness uses Global Motion Estimation (GME) to calculate a four-corner pin, so that camera motion within a single shot can be smoothed out over a range of frames or removed by locking to a specific frame.

F_Steadiness needs to analyze the input clip before it can render useful output. This analysis is done when you press Analyse in the F_Steadiness controls. During the analysis, F_Steadiness keyframes a four-corner pin which will stabilize the clip in subsequent renders. Without having performed an analysis pass, F_Steadiness will not do anything useful on render.

F_Steadiness can work in two ways. These are:

1. **Smooth** - A range of frames around each frame is analyzed for motion and an average of that motion used to calculate the corner pin. Use this to keep the overall camera motion, but to smooth out sharp bumps and kicks.
2. **Lock** - A lock frame is specified and steadiness attempts to register each individual frame to that lock frame. Use this to completely remove camera motion from the sequence.

In lock mode, each frame in the clip must share a substantial amount of the scene with the lock frame, so you can't lock each frame in a 360 degree pan to the first one. However, in smooth mode, because F_Steadiness is only working on a small window of frames around the current frame, you can use it in shots which change completely over time.

The analysis region is used to control which section of the reference frame is being matched to each source frame. In lock mode, the reference is the lock frame, so you should position the analysis region when looking at the lock frame. In smooth mode, it looks at the incremental differences between frames, in which case you should place the analysis region in the area you want to appear with smooth motion.

The transformation in F_Steadiness is concatenated with other NukeX transform nodes. This means that if you add a row of F_Steadiness and NukeX transform nodes to a tree, their functions are combined. Because the image is only resampled once, there is no loss of image quality and...
processing time is decreased. However, as in NukeX, certain nodes, including color correction nodes, can break the concatenation.

For an overview of Global Motion Estimation, and a description of the common way Global Motion Effects work, please see Global Motion Estimation.
Quick Start

This section gives a very brief outline of how to use the plug-in.

Smoothing Out Camera Motion

To keep the overall camera motion but to smooth out sharp bumps and kicks, do the following:

1. Find a shot that has some camera shake in it and select **Image > Read** to load it.
2. Select **FurnaceCore > F_Steadiness** to apply F_Steadiness, and view its output.
   - You will see the following banner in the overlay: *No valid key framed analysis found, please press Analyse*
3. In the **Advanced** F_Steadiness controls, make sure **Mode** has been set to **Smooth**.
4. Click on the **Analyse** button.
   - F_Steadiness will now start analyzing each frame in the shot, figuring out the smoothing corner pin and writing it as key frames to the corner pin parameters, **Bottom Left**, **Bottom Right**, **Top Left**, and **Top Right**.
   - After a brief pause (while F_Steadiness calculates the transforms in the frame range), F_Steadiness will update the timeline and you will see the steadied image render in the viewer.
   - If at any point you interrupt the analysis, the pins it has calculated until that point will be retained.
5. Play or scrub through the stabilized frames.
6. If you want to make the result smoother, increase the **Smoothing** parameter in the **Advanced** parameter group and the corner pin will be recalculated immediately to give a smoother shot. You don't need to re-analyze the sequence if you do this; as F_Steadiness has kept the raw inter-frame transforms cached away, all it needs to do is re-write the keys for the average smoothing pin.

Locking To A Frame

To completely remove camera motion from a sequence, do the following:

1. Find a shot that has camera shake but where all frames share scene information and select **Image > Read** to load it.
2. Select **FurnaceCore > F_Steadiness** to apply F_Steadiness, and view its output.
   - You will see the following banner in the overlay: *No valid key framed analysis found, please press Analyse.*
3. In the **Advanced** F_Steadiness controls, set **Mode** to **Incremental Lock**.
4. Choose a frame somewhere in the middle of the sequence that you want to lock to, and set the Lock Frame parameter to that frame.

5. Scrub back and forth through the shot and look for a region of the shot that is shared by all frames and doesn't change much (for example, avoid a region with people walking in front of it).

6. Whilst looking at the lock frame, position the on-screen widget for the Analysis Region over that region.

7. Hit Analyse.

   The effect will start to analyze, working first forward from the lock frame, then backwards from it, until all frames to be analyzed are done.

   The timeline will immediately update to give you feedback in the viewer.
Parameters

The parameters for this plug-in are described below.

**Analyse** - This is a push button which will trigger an analysis of the input clip and calculate a corner pin. Interrupting the analysis will not delete the corner pin keys that have already been calculated.

**Render During Analysis** - If set, this toggle will cause the effect to update the time line and render a freshly analyzed frame so you can see the progress of the effect. Doing so will slow down the analysis somewhat, so toggle this off to speed up the general analysis.

**Clear Analysis** - Pressing this push button will delete all key frames from the corner pin parameters, allowing you to force a re-analysis if you feel the need to.

**Analysis Range** - This controls the range of frames any analysis will be run over. It can be one of:

- **Specified Range** - which will look at the parameters **Analysis Start** and **Analysis Stop** for the range of frames to analyze,
- **Source Clip Range** - which will automatically determine the range of frames to analyze from the length of the input clip.

**Analysis Start** - The first frame to analyze from if **Analysis Range** is set to **Specified Range**.

**Analysis Stop** - The last frame to analyze from if **Analysis Range** is set to **Specified Range**.

**Scale** - A toggle that indicates whether the calculated corner pin can include a scaling factor.

**Rotate** - A toggle that indicates whether the calculated corner pin can include rotations.

**Translate** - A toggle that indicates whether the calculated corner pin can include translations in x and y.

**Perspective** - A toggle that indicates whether the calculated corner pin can include perspective transforms.

**Analysis Region** - This is the region analyzed to calculate the four-corner pin. This is especially useful when doing any form of frame locking, in which case, go to the lock frame, look at the reference clip and position the box over the area you want locked.

- **Analysis Region BL** - controls the position of the bottom left corner of the analysis region.
- **Analysis Region TR** - controls the position of the top right corner of the analysis region.

**Advanced** - the lesser used refinement controls.
**Mode** - This parameter controls whether F_Steadiness is smoothing the shot while keeping the overall camera motion, or locking the shot to a single frame to completely remove camera motion. It can be set to:

- **Incremental Lock** - in which case, it will calculate the pin that takes each frame to the lock frame. This calculates the pin by working from the lock frame out to each frame, calculating the GME between each frame incrementally and accumulating it to create the corner pin.
- **Absolute Lock** - this also calculates a pin that takes each frame to the lock frame. However, it does so by doing GME directly from the frame in question directly to the lock frame.
- **Smooth** - in which case, we are smoothing the shot for a range of frames described by the **Smoothing** parameter. This is the default value. It can be used to keep the overall camera motion, but to smooth out sharp bumps and kicks.

Incremental locks work better in some situations, whilst absolute locks work better in others. However, absolute locks, when they work, are often more accurate. If, for example, you want to get a lock between frame 0 and frame 100, try using absolute lock first. If absolute lock doesn't work, it can be because frame 0 and frame 100 are too different. In this case, incremental lock can help if there is a gradual spatial progression from frame 0 to frame 100.

**Smoothing** - This controls the range of frames to average motion over when **Mode** is set to **Smooth**. The default value is 10.

**Lock Frame** - This controls the frame which will be locked to when **Mode** is set to either of the lock modes. By default, this is set to 0. Note that what is frame 0 for F_Steadiness is frame 1 in NukeX. Therefore, if you are looking at frame 3 in the Viewer and want to use that frame as the lock frame, you need to enter 2 as the **Lock Frame** value.

**Accuracy** - This controls the time/accuracy trade off. The higher this is, the slower the analysis, but you have a better likelihood of a good result.

**Filtering** - Sets the filtering quality.

- **Low** - low quality but quick to render.
- **Medium** - uses a bilinear filter. This gives good results and is quicker to render than high filtering.
- **High** - uses a sinc filter to interpolate pixels giving a sharper repair. This gives the best results but takes longer to process.

**Invert** - If set, then the inverse of the calculated four-corner pin is used during render. This works best with the lock modes, and can be used to track static locked-off plates back into a shot.

**Auto Scale** - To smooth out or remove camera motion, F_Steadiness translates and rotates the frames in the source clip. This leaves black pixels around the image. The Auto Scale parameter lets you fill in the black gaps at the edges by scaling the output image up. A value of 1 (the default) scales the image up until no black is visible, whereas a value of 0 disables scaling and leaves the black edges
untouched. Auto Scale uses the minimum scale necessary to remove the black gaps at the edges to preserve as much detail as possible.

**Four Corner Pin** - The corner pins calculated during the analysis pass:

- **Bottom Left** - the lower left corner pin.
- **Bottom Right** - the lower right corner pin.
- **Top Left** - the upper left corner pin.
- **Top Right** - the upper right corner pin.
Using the BlinkScript Node

The BlinkScript node runs Foundry's Blink framework enabling you to write your code once and run it on any supported device. This is achieved through code translation, in which the Blink code is turned into specific code for each target device. Code is generated and compiled on-the-fly, allowing you to switch between devices at will.

About BlinkScript

BlinkScript runs a Blink "kernel" over every pixel in the output, where a Blink kernel is similar to a C++ class, but with some special parameter types and functions. Through translation, the code in the BlinkScript node can be turned into normal C++ or SIMD code for the CPU, or OpenCL for the GPU.

The Blink framework streamlines plug-in development workflow significantly, as you no longer have to exit Nuke to compile your code.

Note: GPU acceleration requires an NVIDIA GPU and drivers for CUDA 4.2 or above. To use the GPU when rendering from the command line, add --gpu to your command. If your computer enters sleep mode, the CUDA drivers cannot recover and you must restart Nuke to use GPU acceleration.

Note: Nuke supports AMD GPUs on late 2013 Mac Pro 6,1 and mid 2015 Mac Pro 11,5, running OS X 10.9.3 'Mavericks', or later.

Note: BlinkScript is disabled in the Non-commercial version of Nuke.

You can publish kernels in Group nodes which can then be saved as gizmos, if required. Published kernels can be encoded to protect your IP using BlinkScript's built-in kernel protection. Protected kernels are not readable when the published node is saved to a script.

Warning: BlinkScript is very flexible, as there are no restrictions on the code you can write within a kernel. As a result, code compiled from the Kernel Source can cause Nuke to crash, so please use caution!
For more information, see Nuke’s Help menu under Documentation > Guide to Writing Blink Kernels or navigate to https://learn.foundry.com/nuke/developers/110/BlinkKernels/.
Quick Start

To get started with BlinkScript:

1. Connect the BlinkScript node to your sequence or image. See Connecting the BlinkScript Node for more information.

2. Edit the default InvertKernel, load an example kernel, or write your own from scratch. See Loading, Editing, and Saving Kernels for more information.

3. Set the standard Kernel Parameters, those that are present in all kernels, such as Use GPU if available and format. See Setting Kernel Parameters for more information.

4. Set the available memory and height-per-tile limits to control performance. See Changing Performance Settings for more information.

5. Publish your kernel and create a gizmo, if required. See Publishing and Protecting Your Kernels for more information.
Connecting the BlinkScript Node

1. Select **Other > BlinkScript** to create a BlinkScript node in the Node Graph.
2. Connect the **src** input to your sequence or image.

   **Note:** BlinkScript supports as many inputs as you create in your kernel, but the default InvertKernel only has a **src** input.

3. Connect a Viewer to the output of the BlinkScript node.
   The Viewer displays your inverted image.
Loading, Editing, and Saving Kernels

Kernel management is taken care of on the first tab of the BlinkScript properties panel. Double-click the BlinkScript node in the Node Graph to display its properties.

**Note:** Loading, editing, and saving kernels is only available if you have a NukeX license.

### Loading Kernels

1. Enter the file-path in the **Kernel File** field or click the folder icon to browse to the kernel's location.

   **Tip:** BlinkScript kernels use the `.rpp` file extension.

2. Click **Load**.
   
The selected kernel is compiled and read into the **Kernel Source** field.
Editing Kernels

You can edit existing kernels or write your own from scratch by clicking **Clear** and entering code in the **Kernel Source** field. Bear in mind that you won’t see any results in the Viewer until you click **Recompile**.

**Tip: **You can change how text in the **Kernel Source** appears using the controls in the **Preferences > Panels > Script Editor** tab. After saving your preferences, close and then re-open the **Kernel Source** in the **Properties** panel to apply the changes.

The first line in a kernel is always a declaration, similar to a C++ class and derived from ImageComputationKernel, which describes a kernel used to produce an output image.

**Note: **BlinkScript in Nuke only works with ImageComputation kernels. It does not work with reduction or rolling kernels. Reduction and rolling kernels can, however, be used in the Blink API as part of an NDK plug-in written in C++.

In the case of the default InvertKernel:

```
kernel InvertKernel : ImageComputationKernel<eComponentWise>
```

Parameters for the kernel are declared in the **param** section, in the same way as you would declare member variables in a C++ class. If your kernel doesn’t require any parameters, you can omit this section.

For example, the InvertKernel has a single parameter, **multiply**:

```
param:

    float multiply; //This parameter is made available to the user.
```

When a kernel is compiled inside the BlinkScript node, controls are generated for each of the kernel's parameters and added to the **Kernel Parameters** tab. In the case of InvertKernel, the node has just one custom parameter, **Multiply**.

Kernel parameters can be C++ built-in types such as **float** or **int**. Vector and matrix parameters are also supported. The following table contains the control types that the Kernel Source can expose on the **Kernel Parameters** tab in the properties panel.
<table>
<thead>
<tr>
<th>Control Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bool_knob</td>
<td>Parameters of type <strong>bool</strong> are represented by a Bool_knob, or single checkbox.</td>
</tr>
<tr>
<td>Int_knob</td>
<td>Parameters of type <strong>int</strong> generate an Int_knob with a single numeric input box. Right-clicking in the input box displays a menu allowing you to animate the value.</td>
</tr>
<tr>
<td>MultiInt_knob</td>
<td>Parameters of type <strong>int2</strong>, <strong>int3</strong>, <strong>int4</strong>, or <strong>int[]</strong> generate a MultiInt_knob, with multiple numeric input boxes. As with the Int_knob, right-clicking in the input boxes displays a menu allowing you to animate the values.</td>
</tr>
<tr>
<td>Float_knob</td>
<td>Parameters of type <strong>float</strong> generate a Float_knob with a numeric input box, linear slider, and animation menu button. The slider ranges from 0 to twice the default value of the parameter, or 0 to 1 if no default value is set.</td>
</tr>
<tr>
<td>XY_knob</td>
<td>Parameters with two floating-point values, types float2 or float[2], are interpreted as positions in 2D space and generate an XY_knob with an adjustable handle in the Viewer.</td>
</tr>
<tr>
<td>XYZ_knob</td>
<td>Parameters with three floating-point values, types float3 or float[3], are interpreted as positions in 3D space and generate an XYZ_knob with an adjustable handle in the 3D Viewer.</td>
</tr>
<tr>
<td>AColor_knob</td>
<td>Parameters with four floating-point values, type float4 or float[4], are interpreted as colors with alpha, and generate an AColor_knob. This initially displays a single entry box and slider, a button to split to four entry boxes, a button to display the Nuke color wheel, and a swatch showing the current color with an eyedropper for sampling in the Viewer.</td>
</tr>
<tr>
<td>Array_knob</td>
<td>Parameters with nine floating-point values, including float3x3, are displayed as an Array_knob with a 3x3 grid. Parameters with sixteen floating-point values, including float4x4, are displayed as an Array_knob with a 4x4 grid.</td>
</tr>
<tr>
<td>MultiFloat_knob</td>
<td>Parameters with numbers of floating-point values not already listed above, such as float[5], generate a MultiFloat_knob with a numerical entry box for each value, a single linear slider, and an animation menu button. The slider ranges from 0 to 1.</td>
</tr>
</tbody>
</table>

The rest of the kernel is up to you, within certain guidelines listed in Nuke's **Help** menu under **Documentation > Guide to Writing Blink Kernels** or at [https://learn.foundry.com/nuke/developers/110/BlinkKernels/](https://learn.foundry.com/nuke/developers/110/BlinkKernels/). There are also several example kernels to get you started.
Saving Kernels

1. Enter the file path in the **Kernel File** field or click the folder icon to browse to the intended location.

   **Tip:** BlinkScript kernels use the **.rpp** file extension.

2. Click **Save**.
   The selected kernel is saved to the specified location. There is no compile step during saving as the kernel is compiled when loaded.
Setting Kernel Parameters

Some BlinkScript controls are used by all kernels. Rather than write these individually in each instance, the BlinkScript properties panel includes **Kernel Parameters** and **Settings** tabs, providing easy access to the standard kernel controls.

The **Kernel Parameters** tab also contains any controls exposed by specific kernel commands. For example, the default **InvertKernel** exposes the **Multiply** control. See **Editing Kernels** for more information on exposing controls.

The **Publish** button also resides on the **Kernel Parameters** tab, but we'll get into that in more detail in **Publishing and Protecting Your Kernels**.

**GPU or CPU?**

The read-only Local GPU field displays the graphics card installed locally, if there is one available. You can toggle the GPU on and off using the **Use GPU if available** control. If you have multiple GPUs available, select the required device using the **Preferences > Performance > Hardware > default blink device** dropdown.

**Note:** Selecting a different GPU requires you to restart Nuke before the change takes effect.

**Tip:** Even if there is no GPU available locally, you can still enable **Use GPU if available**. BlinkScript then uses the GPU as and when one becomes available.

You should also enable this if you wish to render from the command line with the **--gpu option**.

Nuke's GPU support includes an **Enable multi-GPU support** option. When enabled in the preferences, GPU processing is shared between the available GPUs for extra processing speed. See **Windows**, **Mac OS X** and **macOS**, or **Linux** for more information on the GPUs Nuke supports.

Additionally, enable **Vectorize on CPU** to use SIMD acceleration on the CPU where possible. See **Help > Documentation** for more information on kernel execution.
Specifying the Output Format

BlinkScript's output defaults to the union of all its inputs, but you can specify a different output format by enabling Specify output format and selecting the required ratio from the format dropdown.

Tip: Enabling Specify output format also adds the format control to the kernel once it has been published. See Publishing and Protecting Your Kernels for more information.

Changing Performance Settings

The performance controls on the Settings tab are common to all Blink kernels and don't affect the output, but can improve the processing speed:

- **Percentage of GPU memory to use** - sets the upper limit on the amount of memory BlinkScript can allocate.

  Higher values than the default 50% can improve render times, but may introduce errors if the GPU is in use by other tasks.

- **Percentage of image height per tile** - processing is done in tiles, where each tile is the full width of the image. Adjusting the percentage of the image height processed in any one tile can affect performance.

  Higher values generally improve performance, providing that there is enough GPU memory available for processing large tiles, whereas lower values produce faster feedback to the Viewer.
Publishing and Protecting Your Kernels

When you're happy with your kernel, you can publish it for use elsewhere in the script or in an entirely separate Nuke script. Published kernels are wrapped in a Group node, so you can save them as gizmos if required.

Blink kernels can also be encoded using the built-in kernel protection. Protected kernels are not readable when the published node is saved to a script.

**Note:** Kernel protection offers a fairly low level of security and might not be sufficient to protect valuable source code.

Certain BlinkScript functions are only available with a NukeX license. The following table describes the various levels of access for the Nuke family.

<table>
<thead>
<tr>
<th>Product</th>
<th>Feature</th>
<th>Create node</th>
<th>Load, edit, and save</th>
<th>Adjust controls</th>
<th>Publish</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nuke Assist</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nuke</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>NukeX</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nuke Studio</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

To publish a kernel:

1. Double-click the BlinkScript node to open the properties panel and click the Kernel Parameters tab.
2. If you intend to encode your kernel, enable Protect kernel.
   In a Nuke script, protected kernels appear as shown in the example below.
3. Click **Publish**.

Nuke creates a Group node using the name of the kernel as a label. For example, using the WipeKernel supplied in the documentation produces the following expanded group.

If you want to save the group as a gizmo, go to the group's properties panel **Node** tab and then click **export as gizmo**. See **Creating and Accessing Gizmos** for more information.
Limitations and Known Issues

RGBA Only

Processing is currently limited to the RGBA channels only. To work with other channels, use a Shuffle node to shuffle them into RGBA before connecting to the BlinkScript node.

GPU

Complicated kernels can cause driver timeouts on the GPU if they take too long to execute. The lower the specification of your GPU, the more likely this is to happen.

Nuke supports AMD GPUs on late 2013 Mac Pro 6,1, mid 2015 MacBook Pro 11,5, and later 2016 MacBook Pro 13,3, running OS X 10.9.3 'Mavericks', or later. While, in some cases the bit-wise equality between GPU and CPU holds, for some operations there are limitations to the accuracy possible with this configuration.

Crashing and Infinite Loops

Since the BlinkScript node allows arbitrary code to be written and run, it is possible to either crash or lock up Nuke. The BlinkScript node does not check your code to make sure it is sensible before running it, so we advise you to use caution!

Known Issues

The following known issues exist in BlinkScript's current form:

**OpenCL on OS X 10.8 (Mountain Lion)**

Running OpenCL kernels on Mac OS X 10.8.2 and 10.8.3 can be unreliable. We recommend updating to 10.8.4 or later.
Written Tutorials

Welcome to the written tutorials!

Introduction

If you've gone through the Getting Started section - which we highly recommend - you already know something about Nuke. These tutorials show how to pull everything together through a series of practical examples.

You can also go through these tutorials using Nuke Non-commercial, but some features used in the tutorials may be disabled in the non-commercial version. For more information, see About Nuke Non-commercial.
The Projects

These tutorials include the following projects:

- **Tutorial 1: Compositing Basics** explains the Nuke user interface, project workflow, and basic compositing tasks.

- **Tutorial 2: 2D Point Tracking** demonstrates how to track image patterns, stabilize footage, lock down images for clean plates, and match-move.

- **Tutorial 3: Keying and Mattes** shows you how to pull mattes with standard keying tools and Nuke's own image-based keyer.

- **Tutorial 4: 3D Integration** shows how you can use Nuke's 3D workspace to help your 2D compositing.
Installing the Project Files

Before you continue, download the tutorial project files from our website and move them to a directory you'll create, called “Nuke_Tutorials”. It's up to you where you put your tutorial files, but here's our recommendations below depending on your operating system. Whatever you do, you'll need to remember where you put these files.

Tip: If you're using a Mac or Linux system, log in under administrator privileges to avoid issues with permissions when installing the files.

To Create the Tutorial Directory (Windows)
1. On the Windows desktop, double-click the My Computer icon to open a file browser.
2. Double-click on Local Disk (C:) and open the C:\Documents and Settings\All Users\Application Data folder.
3. Click the right-mouse button over the displayed directory and choose New > Folder.
4. Name the folder: Nuke_Tutorials.

To Create the Tutorial Directory (Mac or Linux)
1. Open a shell or terminal window.
2. At the command line, enter mkdir ~/Nuke_Tutorials to create the tutorial directory under your user or “home” directory.

To Download and Install the Project Files
1. Click Nuke Tutorials to download the project files to your local computer.
2. Extract the downloaded files and move (or copy) them to the Nuke_Tutorials directory you created earlier.
   You're now ready to start the first tutorial with Nuke.
Tutorial 1: Compositing Basics

Hello! This tutorial is your introduction to Nuke, where you’ll create a simple composite and breeze through most of the windows, on-screen controls, and other user interface items.

Introduction

We've heard rumors that many people would rather march through icy rain than review an introductory tutorial on digital compositing. Certainly, that's not you. When you finish this lesson you'll have a good understanding of the Nuke workflow and should feel confident about approaching the other tutorials.

Your first composite in Nuke.

Before you get into the project, we have some administrative things to do - such as defining a few application preferences and project settings. We know this sort of thing is not terribly exciting, but it is terribly important, so please be patient and we'll get through it as quickly as possible.

Note: If you haven't already downloaded and moved the tutorial project files to the Nuke_Tutorials directory you created, turn to Installing the Project Files for instructions.
Starting Nuke

The Nuke icon may appear on your desktop. If so, double-click it to launch the application. Otherwise, start Nuke with one of the methods described below, assuming you have installed Nuke to the default location.

To Launch Under Windows
• From the Start menu, choose All Programs > The Foundry, and then select Nuke11.3v6.

To Launch Under Mac
• Open the /Applications/Nuke/ folder and double-click the Nuke11.3v6 icon.

To Launch Under Linux
• Open the /usr/local/Nuke11.3v6/ folder and double-click the Nuke11.3v6 icon.

Tip: If you’re operating under Linux, you can also launch Nuke from the command line of a terminal. Simply navigate to the Nuke directory and enter the name of the Nuke application.

A clean copy of the main Nuke window appears. Divider lines organize the window into different panes. Each pane has one or more pages of content, separated by tabs at the top of the pane. The Toolbar appears at the left edge of the main window.
By default, the panes are setup to display the Viewer, the Node Graph/Curve Editor, and Properties. You'll create the script for this project inside the Node Graph page on the Node Graph/Curve Editor pane. We'll talk about each of these on-screen controls when you need them for the project.
Using the Toolbar

The Toolbar includes the options you can use to build your project, such as importing images, layering images, drawing shapes and masks, applying color correction, and so on. Each Toolbar icon displays a menu of operators or nodes that you can select. Roll the mouse pointer over the Toolbar and you'll see pop-up tool tips that identify each icon.
Using the Menus

The Nuke menu bar appears at the top of your screen, outside the main window. This menu begins with the options **File**, **Edit**, **Workspace**, and so on. When instructed to do so, make selections from the menu bar, or click the right mouse button to choose from a pop-up version of the menu bar.

The Nuke menu bar.

The “right-click” menu is highly contextual. Its options change according to the location of the mouse pointer. Right-click over the Node Graph, for example, and you’ll see the options from the menu bar and the nodes you can insert from the Toolbar. Right-click over the Viewer pane and you’ll see a menu of Viewer options.

The "right-click" menu.
Try the right-click menu when you can't find appropriate controls or menu options. Many features are hidden in the menu until you're in the situation where you need to use them.

**Note:** Nuke's menu bar, at the top of the screen, is organized a little differently between the operating systems, but the right-click menu contains the same options, regardless of the system you're using to run Nuke.
Customizing Your Workspace

Nuke gives you several options for customizing the window layout. It’s time for you to claim your copy of Nuke and make it your own! You don’t need to customize the layout for this lesson, but why not try it now for your own personal amusement? Here are some things you can do to reorganize the window layout:

• Drag a divider line between panes to change the size of the panes.

![Resizing a pane.](image)

• To divide a pane, click on the content menu (the checkered box at the upper-left corner of each pane), and choose **Split Vertical** or **Split Horizontal**.

![Splitting a pane.](image)

• To discard a pane, click on the content menu and choose **Close Pane**.

![Closing a pane.](image)

• To add a new tabbed page to a pane, click on the content menu and choose one of content options, such as **New Viewer** or **Curve Editor**.

• Click on the “x” inside a tab to discard a tabbed page.
Closing a tab.

• To move a tabbed page, drag the tab to another pane inside the main window.
• To tear-off a page as a floating window, drag the tab outside the borders of the main window, or simply Ctrl+click (Mac users Cmd+click) on the tab name.
• Drag a floating window into a pane, inside the main window, to convert it to a tabbed page.
• From the menu bar, choose **Workspace > Save Workspace** to save the current layout. Choose **Workspace > Restore Workspace x** to apply a previously-saved layout.
Saving Files and File Backup

We assume you already know how to save files (Hint: choose File > Save Comp As). In addition, Nuke includes an autosave feature, which helps recover project files after a system failure. Yes, we know that never happens to you, but in the unlikely event that it does, you won’t lose your work when you have autosave enabled.

Defining File/Saving Options

1. Click the right mouse button over the Node Graph pane, and choose Edit > Preferences.

   Notice the autosave filename directory is set to:
   [firstof [value root.name] [getenv NUKE_TEMP_DIR]/].autosave

   You don’t need to make a change; this simply tells Nuke to store automatic backup files in the same directories as your project files or the path supplied by the NUKE_TEMP_DIR environment variable (for more information on environment variables, see Configuring Nuke).

   Now, how often would you like Nuke to generate an automatic backup while you’re working? Every five minutes?

2. Change the force comp autosave after option to 300 seconds, to generate an automatic backup every five minutes.

3. Click Save Prefs to keep the changes and then Close to return to the main window.

   If you close this dialog box without clicking the Save button, then the changes affect only the current session of Nuke.
Recovering Back-Up Files

You may ask, “How do I recover a back-up file in the event of a system or power failure?” Good question! When you relaunch Nuke, you'll see a message that asks if you want to recover the .autosave file for the project that was last open. Click Yes and Nuke opens the back-up file.

Tip: The .autosave files can still be useful, even when you properly exit Nuke, because they are not deleted from the directory. You can, for example, rename an .autosave file to create an archive of the previous version of your project file.

Sometimes you may see the recovery message even though you have not experienced a system failure. This happens when you exit Nuke without saving the changes to a project file, and Nuke recognizes that the time stamp on the .autosave file is later than the Nuke project file you're trying to open. In this case, you decide which version of the project file you want to open.

Turning off Automatic Back-Up

Okay. You're reading this, so we assume you're a freewheeling rebel who possibly enjoys the risk of losing your work. It's an adrenaline thing. Or perhaps you prefer to do everything yourself, manually, and you have a secret obsession for saving your files. Whatever the reason, you can disable the autosave features by setting the intervals for both “autosave idle” and “force autosave” to zero seconds. That's all you need to do. Good luck.
Setting Up the Project

When you start a new project, you need to define project settings for length or frame range, the number of frames per second for playback, and the output format. These options appear on the Project Settings dialog box.

To Set Up Your Project

1. Click the right mouse button over the Node Graph, and then choose Edit > Project Settings.
2. In the **frame range** fields, enter a range of 1 to **28**. This is the length of the shot we create for the project.

3. Enter **24** as the frames per second (fps).

4. Click the **full size format** dropdown menu and choose **PC_Video 640 x 480**.

5. Close the **Project Settings** control panel.

**Note:** On the **Project Settings** control panel, the **Color** tab includes options that ensure color integrity for your display and output devices. You don't need to change the LUT for these tutorials, but we recommend that you research and set these options for your own projects.

Until now, everything you've done is standard procedure for a new project. You used the menu bar to access several features during the setup process, and now you'll use the Nuke toolbar to insert nodes and create a compositing tree.
Working with Nodes

A *node* is simply one of the building blocks for the list of operations you want to complete. A *node tree* is a diagram that shows the order in which the operations are performed. Do the following to add a few nodes and start your node tree. The result creates the background for the project.

Inserting Nodes

To insert nodes:

1. On the Toolbar, click the first icon to display a menu for nodes that are in the Images category.
2. Select **Constant** from the menu to insert this node into the Node Graph pane.

When you insert a new node, its control panel also displays with parameters that let you define what the node produces. In this case, the Constant node creates a solid color backdrop.

3. In the Constant control panel, click on the color wheel to open the **Color Picker**.
4. Drag the color sliders and the cursor inside the wheel to choose a light color, something appropriate for the “horizon” of the composite background. Then, close the color wheel window. At this point, you should probably rename “Constant” to something more descriptive.

5. Inside the control panel, click on the Constant name. You can now edit the name, so type **Background** and press Enter.

   From here onward, we’ll call this node the “Background” node.

6. Close the control panel for the Background node. When you need to reopen it, just double-click the node and the control panel reappears.

7. Click on the **Background** node to select it. Then, click the right mouse button and choose **Draw > Ramp**.
8. Drag the tail of the arrow from the **Viewer1** node to the center of the **Ramp1** node. You’ll see the output of the Background node and the ramp controls displayed in the Viewer window.

9. Click the **Color** tab inside the control panel for **Ramp1**. Then choose a dark color that blends well with the color you selected for the Background node.

10. Click the Ramp tab in the control panel to reactivate the overlay controls. Then, drag the **p0** and **p1** control points to adjust the spread and angle of the ramp over the background.
11. When you’re happy with the results, close the Ramp1 control panel to remove the overlay.
Connection Tips

Most nodes have input and output connectors that are used to establish the order in which the operations are calculated.

Connectors on a node.

Try the following to connect nodes after you insert them into the Node Graph:
• Drag an input or an output connector onto another node to establish a connection.
• Select a node, press the Shift key and select a second node. Then press Y to connect the first node to the output of the second node.
• Select a node, press the **Shift** key and select a second node. Then press **Shift+Y** to connect the second node to the output of the first node.

• Select a node and press **Ctrl/Cmd+Shift+X** to extract the selected node from the tree.

• For nodes that have two inputs, select the node and press **Shift+X** to swap the A/B inputs.
• Drag the mask connector to the node that provides the image you want to use as the mask for the selected node.
Importing Image Sequences

For this project, you need to import a few image sequences for the foreground elements and a background plate.

To Read the Images

1. Click on a blank space in the Node Graph. This ensures none of the nodes are selected.
2. Click the right mouse button over the Node Graph and choose Image > Read (or press R over the Nuke window).

**Tip:** Pressing R with an existing Read node selected, opens the file browser at the location specified by that node.

A file browser appears. This is where you select the image file you want to import. When you browse through your directories from this window, Nuke displays sequentially-numbered files as one item in the directory.

4. Add a bookmark to this directory. Right-click over the list, on the left side of the file browser window, and choose Add from the menu.
5. Type a name for the bookmark or keep the default, which is the directory name. Then click **OK**.

![Edit favorite dialog](image)

6. Open the `engine_rgb` directory, select the `engine.v01.####.exr` image sequence, and click **Open**.

![Image node](image)

Nuke retrieves the image sequence and displays it as a thumbnail on the node. The Read control panel displays the resolution and the frame range for the image.

**Note:** Nuke reads images from their native format, but the Read node outputs the result using a linear color space. If necessary, you can change the Colorspace option in the Read node's control panel, or insert a **Color > Colorspace** node to select the color scheme you want to output or calculate.

7. Drag a marquee (hold down the left mouse button while dragging) around the Background and Ramp nodes to select them. Then drag them to the right to make room for additional nodes.

![Node selection](image)
8. Choose **Image > Read** from the right-click menu to import another image sequence. Use the file browser to select the image sequence stored in **Nuke_Tutorials/CompBasics/smoke_left.wh/smoke_left.####.rgba**.

9. Add one more Read node and retrieve the image sequence stored in **Nuke_Tutorials/CompBasics/smoke_right.wh/smoke_right.####.rgba**

10. Arrange the nodes, as shown above, to allow some room to create the connections for the node tree.
Navigating Inside the Windows

The Node Graph panel can seem very small, especially when your node tree grows. True, you already know how to resize and tear-off the windows, but sooner or later you may run out of display real estate. It's time to learn some navigation controls that can help you work in the Node Graph (and other windows) in Nuke. Try the following navigation controls:

Panning Your View

• Windows/Linux: While pressing the **Alt** key and the left mouse button, drag the mouse pointer across the Node Graph.
• Mac: While pressing the **Option (alt)** key and the left mouse button, drag the mouse pointer across the Node Graph.

As you drag the mouse, you pan your view of the Node Graph.

Zooming or Magnifying Your View

• Windows/Linux: While pressing **Alt** and the middle mouse button, drag the mouse pointer across the Node Graph.
• Mac: While pressing **Option (alt)** and the middle mouse button, drag the mouse pointer across the Node Graph.

Drag to the right and you'll zoom-in. Drag to the left and you'll zoom-out.
• Keyboard zoom-in/out. Tap the plus (+) key to zoom-in. Tap the minus key (-) to zoom-out.

Using the Node Graph Overview

• When the node tree extends beyond the borders of the window, a navigation box appears in the lower-right corner of the Node Graph. Drag the shaded rectangle inside the box and you'll quickly pan to another view of the node tree.
Framing the View in the Window

- Press the letter F on your keyboard to fit the entire contents of the node tree within the borders of the Node Graph.

The navigation controls for the Node Graph also work inside the next window on our agenda, the Viewer.
Working with Viewers

The *postage stamps* on the nodes - those little pictures, often called thumbnails - show what each node passes onto the next node in the tree. Although quite lovely, they won't do for real compositing work. You need to open a Viewer window to see the full picture.

You can open several Viewers at once. In addition, you have up to 10 pages, or buffers, for each Viewer window; these allow you to toggle between different views along the node tree.

When you start Nuke, you see a default Viewer node in the Node Graph. You can easily drag the connection arrow from a node onto the Viewer to display the node's output. You can open additional Viewers by choosing **Viewer > Create New Viewer** from the menu bar or by pressing **Ctrl+I**.
Displaying the Images in a Viewer Window

To display the images in a Viewer window:

1. Drag the connector from the Viewer node onto the Read node for the engine.v01 clip.

   ![Diagram showing connector from Viewer node to Read node]

   Here's an alternate method: Select the engine.v01 clip node and then press 1 to connect to the Viewer node. Nuke displays the node's output in the Viewer window.

   ![Viewer window with red, green, and blue channels displayed]

2. Press the Alt key (Mac users press Option) and the left mouse button, and drag the mouse pointer across the Viewer window to pan.

3. Press Alt (Mac users press Option) and the middle mouse button, and drag to zoom in/out. You can also use the “zoom” dropdown menu at the top of the Viewer to magnify the view.

   ![Zoom settings and channels in Viewer window]

4. Press F to fit the current image into the borders of the Viewer window.
This image has different channels of information you can view. The “RGB” label appears at the top because the Viewer now shows the result of the red, green, and blue channels.

5. To view individual color channels, press R (red), G (green), B (blue) or A (alpha). As you press each keyboard shortcut, the label at the top of the Viewer reflects the displayed channel.

6. Press one of the channel keyboard shortcuts again to return to the “RGB” display, or choose RGB from the Viewer’s channel dropdown menu.

In addition to the standard color channels for red, green, blue, and alpha, this image also includes channels for specular highlights, reflections, and other masks.

7. To view additional channels, press A to display the alpha channel, and then select the lightingpasses.reflection channel from the Viewer channel dropdown menu.

You now see the reflection mask from the image file.

8. Select rgba.alpha from the Viewer channel dropdown menu to reset this as the preferred channel when you press the A key.

9. Press A again to toggle the display and show all color channels.
Viewing Multiple Inputs

To view multiple inputs:

1. Select the Read node for the smoke_left clip, and press 2 at the top of your keyboard or on the numeric key pad.
   
   This creates a second connection to the Viewer from the selected node. When the cursor is over the Viewer, you can press a number on the keyboard to pick the connection you want to view.

2. Move the mouse pointer over the Viewer and press 1 to display the engine.v01 clip. Press 2 to display the result of the smoke_left node.

   In this manner, you can connect multiple images to the same Viewer and then switch between the images.

3. Select each of the other nodes and press a number to establish a connection to the Viewer.

4. Move the mouse pointer over the Viewer and press the numbers on your keyboard to display each of the connected nodes.

   As you switch between the different views, the images may appear to be the same size. However, if you look in the lower-right corner of the Viewer, you’ll see the images have different resolutions.
These images have different resolutions.

Nuke allows multiple resolutions in one composite, but you need to conform these images to match the project resolution. This allows the elements to be properly aligned in the composite.
Reformatting Images

Elements created within the Nuke script, such as Background and Ramp, automatically inherit the global format and that’s how you want it for this project. The imported images, however, do not conform to the project settings and must be reformatted.

To Conform Images to the Project Format

1. Click the Read node for the engine.v01 clip to select it.
2. Click the right mouse button and choose Transform > Reformat.
3. Repeat steps 1 and 2 for all the Read nodes in the Node Graph.
4. Move the mouse pointer over the Viewer, and press the keyboard numbers (1, 2, and 3) to switch between the connected images.

   Each image should now conform to the project format.

If you change the delivery format in the project settings, then all elements set to “root.format” also change to the new project settings. If you neglect to reformat images when you read them into the project, the images retain their original format, independent of the project settings.
Using Proxies and “Down-res”

Proxies are low-resolution versions of the final image you intend to create. For many compositing tasks, the low-res version can help you work faster. Then, when you’re ready to create the final output, switch proxy mode off and return to the full-res version.

Nuke can generate proxies on-the-fly, according to the scale or format of your images. You select the method under Edit > Project Settings.

To toggle the proxy resolution defined under Project Settings, you use the “proxy” button on your Viewer. Alternatively, you use the “down-res” button to lower the display resolution of individual Viewers. The down-res button works both in the full-res and proxy mode.

To Activate Proxy Mode
1. Click the right mouse button over the Node Graph and choose Edit > Project Settings.
2. Make sure the Viewer window is open.
3. Press the keystroke to toggle Proxy mode, Ctrl+P.
   A label inside the Viewer indicates that you are now in proxy mode.
4. Move the mouse pointer over the Viewer, and press the plus (+) key several times to zoom-in.
5. Press Ctrl+P a few times to toggle between hi-res and proxy mode.
6. Before you continue, press Ctrl+P to switch back to full resolution.

To Activate “Down-Res”
1. Choose 1:4 from the “down-res” dropdown menu to change the display resolution to 25% of full resolution.
   With a reduced resolution, Nuke requires less time to calculate and display your images.
2. Change the “down-res” setting back to 1:1, which is 100% of the active resolution.
If you turned off proxy mode, you should be back to full resolution. If proxy mode is turned on, the display resolution is 100% of the proxy.
Compositing Images

The Merge nodes create a composite with two or more images, using various compositing algorithms. In this example, we'll do a very simple “A over B” composite to layer the foreground image over the background.

You can insert a compositing node from the Toolbar or menus, but we'll show you a keyboard shortcut that bypasses both of these. The trick is the select both nodes you want to composite and then press a keyboard shortcut to assign a compositing node.

To Composite Two Nodes

1. Select the Reformat1 node, attached to engine.v01. This provides the foreground image for the first compositing operation.
2. Press the Shift key and select the Ramp1 node. Both “engine.v01” and “Ramp1” nodes should be selected.
3. Press the letter M to insert a Merge node.
   The first node you selected is attached to the A input on the Merge node, as the foreground input. The second node you selected is attached to B, the background input. If necessary, you can swap the A and B inputs of a merge node by pressing Shift+X.

In the Merge node control panel, the operation parameter determines the compositing algorithm used to generate the result of the two inputs - the selected operation becomes the name of the node in the Node Graph.
4. Rearrange the nodes, so that the node tree looks similar to this:
5. For the next layer, select the **Reformat3** node, attached to **smoke_right**. Then hold down the **Shift** key and select **Ramp1**.

6. Press **M** to insert a Merge node and composite one image over the other. This composites the “**smoke_right**” image over the background.

7. The default compositing algorithm, “Over,” isn’t what we need here. In the Merge2 control panel, click on the **operation** dropdown menu and select **screen**.

8. In the Merge2 properties panel, drag the **mix** slider and change its value to **0.30** to reduce the amount of the image supplied by the **A** input.

9. An additional Merge node is required. Select **Reformat2** for **smoke_left**. Hold down the **Shift** key and select the **Over** node (the first Merge node you inserted).
10. Press **M** to composite the two nodes. In the Merge3 control panel, change the **mix** slider to **0.75**. The result of your composite should look similar to the example below.
Color-Correcting Images

Color-correction and filters can help you integrate the elements for a better composite. In our example, you want to limit the correction to the foreground element only, so you'll insert a color correction node before the Merge nodes.

1. Select the **Reformat1** node. Then, right-click over the Node Graph and choose **Color > Exposure**. This inserts the Exposure1 node.

2. Suppose you want to adjust the value of the red color channel. Move the mouse pointer over the Viewer window and press **R** to display the red channel.

3. In the Exposure1 control panel, uncheck the box for **gang** sliders. This allows you to adjust individual color channels.

4. Drag the **red** slider to adjust the color values. When you are finished, press **R** over the Viewer to display all channels.

The Exposure node worked as expected, but the result is less than spectacular. The color change is too uniform. If only there were a way to limit - or, in fact, **mask** - the color correction, perhaps we'd see a better composite. Hmm...
Masking Effects

You can apply masks to limit how each of these nodes affects the images. The following shows how to create a Bezier mask to limit color-correction.

To Create and Apply a Bezier Mask

To create and apply a Bezier mask:
1. Click on a blank space in the Node Graph, so that nothing is selected in the node tree.
2. From the Toolbar, choose Draw > Roto to insert a Roto node.
3. Click inside the Viewer window to draw a Bezier shape over the image, like this:

   ![Bezier mask example]

4. To refine the shape, click on a point to select it and then drag to adjust its position.
5. To create sharp corners, select a point, right-click and choose Cusp.
6. To add points to the shape, simply select the Add Points tool and click on the shape’s outline.
7. When you’re satisfied with the shape, drag the mask connector from the Exposure1 node to the output of the Roto node.

   In the Exposure1 control panel, the mask channel option is now set to the rgba.alpha channel of the node that is connected to the mask input. In this case, this is the alpha channel of the Roto node.
Creating Flipbook Previews

On the Viewer window, the timeline buttons let you play the project, but if you pay attention to the frames-per-second (FPS) field at the top of the Viewer window, you may notice that Nuke doesn't provide real-time playback. This is because Nuke renders on-the-fly to display images in the Viewer. It's fast, but also limited by the amount of memory and computer-processing power available to you.

The Flipbook feature provides better real-time preview, because it is prerendered for the Flipbook viewer. Keep in mind that the Flipbook feature renders a preview that matches the active resolution; if you're in proxy mode, for example, that's the resolution you'll get in the flipbook.

Note: The Flipbook feature renders temporary files in the directory you specified for disk cache under Nuke > Preferences. You'll also find an option there that allows you to limit the amount of disk space the flipbook feature uses.

To Generate a Flipbook
1. Select the Over node at the bottom of your node tree.
2. From the menu bar, choose Render > Flipbook selected.
3. Enter 1-28 as the number of frames to preview and click OK.
4. When the flipbook is ready to view, a Flipbook Viewer is launched. Click the Play button to view the results.
5. Close the Flipbook Viewer to return to your project.

Rendering Final Output

When you're ready to render the results of your composite, you insert a Write at the bottom of the node tree, and specify the path name for the rendered images. Although we'll use just one here, you can place several Write nodes in your script, anywhere you like, to render output from different places in the tree. When the render order is important, use the render order option in the Write nodes to specify the order in which multiple renders should be executed.

To Render the Result of Your Composite
1. Select the last Over node at the bottom of the node tree.
2. Right-click and choose Image > Write to add a node for output.
3. In the control panel for the Write node, click the file folder icon.


5. Click the “new folder” icon, in the upper-left corner of the browser, and type Rendered as the name for the new folder. Click OK.

6. Select the folder you just created.
   
   You should see the “Nuke_Tutorials/Rendered/” path name displayed at the bottom of the browser.

7. At the end of the “Nuke_Tutorials/Rendered” path name, type first_comp.###.exr as the name for the rendered image sequence, and then click Save.

8. Choose Render > Render All to render the images, or simply click the Render button inside the Write control panel.

9. Nuke prompts you to specify the frames to render. Enter 1-28 as the frame range and click OK.
A status window appears that shows the progress of your render. When the render is complete, you'll find the sequential images in the “Nuke_Tutorials/Rendered” directory. To check the results, simply insert a new Read node, point to the new image sequence, and then generate a flipbook with the Read node selected.

Using the Nuke Frame Number Variable

What’s that “####” bit in the filename, you say? That’s the variable that tells Nuke where to place the sequential numbers or frame numbers. You only type one name to represent the image sequence, but Nuke creates one image file for each frame in your shot.

So, in this case, you entered “first_comp.####.exr” but Nuke renders these files for frames 1 through 5: “first_comp.0001.exr,” “first_comp.0002.exr,” “first_comp.0003.exr,” “first_comp.0004.exr,” and “first_comp.0005.exr.” You can change the number of hash marks in the variable - ##, ###, ###### - to change the number of padded digits for the frame numbers.

An alternative way of marking frame numbers is the Printf (%d) notation. In this case, the same frame numbers would look like this: “first_comp.%0d.exr”. Instead of the number of hash marks, with the printf notation you would change the number before d to adjust the number of padded digits, for example “%03d” or “%05d”. You can choose which style you want to use by setting sequence display mode option on the Appearance tab of the Preferences.

Image Formats

If you don't specify a file format inside the Write node control panel, Nuke uses the format specified by the file name extension you typed. For example, in this tutorial, you used the “.exr” extension to tell Nuke to save the images as OpenEXR files.

Rendering with the Active Resolution

When you execute a render or a flipbook, Nuke assumes you want to render the active resolution. When you're in full-res mode, for example, Nuke renders full-resolution images to disk. When you're in proxy mode, Nuke assumes you want to render the proxy resolution - defined in the Project Settings window - to the path and file name you specified as the proxy file name in the Write node. If the proxy field is empty or pointing to an invalid path, Nuke returns an error.

It's easy to toggle to proxy mode and then forget your images are rendered in the lower resolution. Before you execute a render, it's always a good idea to check which resolution is active. In the Viewer, the label at the lower-right corner of your image indicates whether you are in full-res or proxy.
Rendering Multiple Channels

When you insert a Write node, Nuke assumes that you need only the RGB channels in the final render. In many cases, this is acceptable because you won’t need the alpha channel or other channels from the node tree when you deliver final shots to your clients. However, sometimes you need to render intermediate files - such as mattes, projection elements, or subcomps - and include all the channels in your node tree.

For example, rather than manage several elements for an animated character, you could combine the character animation, the lighting passes, alpha channel, and a depth mask in one image sequence on disk. This makes it easier to manage elements in the final composite and simplifies the artist's workflow.

To output all channels, change the Write node’s **channels** dropdown menu from **rgb** to **all**, select the OpenEXR file format, and then execute the render. Currently the OpenEXR format (.exr) is the only file format that supports unlimited channels.
Epilogue

In this tutorial, you set up a new project and created a simple composite. You learned how to use (or at least, locate) practically every Nuke window and tool, and you rendered out the result of your composite. You’re finished! Go home!

Well... there might be a few more things you want to know. After this tutorial, you should feel comfortable with the Nuke user interface, so put on your explorer hat and review the other tutorials. There’s no specific order from here, so look through the following pages until you find what interests you.
Tutorial 2: 2D Point Tracking

This tutorial teaches you how to use Nuke's Tracker node for tracking, stabilizing, and match-moving.

Introduction

Every filmmaker knows the challenges of putting together a vision. You may not have the money to build post-apocalyptic Montreal, but you might have enough to create it in post. You may have brilliant performances by your actors - but not together in the same shot. Fortunately, you can composite the best takes. Your battle sequence with 5 A-list actors, 100,000 extras and 57 elephants, comes back from the lab with scratches on the negative. You can fix it. You can. A savvy production team knows how to leverage digital technology to make it possible, and Nuke's tracking tools are indispensable for these situations.

As you may know, tracking is the process of recording the location of features as they move through the scene. The result is stored as 2D coordinates on the image plane. Once you have the tracking data, you can use the movement to perform a variety of useful tasks, such as stabilizing the footage, applying the movement to other elements in your composite, and improving the accuracy of roto mattes.

Tracking image features.

An important aspect of the tracking process involves carefully reviewing your footage before you attempt to track. Play through your sequences several times and look at the direction of movement for the features you want to track. Note potential problems with motion blur, obscuring objects, or frames where the features are hidden or move off screen.
Tip: Nuke can often compensate for problem footage, but tracking works best when you can identify distinct features throughout the length of the shot.
One-Point, Two-Point, Three-Point, Four

Before we get into the first example, let’s review a few tracking concepts. You can track as many features or patterns as required with the Tracker node in Nuke. How do you decide whether to track one, two, or more features? It depends on what you want to do with the data and the level of accuracy you need in the result. Here are some general guidelines:

- **One-point tracking** - Track one feature's horizontal (x-axis) and vertical (y-axis) position, with little or no perspective change on the image. You can apply this information to move other elements in the composite or apply the inverse to stabilize the image.

- **Two-point tracking** - Track horizontal and vertical position for two features. The feature positions, relative to each other, indicate whether the image is rotating clockwise or counter-clockwise (z-axis rotation). In some cases, two tracking points are sufficient to calculate the scaling of the features as well.

- **Three-point tracking** - Track horizontal and vertical position for three features. Provides all the benefits of two-point tracking with an additional set of tracking data for more accuracy on z-rotation and scaling.
• **Multi-point tracking** - Again, all the benefits of fewer tracks with additional sets of tracking data. Three-point is usually sufficient for most 2D tracking needs, but multi-point makes it possible to distort and match-move another element into the points of the features you track, for example, using four tracks and a CornerPin2D node.
Open the Tutorial Project File

In this tutorial, you work from a project file that already includes the node trees. Each tree is setup for the examples that follow.

To Open the Project File

1. Launch the Nuke application and choose File > Open Comp from the menu bar.
2. In the file browser, navigate to your Nuke_Tutorials/Tracking/ folder, select the tracking_tutor.nk project file and click Open.
3. It should show some nodes in error. Don't worry! It can't find the tutorial files. So before doing anything else you have to tell this script where to find these tutorial images. Double-click on the NoOp node in the top left corner of your Node Graph. It's called Tutorial_Path. Double clicking brings up a Properties panel on the right. Enter the path to the tutorial files in the Tutorial Project Directory. Use the file browser as that's often easier than typing it in. You should then see tutorial images appear.
4. Move the mouse pointer over the Node Graph, and press F to frame the entire contents of the project file.

The examples in this project file are grouped with colored boxes, called backdrops, and each contains a node tree for the tutorial examples that follow.

Tip: Backdrops let you organize groups of nodes, like those shown in this project file. Select Other > Backdrop from the Toolbar and drag the backdrop title bar to move it. Drag the backdrop corner to resize it. Any nodes surrounded by the borders of the backdrop move with the backdrop when you drag its title bar.
Tracking a Single Feature

In this first example you’ll learn how to set up a tracking anchor and then track a single feature, which is the most basic 2D tracking operation. After you achieve a solid track for one feature, you can build on that and track other features as needed.

Setting a Tracking Anchor

1. In the project workspace for the tracking_tutor.nk file, locate the node tree labeled Tracking an Image.
2. Click on the LondonEye Read node to select it.
3. Play through the sequence several times, using RAM cache or a Flipbook of your choosing, to review the footage.
4. Look at the features in the image and notice the amount and direction of movement as the clip plays.
5. Choose Transform > Tracker from the Toolbar to attach a new Tracker node to the LondonEye Read node.

6. Connect the Viewer node to the Tracker node, as shown.
7. In the Viewer, scrub the time slider to frame 1, to make sure you’re at the beginning of the shot.
8. Double-click the Tracker node to display its Properties panel.
9. Click add track to create a tracking anchor in the Viewer.
10. Drag the tracking anchor over the tower spire, as shown. Use the zoom window in the Viewer to help you with positioning.
11. Click on the pattern box (inner box) of the tracking anchor, and adjust its size to contain the feature.

12. Click the search area (outer box) of the tracking anchor, and adjust its size to enclose the amount of space you think the feature may move between frames.

   Large search areas require more calculation time, so keep it as small as possible. However, when the search area is too small, the feature may move outside the box and you'll lose the track. If you aren't sure how large to make the search area, go back and review the flipbook of your image.

### Auto-Tracking vs. Keyframe Tracking

After placing a tracking anchor, you're ready to calculate your track. Nuke's Tracker provides two calculation methods:

- **Automatic tracking** - ideal for simple tracks, there are no extra preparation steps once you've set your tracking anchors.
- **Keyframe tracking** - a more involved method, requiring you to set keyframes on the sequence in order to calculate tracks. Keyframe tracking may be the better option for more complicated patterns and movement.

### Using Auto-Tracking

1. Enable **show error on track paths** by clicking the traffic light icon in the Viewer tools.

   This color codes tracks showing their pattern matching error values, green for a good match through to red for a poor match.

2. In the Tracker Properties panel, select the track you wish to calculate in the **Tracks** list.

3. Select the type of movement the track is intended to output: translation, rotation, or scaling. In this simple example, you only need to select **Translation**.
4. At the top of the Viewer, click the track forward button to generate the track.

![Viewer screenshot](image)

If you only need a certain frame range, use the button and enter the required range. When Tracker has finished, you’ll see the track curve with color-coded points along the curve.

![Track curve](image)

**Note:** If tracking fails, try resizing the pattern or search boxes, as described in Setting a Tracking Anchor and retracking.

5. Use the next frame and previous frame buttons on the timeline to step through the track to verify its accuracy.

![Step buttons](image)

The track is fairly solid in this example. However, some images don't track as easily as this one. What are your options? You could retrack Using Keyframe Tracking as described below, or you could edit your existing track in the Curve Editor, see Editing Track Data on page 1884.
**Note:** Bear in mind that a red keyframe doesn’t necessarily mean that the tracking result is poor, only that Tracker couldn’t reliably match the pattern from one keyframe to the next.

**Using Keyframe Tracking**

1. In the Tracker Properties panel, select the track you wish to calculate in the **Tracks** list.
2. Select the type of movement the track is intended to output: translation, rotation, or scaling. In this simple example, you only need to select **Translation**.
3. Scrub through the sequence a few frames and adjust the position of the tracking anchor by dragging the anchor to the location of the pattern. You can use the zoom window to fine-tune your positioning. Continue on through the sequence as required.

At each frame, a new keyframe window is added to the right of the zoom window. The keyframe closest to the current playhead frame is highlighted in orange.

It’s a good idea to place more keyframes around areas of complexity or greater movement and fewer on straight forward translation. Generally speaking, a greater number of keyframes produces a better track, but at the expense of processing time.

4. When you’re satisfied with your keyframes, make sure your track is selected in the **Tracks** list and then click [ ] to track all keyframes.

   You can also force the selected tracks to recalculate between the two nearest keyframes by clicking [ ] in the Viewer toolbar.

5. Use the **next frame** and **previous frame** buttons on the timeline to step through the track to verify its accuracy.

**Note:** Bear in mind that a red keyframe doesn’t necessarily mean that the tracking result is poor, only that Tracker couldn’t reliably match the pattern from one keyframe to the next.

6. Proceed to **Editing Track Data** if the track is still not satisfactory.
Editing Track Data

1. In the Tracker Properties panel, select the tracks you want to view.
2. Click the animation button next to the Tracks list, and choose Curve editor.
3. Click the track_x and track_y items in the Curve Editor tree and you'll see values recorded for each of the parameters during the tracking process.
4. Select both curves by holding down the Shift key and clicking the track_x and track_y curves, under tracks.1.
5. Press the F key to “frame” the curves in the Curve Editor.
   • To adjust a value, select a point and drag it up or down.
   • To change the frame for a particular point, select it, hold down the Ctrl key and drag the point left or right.
6. Let's assume you want to smooth a curve by applying a filter to the values. Draw a marquee - drag while pressing the left mouse button - around a section of the curve to select multiple points.
7. Click the right mouse button to display a menu of Curve Editor options. Choose *Edit > Filter* and enter 2 as the number of times to filter the key frames.

Nuke averages the location of each point based on the values of the surrounding points, smoothing the curve.

8. Close the Curve Editor window and then play the result in the Viewer.

Those are the basics for tracking and editing the results for a single feature. In the next example, we'll make it a little harder - tracking a feature that moves out of view.
Tracking Obscured Features

At the end of the previous example, you may have noticed the track was dropped at frame 58 when the feature moved off the screen. When features move out of frame, or become obscured by other elements in the image, you can use the track offset feature to pass the tracking operation to another feature in the image. Nuke then attempts to continue the track along its current course.

To Track a Feature That Moves off Screen

**Note:** This example uses auto-tracking, but the offsetting principle is the same for keyframe tracking.

1. In the project workspace, locate the node tree Tracking Obscured Features.
2. Double-click the Tracker2 node to open its control panel. This node tracks one of the chimneys in the clip you used from the previous example.
3. Attach a Viewer to the Tracker2 node and scrub the timeline until you see the tracked feature move out of frame.

As you can see, track1 accurately tracks its feature through most of the clip - until it moves off the screen at frame 44. This is where the problem starts.

4. Press the plus key (+) on your keyboard a few times to zoom in on the Viewer. Examine the sequence to find an alternate feature that stays in view during the length of the clip.
5. At frame 44, press the Ctrl/Cmd key and drag the track1 anchor to the first chimney on the right of the building.
A line connects the new feature to the original feature indicating the offset, and the Tracks list is updated to show the x and y offset values.

6. In the Tracker control panel, press the button to continue the off screen track using the offset feature.

How is this useful? Well, now you can use the track data to matchmove an element - a trail of chimney smoke, for example - that locks to the feature even after it moves off the screen.

7. The track is now complete, so you can clear the offset by clicking in the Viewer tools.

8. Deselect track 1 in the Tracks list to prevent it from being recalculated.

9. Before you continue, close all Tracker control panels that are currently open.

The offset doesn't change the track location. Instead, it allows Nuke to continue the track with the assumption that the offset feature remains at the same relative distance to the original feature. Later in this chapter, you'll see how to use this tracking data to composite another element to match the background plate.
Stabilizing Elements

Stabilization is the process of removing motion - camera-shake, for example - and locking down the element for your composite. A one-point track provides enough information to stabilize horizontal and vertical motion along the image plane. A two-point track lets you stabilize horizontal and vertical motion, and remove rotation in the image, as well.

To Track and Stabilize

Note: This example uses auto-tracking, but the stabilizing principle is the same for keyframe tracking.

1. Locate the node tree labeled Stabilizing Elements.
2. You'll see a copy of the same LondonEye Read node that we've been using for the other examples. Click on it to select it.
3. Choose Transform > Tracker and then attach a Viewer to the new Tracker3 node. Double-click the node to open up the Properties panel.
4. In the Properties panel, click add track twice to create two tracking anchors.
5. For each track in the Tracks list, check the boxes for T (translate), R (rotate) and S (scale).
6. In the Viewer, scrub to the end of the sequence and adjust the size and position of each tracking marker for the features shown in the image below.

7. Select both tracks in the Tracks list.
8. At the top of the Viewer, click the track backward button to generate the tracks.
Now you have positional data for two tracks, and you can use this information to remove the unwanted movement in the image.

9. In the Tracker3 Properties panel, click the **Settings** tab and open up the Auto-Tracking sub-menu.
10. Set the **warp type** to **Translate/Rotate/Scale** so that Tracker expects all three transforms.

**Note:** If you’re using keyframe tracks, there’s no need to set the **warp type**.

11. Click the **Transform** tab and choose **stabilize** from the **transform** list.

**Note:** When you’re tracking simple translation using a single track, you can use **stabilize 1-pt** for faster calculation.

12. In the Viewer, click play to see the results.
As the clip plays, you'll see the features remain locked to the same position within the compositing frame.

**Tip:** After you track and stabilize footage, you can add a Transform > Transform node after the Tracker3 node to adjust the position and the rotation of the stabilized image for a final composite.
Match-Moving Elements

Match-moving is the opposite of stabilization. The intent is to record and use the motion in an image and apply it to another element. In the following example, you'll use the tracker to match-move and composite a mask image onto the performer in a background plate.

To Match-Move an Element

**Note:** This example uses auto-tracking, but the match-move principle is the same for keyframe tracking.

1. Find the node tree labeled **Matchmoving Elements**.
2. Drag the time slider to the beginning of the timeline. Select the **ColorCorrect1** node and then choose **Transform > Tracker**.

3. Attach a Viewer to the new **Tracker4** node, create a tracking anchor, and position the **track 1** anchor over the performer's right ear.

4. Adjust the size of the pattern box and the search area as shown.
5. In the Properties panel, check the boxes for **T** (translate), **R** (rotate) and **S** (scale) for **track 1**.
6. Adjust the size and position of the second tracking anchor, as shown below.
7. Create another tracking anchor, **track 2**, and check the boxes for **T** (translate), **R** (rotate) and **S** (scale) on this track.

8. Select both tracks in the **Tracks** list and click track forward at the top of the Viewer to generate the tracks. Edit the tracks as described in **Tracking a Single Feature** if necessary.

9. Once you have two solid tracks on the performer, make a copy of the Tracker4 node by selecting it and pressing **Ctrl+C** to copy it.

10. Select the **Transform1** node and press **Ctrl+V** to paste the Tracker node copy (Tracker5).

11. Connect the Viewer to the **Over** node. Your node tree should now look similar to this:

12. In the Tracker5 Properties panel, click the **Transform** tab and choose **match-move**. Then close the Tracker5 Properties panel.

13. Click play in the Viewer or render a flipbook and you should see the Mardi Gras mask transform to match the movement of the performer.

If you see jitter in the movement, you can edit the track data in the Curve Editor to smooth out the data. You can also add values to the smooth T, R, and S controls on the **Transform** tab to filter the tracks.
Epilogue

In this tutorial, you worked with several examples for the Tracker node. You learned how to record the locations for multiple features and you applied the tracking data for other tasks in the composite, such as stabilization and match-moving.
Tutorial 3: Keying and Mattes

This tutorial introduces you to keying in Nuke. You will learn how to use the Primatte and IBK nodes.

Introduction

Keying is one of those fundamental compositing skills. You can’t composite anything until you have mattes pulled for the elements you want to layer together. It’s nice to say you could just push a button to complete this task, but as you probably know, one keying operation seldom produces an acceptable matte. Image quality, lighting conditions, subject motion, colors - even camera moves - affect the steps required to get a clean matte for your composite.

So how do you get a clean matte in Nuke? The best approach is to understand the strengths of each keying tool and combine them as needed. This tutorial shows how to pull keys in Nuke and how to layer the results with channel operations, merge nodes, and roto shapes.
Open the Tutorial Project File

The project file for this tutorial includes several node trees for the keying operations described in this chapter.

To Open the Project File

1. Launch the Nuke application and choose File > Open Comp from the menu bar.
2. In the file browser, navigate to your Nuke_Tutorials/Keying/ folder, select the keying_tutor.nk project file and click Open.
3. Double-click on the Tutorial_Path node, located on the left side of the script, to open its control panel.

4. In the Tutorial_Path control panel, click the “file folder” button. Browse to the location where you installed the tutorial project files, and then click Open to select the location.

After you select the correct path, the error messages should clear from the Read nodes, and the thumbnails in the script update with the correct images.

5. Close the Tutorial_Path control panel. Then, choose File > Save Comp As to save a copy of the project file.
6. Move the mouse pointer over the Node Graph, and press F to frame the entire contents of the project file.
   The green arrows (lines) show the links between the Tutorial_Path node and the Read nodes.
7. If you wish, press Alt+E to hide the expression arrows.
   The Tutorial_Path node saves the location of the project files on your computer, so you don't need to repeat this for future sessions.
Keying with Primatte

The Primatte keyer includes a quick “Auto-Compute” option that evaluates your image and determines a good baseline key. From there, you can easily tweak the settings and generate an acceptable matte.

The two examples in this section show how to pull a key with the Auto-Compute option (method 1), and also how to manually sample a color from the screen background and build your key from there (method 2).

To Pull a Key with Primatte (Method 1)

1. In the project file, locate the node tree labeled “Keying with Primatte,” and make sure a Viewer is attached to the Reformat1 node.
2. Choose Keyer > Primatte to insert the keyer between the foreground image and the Viewer.
3. Drag the bg connector from Primatte1 to the Reformat2 node, which supplies the background image for this example. The fg connector should be attached to Reformat1.
4. Move the time slider to frame 50, and click the Auto-Compute button inside the Primatte1 control panel.
That's it. You're done... well, nearly done. We need a “free-floating” goldfish, but the reflections in the aquarium glass clearly indicate “captivity.”

A garbage matte easily removes the reflections, and you'll learn how to do that later in the section on rotoscoping. For now, let's keep working with Primatte.

As you've seen, Primatte's auto-compute option can quickly pull keys on certain images. However, you should also know how to pull and tweak keys manually. You might, for example, need more control over the transparency of the fins on the goldfish.

**To Pull a Key with Primatte (Method 2)**

1. Continuing from the previous example, open the Primatte1 control panel.
2. Click the “undo” button at the top of the control panel to step back to the previous state of the Primatte1 node. Or, you can also delete the current Primatte1 node and insert a new one.

3. Scroll down through the Primatte options and set the keying operation to Select BG Color.

4. The current color chip should display the eyedropper icon. If it doesn't, click on the color chip to toggle the eyedropper.

5. Hold down the Ctrl+Shift keys (Mac users, hold down Command+Shift) and drag - or scrub - over a portion of the greenscreen in the image displayed in the Viewer.
This returns an average color-pick of the sampled pixels. If you want a color pick from a single pixel, press Ctrl or Command and click once over the greenscreen. After you pick, you can clear the red square by Ctrl- or Command-clicking again.

6. Press A over the Viewer to toggle to the alpha channel display. Looks like the aquarium is not as clean as we thought. Our color pick gave us a fairly noisy key, so let's clean it up.

![Image of the viewer with an alpha channel]

Now you'll sample a few areas of the image to “push” selected pixels to one of three areas: the transparent matte, the opaque subject, or the semi-transparent part of the matte.

7. In the Primatte1 control panel, change the keying operation to Clean BG Noise.

![Image of the Primatte1 control panel with Clean BG Noise highlighted]

8. Press Ctrl+Shift or Command+Shift and drag a small square over the dark area in the lower-right corner of the image.

![Image of the viewer with a small red square over the dark area]

This second color sample cleans the background by “pushing” the selected pixels into the transparent area of the matte. You probably need a few more samples to get a better key.

![Image of the viewer with a better key]

9. Scrub a few small areas in the background, focusing on the gray pixels until the matte improves.
The background doesn’t need to be solid black. We’re just trying to get a good separation between our foreground subject and the greenscreen background.

10. Change the keying operation to **Clean FG Noise**. This time, sample areas of gray pixels inside the goldfish.
   One or two small samples should be enough. The color pick pushes the selected pixels to the opaque part of the matte.

   ![Image of goldfish with keying applied](image)

   You want to keep the gray pixels inside the fins to retain a semi-transparent matte in these areas. If you go too far, you can always press the undo button in the control panel to step back to the previous action.

11. Press **A** again over the Viewer to toggle to all color channels. Your image should look similar to the example shown below. You may see some detail dropping out from the fins.

   ![Image of goldfish with keying applied](image)

12. Change the keying operation to **Restore Detail**, and scrub over the fins to bring back some of the edge detail.
You may get different results than those shown here, depending on the pixel values you sample from the image.

Use Restore Detail to push the selected pixels back toward the opaque part of the matte. Use the **Make FG Transparent** operation to fine-tune the semi-transparent area.

You could go back and forth, between cleaning the background and foreground, but this usually produces a matte with “crunchy” edges. The goal is to find the balance between foreground and background that produces an acceptable matte for your subject.

Later in this chapter, you’ll use the rotooning tools to clean-up this matte and combine this with the image from the next example.
Image-Based Keying

Many keying tools, like Primatte, use a color-pick as the baseline for the matte extraction process and then require the artist to tweak the matte from that baseline. Nuke’s image-based keyer (IBK) uses the pixel values of the compositing images, instead of a color-pick, to generate the best matte for the image you want to extract. It works by generating a processed screen image that preserves the color variations of the blue- or greenscreen and using this - rather than a single color - to pull the key. This generally gives good results and speeds up the keying process when working with uneven blue- or greenscreens.

Image-based keying requires two nodes in Nuke. First, you insert an IBKColour node to process the screen image, which is preset to work with either greenscreen or bluescreen. This node generates the processed screen image that preserves the color variations in your blue- or greenscreen. Then, you insert an IBKGizmo node to generate the matte using the processed screen image, the original image, and also the background image for the composite.

To Pull a Key with IBK

1. In the keying_tutor.nk project file, locate the node tree labeled, “Image-based Keying”.
2. Right-click over the Reformat3 node and choose Keyer > IBKColour. Drag the IBKColourV3_1 node to the right.
3. Click an empty spot in the Node Graph to deselect all nodes. Then, right-click and choose Keyer > IBKGizmo.
4. From IBKGizmoV3_01 node, connect fg (foreground) to the Reformat3 node. Connect c (color screen) to the IBKColourV3_1 node.
5. Connect bg from IBKGizmoV3_1 to the Reformat4 node, which supplies the background for the comp.
6. Connect the Viewer to the IBKGizmoV3_1 node, and your node tree should look similar to this:

![Node Tree Diagram]

7. Open the control panel for IBKColourV3_1 and change the screen type to green.

![Control Panel]

8. Open the control panel for IBKGizmoV3_1, and change its screen type to C-green.

![Control Panel]

You should see an acceptable matte, shown in the screen capture below, on frame 50.

![Screen Capture]

This is a very good start for this image.
9. Connect the Viewer to the IBKColourV3_1 node. You'll see the processed screen image, which is essentially a Gaussian-filtered high-contrast key.
10. Choose **Merge** > **Merge** (or press **M** over the Node Graph) to insert a **Merge (over)** node.

11. Connect **IBKGizmoV3_1** to the **A** input of the **Merge (over)** node. Then connect the **B** input to the **Reformat4** node.

The color of this greenscreen is completely out of the region of the acceptable industry standard, but IBK does a good job anyway, by smoothing the screen and using the result to essentially create a difference matte with the foreground.

**Tip:** IBK has presets for green and blue screens, but you can also do a color-pick for any screen color inside the IBKGizmo node.

If you zoom-in on the image, you’ll see small areas near the subject’s hair, where the matte is compromised.
12. Connect the Viewer to **IBKColourV3_1** and you’ll see color artifacts around the edges of the matte.

![compromised matte edges](image1)

When you look at the smoothed screen produced by IBKColour, you should see only values of your screen color and black.

13. In the **IBKColourV3_1** control panel, lower the *darks, g* (green) setting to **-0.08**. (If you were keying a bluescreen image, you would lower the “b” value for “darks.”).

![IBKColourV3_1 control panel](image2)

This fixes most of the problems at the hairline, but destroys the good key for the lower portion of the image.

---

**Tip:** The artifacts in the IBKColour image appear as specific color shades: light green, dark green, light red, dark red, light blue, and dark blue. To remove these, simply adjust the appropriate controls for the artifacts you want to remove: *lights/g, darks/g, lights/r, darks/r, lights/b, and darks/b.*
14. In IBKColourV3_1, raise the darks, g value to -0.03. Then change the lights, g value to 0.75. This corrects the artifacts of the screen image.

15. Now, change the patch black setting to 1.5 to restore the edge detail of the hairline.

16. Connect the Viewer to the IBKGizmoV3_1 node. Press A and you'll see the current alpha channel produced by the IBK system.

The displayed alpha image shown is correct for the IBK. If the intensity of the noise in your alpha channel is greater than the example shown above, you may need to adjust - in very small increments - the dark and light values for the color channels in the IBKColour node.

17. Press A again over the Viewer to toggle back to display all color channels, and scrub through the timeline to check the matte at various points in the clip.

18. If you haven't already done so, save your project under a new file name to save the changes you've made to project.
Rotoscoping

In this example, we'll return to our first keying example to apply a garbage matte and clean-up the aquarium image.

To Draw a Garbage Matte

1. Go back to the node tree from the first example, and connect the Viewer to the Primatte1 node. Drag the time slider to frame 50.
2. Click an empty spot on the Node Graph to deselect all nodes. Then, right-click and choose Draw > RotoPaint.

3. At this point, you don’t need to connect the RotoPaint1 node to anything, but its control panel must be open, and the first tab, RotoPaint, should be active.

4. Inside the Viewer, you’ll see the goldfish image. Click the Bezier tool in the RotoPaint toolbar on the left side of the Viewer. Then in the Viewer, click four points around the goldfish to create a roto shape. You can drag+click to draw a point and adjust its curve at the same time.
**Tip:** As long as the RotoPaint1 control panel is open, you can view and edit the roto shape. You can press Q over the Viewer to toggle the display overlay, if necessary. Click the right mouse button over any point to select options for the roto shape.

Because this is a garbage mask, we want to edit the shape to remove elements from the glass aquarium.

5. Drag the points and adjust the tangents - the handles on each of the points - to refine the roto shape.

Now we need to animate the garbage matte to follow the motion of the fish.

6. In the RotoPaint tool settings panel, on top of the Viewer, the **autokey** option should be active. If not, click the box for this option.

7. Move the time slider to frame **1** and click the **Transform** tab in the RotoPaint control panel. Then select the entire Bezier shape in the stroke/shape list (at the bottom of the control panel) or by clicking one of the points in the shape using the **SelectAll** tool. A transform jack appears.

8. Drag the center point of the transform jack, and move it over the current position of the goldfish.

9. Go to end of the timeline, to frame **60**. Drag the shape once more to adjust for the movement of the goldfish.
If your Bezier shape is similar to the one shown above, then you probably don’t need more than the three key frames at frames 1, 50, and 60. However, you may want to scrub through the timeline and make adjustments.

10. Scrub to frame 60 on the timeline and you’ll see the roto gets a little close to corner-line that we want to remove from the aquarium glass.

11. Click on an empty spot in the Viewer to deselect all points. Then, press Ctrl/Cmd and click on the point near the goldfish’s nose, to temporarily break the point’s tangent handle.

12. Adjust the handles to create a peak at the fish’s nose.

Now, for good measure, let’s create a feathered edge for this particular point.

13. With the point selected, drag the feather handle away from the fish to create a feathered edge for this point, at this frame.

So you’ve drawn and animated the roto shape. Let’s wire it into the node tree to mask out the “garbage.”

14. Drag the bg connector off the Primatte1 node to disconnect it from the Reformat2 node.

15. Choose Merge > Merge from the right-click menu. Connect Primatte1 to the A input on the over node. Connect Reformat2 to the B input.
16. Connect the RotoPaint1 node to the mask connector on the over node. This effectively removes the aquarium reflections in the image.

![Image](image1.png)

You might want to scrub through the timeline to see if there are places where you need to adjust the roto shape.
If you want to take this a little further, you can now add the goldfish to the composite from the second example.

17. Select and drag the **Merge (over)** node and the RotoPaint1 node below the node tree you used for the IBK example.

![Image](image2.png)

18. Drag the **Viewer** node over, as well, and keep it connected to the **Merge (over)** node.

19. Drag the B connector from the **Reformat2** node and connect it to the over node in the IBK node tree.
The Viewer shows the result. Of course, you might want to add a **Transform** node after the first **Merge (over)** node, to size and position the goldfish. Otherwise, this project is completed.
Epilogue

Keying is rarely a simple matter of picking the screen color you want to remove. To get the very best mattes, you often need to combine several techniques and you've learned several in this chapter. You've pulled mattes with Primatte and Nuke's Image-based Keyer, and you've used the rotoscoping tools to cleanup a matte and control the parts of the image you want to use in the composite.
Tutorial 4: 3D Integration

This tutorial teaches you the basics of using Nuke's 3D workspace.

Introduction

Nuke's 3D workspace creates a powerful compositing environment within your project script. This workspace combines the advantages of cameras, lighting, and a three-axis (x, y, and z) environment, with the speed of node-based compositing. You pipe 2D images into the 3D space, setup a camera, animate your scene, and then render the results back to the 2D composite.

Compositing in the 3D workspace.
The Basic 3D System

The 3D workspace is defined by a group of nodes in your script. The most basic setup includes a Camera node, a Render node, a Scene or Geometry node, and nodes that provide the 2D images you want to pipe into the 3D compositing space.

![The basic 3D node tree: 2D image, geometry, scene, render, and camera.](image)

The 3D Viewer

Once you have the 3D node structure, you can use any Viewer in Nuke as a gateway to the 3D compositing space. Choose 3D from the view dropdown menu, or press the Tab key over the Viewer to toggle between the 2D and 3D views.

![The 3D Viewer](image)

On the view dropdown menu, you'll also see orthographic views - rtside, lfside, top, bottom, front, back - which provide non-perspective views into the scene. In three-quarter perspective, it can be difficult to accurately place objects on the axes, and the non-perspective views make it easier to line things up.
The Geometry or Scene Node

Every 3D system needs a piece of geometry - a card, a sphere, a cube, something - to receive an image or clip that the camera can “see.” One is all you need, but you can setup complex systems with a large amount of 3D data. When you have two or more objects for a 3D system, you need a Scene node to create a “place” where the camera (and the ScanlineRender node) can see all the objects at once.

The Camera Node

The Camera node creates your view into a scene. It has several controls to help you match the properties of a physical camera. You can animate its position or import animation or tracking data to matchmove your 3D scene with a background plate. A 3D system can have multiple cameras connected to the Scene node, to create different views on a 3D scene.

**Tip:** Only one camera can be connected to the ScanlineRender node to generate the output, but you can insert multiple ScanlineRender/Camera node pairs to generate output from various perspectives.

The ScanlineRender Node

The last node, ScanlineRender, sends the results of your 3D scene back into your composite as a 2D image. It’s always 2D in, 3D manipulation, and then 2D back out, which is why this is often called “2-and-a-half-D.”
The scanline render node converts the image back to 2D.

The image created by the ScanlineRender node is the same resolution as your project settings. When you need to render a specific resolution, use the optional `bg` pipe. Connect a Constant node with the resolution you want and that defines the output of the ScanlineRender node.

So now you know the basic 3D setup for your compositing script. Let's take a test drive.
Open the Tutorial Project File

The “3Dinteg_tutor.nk” project file includes the node trees for the first part of this chapter.

To Open the Project File

1. Launch the Nuke application and choose File > Open Comp from the menu bar.
2. In the file browser, navigate to your Nuke_Tutorials/3DInteg/ folder, select the 3dinteg_tutor.nk project file and click Open.
3. Locate the Tutorial_Path node, on the left side of the script, and double-click it to open its control panel.

4. Click the “file folder” button. Browse to the location where you installed the tutorial project files, and then click Open to select the location.

After you select the correct path, the error messages should clear from the Read nodes, and the thumbnails in the script update with the correct images.

5. Close the Tutorial_Path control panel. Then, choose File > Save Comp As to save a copy of the project file.

6. Move the mouse pointer over the Node Graph, and press F to frame the entire contents of the project file.

   The green arrows (lines) show the links between the Tutorial_Path node and the Read nodes.

7. If you wish, press Alt+E to hide the expression arrows.

The Tutorial_Path node saves the location of the project files on your computer, so you don't need to repeat this for future sessions with this project file.
Setting Up a 3D System

Let's start with the basics. In this first example, you'll create a basic 3D node tree, map an image to a 3D card, manipulate it, and then render the result back out to the 2D composite.

To Set Up a 3D Node Tree

1. In the “3Dinteg_tutor.nk” project file, locate the backdrop node labeled “Setting Up a 3D System.” You'll see a Read node with the image you'll use for this example.
2. Right-click over the nuke_sign.jpg node, and choose 3D > Geometry > Card.

This attaches a “Card1” node. Let's see what it looks like in 3D.

3. Attach a Viewer to the Card1 node and Nuke switches the Viewer to 3D.

Wow, that's amazing. It looks exactly like the 2D Viewer. How can anyone tell the difference? Check the lower-left corner of the Viewer and you'll see an orientation marker for the three axes in 3D. You'll also see that “3D” is displayed on the view dropdown menu.
That sign is a little darker than expected, isn't it? Actually, you can't see the image yet because the default view of the 3D workspace is at the origin or center of the space. Perhaps zooming-out may improve the view.

4. Press the **Alt** key (Windows /Linux) or the **Option** key (OS X), and drag with the middle mouse button to zoom or “dolly.” Drag to the left and you'll zoom out.

![Image of NUKE emblem](image)

Hey, look. There's the Nuke emblem. In the 3D Viewer, the “pan” and “zoom” controls are exactly the same as what you've used for the node tree and the 2D Viewer, but let's try “tumbling” to get a better view.

5. **Alt**- or **Option**-drag with the right mouse button to rotate around the origin point of the 3D workspace. You now see the 3D grid and the image mapped to the card.

![Image of 3D workspace](image)

When an image is connected directly to a Card node like this, it is applied as a flat or “planar” map. The size of the card adjusts to the dimensions of the image.

6. Click on the card and you select the node in the node tree and also the card inside the 3D workspace.

7. Use the mouse (and the **Alt** key) to navigate through the workspace. Go ahead, pan, dolly, and rotate at will. Then, press **F** over the Viewer to frame the 3D view.

---

**Tip:** If you don't like the standard navigation controls, open the **Preferences** control panel (**Shift+S**), select the **Viewers** tab and change the **3D control type** to **Maya, Lightwave**, or **Houdini**.
8. Click on an empty spot in the Node Graph to deselect all nodes. Let's add the other nodes you need.

9. Right-click on the Node Graph and choose 3D > Camera. Keep its control panel open so you can manipulate the camera in the Viewer.

10. Right-click and choose 3D > ScanlineRender to insert a render node, and then connect the nodes as shown below.

11. Connect the Viewer to the ScanlineRender node, and you have the most basic 3D system in Nuke.

12. Press Tab over the Viewer to change to the 2D view. You won't see the Nuke emblem - hey, where did it go? We saw it before.

13. Press Tab again to switch back to 3D. You'll see the default camera position is too close to view the card. Let's move things around to get an image for 2D.

To Position Objects in the Scene

1. Alt- or Option-drag with the middle mouse button to dolly out and show more of the 3D workspace.

2. Select the camera. You can do this by clicking the camera object in the Viewer or clicking the Camera1 node in the Node Graph.

3. Drag the transform handles to move the camera away from the card, along the z-axis.
As you drag the camera, look at the camera's control panel. You'll see the x/y/z transform values reflect the camera's current position.

4. Press and hold Ctrl (Mac users press Command) over the Viewer and the transform handles change to rotation rings.

5. Drag the green ring to rotate the camera around the Y-axis. Notice the x/y/z rotation values in the control panel reflect the angle of the rotation.
   The blue handle “rolls” or rotates on the Z-axis, and the red handle rotates on X.

6. Now, select the card object and move it away from the camera.
   Keep the control panel open for the Card node. As with the Camera node, the transform handles disappear from the Viewer when you close the control panel.

7. Drag the card's transform handles to position it in the 3D workspace. If you wish, press the Ctrl key (Mac users press Command) over the Viewer and rotate the card.
8. Press **Tab** over the Viewer to switch between the 2D and 3D views to see the image the ScanlineRender node produces.

9. Before you continue to the next example, close all the control panels that are currently open. In this example, it doesn't matter where you move the camera or card. In reality, however, you often need to use specific values, which you can enter directly in the control panels. You can also import camera data or animation curves - did you notice the **import chan file** button in the camera's control panel? - and apply them to the objects in the workspace.
Making a Scene

We mentioned earlier the Scene node creates a place where multiple objects may be seen by the camera and the render node. If you only have a single object, you don't need a Scene node, but where's the fun in that? Scene nodes make it possible to really tap into Nuke's ability to handle huge amounts of 3D information, and you should know how to use it.

To Set Up a Scene

1. Inside the “Setting Up a 3D System” node tree, drag a selection around the nuke_sign.jpg node and the Card1 node to select them.

2. Press Ctrl+C (Mac users press Cmd+C) to copy the selected nodes or choose Edit > Copy from the right-click menu.

3. Press Ctrl+V (Mac users press Cmd+V) or choose Edit > Paste to insert a copy of the nodes. Press Ctrl+V or Cmd+V again to insert a second copy, and then arrange the nodes as shown below.

4. There are multiple cards now, and you need a Scene node to create a space where the rendering node can see all cards at once.

5. Click on an empty space in the Node Graph to deselect all nodes. Right-click and choose 3D > Scene to insert the “Scene1” node.

6. Drag the Scene1 node onto the obj/scn connector to insert the Scene1 node between Card1 and ScanlineRender1.
7. Connect the obj/scn connector from the ScanlineRender1 node to the Scene1 node. Connect each Card node to the Scene1 node.

8. Double-click on the Card1 node to open its control panel. In the Viewer, you’ll see the transform handles for the first card.

9. Move and rotate the card to a different position. For this example, it doesn't matter where you place it.

10. Open the control panel for Card2 and move its card to a different place in the scene.

11. Open the Card3 control panel and move that card, also.

You could switch to the 2D view to see the result, but why not just look through the camera? Next to the view dropdown menu, you see the button that locks the 3D view to the camera.
12. From the view dropdown menu, choose 3D (V) to switch to a 3D perspective view. Then click the “lock view to 3D camera” button.

13. Turn off the “lock 3D view to selected camera” button. You won’t need it during the rest of this tutorial.
Merging and Constraining Objects

You can merge objects and move them together as a group. To do so, you need to insert MergeGeo and TransformGeo nodes after the objects. The MergeGeo node first merges the objects together, after which you can use the controls of the TransformGeo node to move the merged objects in the 3D space. You can also use the TransformGeo node to constrain objects, as you may notice later in this tutorial.

To merge the three card objects together, right-click on the Card1 node and select **3D > Modify > MergeGeo**. This inserts a MergeGeo node between Card1 and Scene1. Disconnect the Card2 and Card3 nodes from the Scene node and connect them into the MergeGeo node. Then, right-click on the MergeGeo node and select **3D > Modify > TransformGeo**. Your node tree should now look like the following:

![Node Tree Diagram]

On the TransformGeo nodes, you see multiple connectors. The connector without a label should be attached to a geometry object or a MergeGeo node. The other connectors act as constraints on the connected object’s position.

You can connect ‘look’ to the object or camera that the current object should ‘face’.

You can connect ‘Axis’ to an Axis node. This links the position of the current object to the position of the Axis node.

When a camera or object is connected to the optional **look** connector, the TransformGeo node adjusts the rotation so that the object’s z-axis always “points” to the camera or object.
The axis connector can be used to link the current object to the position, rotation, and scale of a special 3D object called the Axis node. If you've worked with other 3D applications, you know the Axis node as a “null” or “locator” object.

You are still working with the “Setting Up a 3D System” node tree. The following steps show how you can move the merged nodes, and also how to make objects “look” at the camera and other objects.

To Move the Merged Objects Together

1. Click on the TransformGeo1 node to select it. Its control panel should also be open and you'll see its transform handles in the Viewer.
2. Drag the handles to move all the cards merged with the MergeGeo node.
3. Press the Ctrl or Command key and drag the rings to rotate the cards as a group.
4. In the TransformGeo1 control panel, drag the uniform scale slider to increase the size of the entire group of cards.

To Make Objects ‘Look’ at the Camera

1. Drag the look connector from the TransformGeo1 node onto the Camera1 node.
In the Viewer, you'll now see the TransformGeo1 node is constrained to the location of the camera.

2. Select and move Camera1 in the Viewer window. As you do so, the three cards controlled by the TransformGeo1 node rotate to “look at” the camera location.

Why is this useful? Let’s assume you have a 2D matte painting mapped to a card in your scene. The “look” option ensures that the plane of the painting always faces the camera, regardless of the camera position, and maintains the illusion depicted by the painting.

Before you move on, disconnect the TransformGeo node’s look connector from the camera.
Animating a Scene

The little scene you've created would be more interesting with a camera move. You can animate both cameras and objects; in each control panel, you'll see an Animation button next to each parameter you can animate over time.

To Animate the Camera

1. In the Viewer, drag the time slider to frame 1 on the timeline.
2. Let's switch to an overhead view to move the camera. Choose top from the view dropdown menu.
3. Double-click on the Camera1 object (either inside the Viewer or on the Node Graph) to open its control panel.
4. Move the camera to the right and rotate it to “look” at the center of the 3D workspace.
5. Click on the animation button next to the camera’s translate parameters, and choose Set key.

The background of the parameter boxes change color to show that a keyframe is now set for these values at the current frame in the timeline. Now, you need to set a keyframe for the rotation values.
6. Next, click the animation button next to the camera’s *rotate* parameters, and choose **Set key**.
7. In the Viewer, scrub to the end of the timeline. Then, move the camera to the left side and rotate it to face center. This automatically sets keys for the *translate* and *rotate* parameters, for the last frame.

![Image of animation parameters](image)

8. Drag through the timeline and the camera moves between the positions recorded by key frames. Yawn. With only two key frames, the camera moves in a straight line, from start to finish. Let's edit the animation curves to make this more interesting.
9. Click the animation button next to the camera's translate parameters and choose **Curve editor**.

![Image of Curve Editor](image)

This opens the Curve Editor window in the same pane as the Node Graph. The outline at the left side of the Curve Editor displays a list of the parameters you've animated, and the curves show the values plotted over time. Each dot shows the value recorded for a value on a keyframe.

![Image of Curve Editor](image)

Yes, we know. They don't look like curves - yet. You only set two key frames, remember? You can press **Ctrl+Alt** (Mac users press **Cmd+Alt**) and click on any line to add more key frames. However, let's assume you want to create a smooth arc between the first and last recorded position of the camera. Rather than set more key frames, let's just change the shape of the curve.
10. You want to control the shape of the camera path along the z - the distance between the origin point and the camera, so click the *translate / z* parameter in the Curve Editor. Click on the first point of the translate/z curve to select it, and drag the tangent handle upward, as shown below.

![Image of Curve Editor](image)
From this point forward, the curve increases the distance of the camera on the z-axis, which you’ll now see in the Viewer.

11. Click on the last point of the translate/z curve to select it, and drag it upward also to finish the desired shape. This eases the distance back toward the value of the keyframe at the end of the timeline.

Select the camera in the Viewer and you should see the gradual slopes of the curve create a more interesting arc for the camera move.

Switch to the 3D (V) perspective view and scrub through the timeline to see the new camera move.

Your version may look a little different than this, depending on the positions and rotations you defined, but you get the idea.

12. If you wish, you can set additional key frames to refine the camera path. Hold down Ctrl+Alt or Command+Option and click on the z-axis curve in the Curve Editor, to add new points to the curve, and then adjust their positions.

13. Before you continue, click the Node Graph tab to hide the Curve Editor and return to the node trees.

14. Close all the control panels that are currently open.
Working with Geometry

In the previous example, you worked with the card object. Nuke also includes primitive geometry, which can be used as set-extension geometry or placeholders for other elements you plan to add to scene.

To Add Primitive Objects to the Scene

1. In the “3Dinteg_tutor.nk” project file, locate the node tree labeled “Working with Geometry.” We've already supplied the 3D node tree with a camera for you, so you need to add the geometry objects, and also create a "scene" where they can co-exist.
2. Right-click over the Node Graph and choose 3D > Scene. Connect Scene2 to ScanlineRenderG.
3. Connect a Viewer to the ScanlineRenderG node and switch to the 3D perspective view.
4. Right-click and choose 3D > Geometry > Cube.

The default cube primitive appears at the center of the 3D workspace. Let's reduce the number of subdivisions on the cube.

5. In the Cube1 control panel, change the **rows** parameter to 4. Change the **columns** parameter to 4, also.
6. Connect the **Cube1** node to the **Scene2** node. Now let’s adjust the shape of the cube.

7. Reduce the height of the cube by dragging the top-center point down.

8. From the view dropdown menu, choose the **front** view to see a non-perspective view of the cube. These non-perspective views can help you size and position objects with more accuracy than you might get in a perspective view.

   Mm... the cube is actually below the x-axis. Let’s move it up, but check the values in the **Cube1** control panel.

9. Drag the top of the cube until the **t** (top) value in the **Cube1** control panel is about **0.3**. Drag the bottom of the cube to align it with the x-axis.

10. It looks like you don’t need 4 divisions on the sides of the cube, so change the number of rows to **2** in the **Cube1** control panel.
Now let's add a few more primitives - a cylinder and a sphere.

11. Right-click on the Node Graph and choose 3D > Geometry > Cylinder. Connect the Cylinder1 node to the Scene2 node.

12. Change the view to 3D (V) and zoom out a little to see the whole cylinder.

13. In the Cylinder1 control panel, set the rows to 1, the columns to 20.

14. Set the radius to 0.35 and the height to 1.5. Also check the box for close top.
So now you have a cylinder in the scene.

15. Choose front from the view dropdown menu and move the cylinder up to rest on top of the cube.

16. Now add a sphere. Choose **3D > Geometry > Sphere**. In the Sphere1 control panel, set both the **rows** and **columns** to 15, and change the **radius** to 0.35.

17. Make sure the Sphere control panel is open and move the sphere object to cap the top of the cylinder.
18. Select the **3D** from the view dropdown menu and rotate the view around the objects in your scene. At this point, they have no surface properties, so you’ll need to connect a 2D image from the Node Graph to each object.

19. In the Node Graph, connect the **concrete.jpg** to each of the objects.
Lighting and Surface Properties

Nuke includes lighting tools to enhance the existing lighting in the plates and images you include in a 3D scene. Also included are fundamental surfacing tools to control the attributes of the objects in the 3D workspace.

These tools are not designed to replace the use of true 3D lighting and surfacing, but they can definitely help you punch up the scene and quickly tweak the settings without sending elements back to the 3D application.

Nuke’s lighting objects introduce lighting and surface attributes into your scene. When the scene has no lighting objects, then all surfaces are illuminated with the same properties and level of “brightness.”

In the following steps, you’ll first add nodes that define surface properties for the objects, and then you’ll add the light objects to illuminate them.

To Define Surface Attributes to Objects

1. Click on an empty place in the Node Graph to deselect all nodes. Then, choose 3D > Shader > BasicMaterial.
2. Drag the BasicMaterial1 node onto the connector between concrete.jpg and Sphere1.

In the Basic Material control panel, you can see parameters to define the amount of light emission, diffusion, and specular properties of the surface. You can mask these properties by connecting images to the mapS (specular), mapE (emission), and mapD (diffuse) connectors, but this is not required for this example.
3. Set the light emission control to 0.25. Set diffuse to 0.18, and specular to 0.75.

4. Adjust the min shininess and max shininess values to adjust the quality of the specular highlights. Once again, nothing seems to happen! That's because you haven't yet added a light into the scene. All surfaces are illuminated with the same level of brightness, so there are no specular highlights or other controllable light properties visible. It's not very exciting, but don't worry - you get to add a light into the scene soon.

5. Make two copies of the BasicMaterial1 node and attach a copy before Cylinder1 and before Cube1.

To Add Light Objects to a Scene

1. Choose 3D > Lights > Spot and connect the light node to the Scene2 node.

2. In the Spotlight1 control panel, rename the light to Keylight.

3. Switch to the top view and drag x-axis handle (red) to move the light to the left.

4. Drag the z-axis handle (blue) to move the light closer to the bottom edge of the screen. Then, press the Ctrl or Command key and rotate the light to face the pillar object.
5. Switch to the 3D (V) view and rotate the view so you can see both the Keylight and the pillar geometry.
6. Drag the y-axis handle (green) to move the light up above the pillar.
7. Press the Ctrl or Command key and rotate the light down to shine on the pillar.

That's the basic setup for lighting and surfaces, but there are other tools for more complex setups. Refer to 3D Compositing for more information on the 3D lighting and surfacing tools.
Epilogue

In this chapter, you learned how to setup a 3D workspace for your composite, and how to work with 3D geometry and lighting. These are all prerequisite skills for more advanced topics, such as camera projections, match-moving, and set replacement.
Appendices

This section contains supplemental reference information that you may need when using Nuke.

Organization of the Section

The section consists of the following appendices:

- **Appendix A: Preferences** lists the preferences supported by Nuke.
- **Appendix B: Keyboard Shortcuts** lists the keyboard shortcuts you can use for quicker and easier access to Nuke’s features. You can also open a list of keyboard shortcuts from the application by selecting Help > Timeline or Comp Keyboard Shortcuts.
- **Appendix C: Supported File Formats** lists the image, video, and audio file formats Nuke supports.
- **Appendix D: External Software** lists third party libraries used in Nuke.
Appendix A: Preferences

The Available Preference Settings

The Preferences dialog is divided into the following sections:

<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>General</td>
<td>Settings for auto-saving and path substitutions.</td>
</tr>
<tr>
<td>Project Defaults</td>
<td>General project settings, and settings for color management.</td>
</tr>
<tr>
<td>Performance</td>
<td>Settings for caching, hardware, localization, and threads/processes.</td>
</tr>
<tr>
<td>Behaviors</td>
<td>Settings for start up, file handling, export options, scripting, node behaviors, and more.</td>
</tr>
<tr>
<td>Panels</td>
<td>Settings for the interface appearance, file browser, control panels, nodes, Viewers, script editors, and scopes.</td>
</tr>
</tbody>
</table>

General

<table>
<thead>
<tr>
<th>General</th>
</tr>
</thead>
<tbody>
<tr>
<td>Autosave</td>
</tr>
<tr>
<td>force project autosave after &lt;300&gt; seconds</td>
</tr>
<tr>
<td>idle comp autosave after &lt;5&gt; seconds</td>
</tr>
<tr>
<td>force comp autosave after &lt;30&gt; seconds</td>
</tr>
<tr>
<td>autosave comp filename</td>
</tr>
</tbody>
</table>
**General**

**Path Substitutions**

| path substitutions | Allows you to remap file paths in order to easily share projects across different operating systems. When the application encounters a file path, any text in the **OSX/Linux** column is replaced with the text in the **Windows** column, or vice versa. For example, if you enter `/Volumes/networkmount` in the **OSX/Linux** column and `Z:` in the **Windows** column:  
- On Mac and Linux, any file paths that start with `Z:` are converted to start with `/Volumes/networkmount`.  
- On Windows, any file paths that start with `/Volumes/networkmount` are converted to start with `Z:`.  
To be able to enter text in either column, you need to click on the + button below to add a row to the table. |
| + | Adds a row under **path substitutions**. |
| - | Deletes the selected row(s) under **path substitutions**. |

**Project Defaults**

**Note:** You must restart the application for changes to Project Defaults preferences to be applied.

**Channel Management**

**Channel Management**

| Channel Warning Threshold | Sets the total number of channels required in a script to trigger the Channel Warning. Nuke only supports 1023 uniquely named channels per script.  
The **Channel Count** is displayed in the bottom-right of the interface, next to the Localization Mode indicator. |
## Channel Management

**Note:** Nuke does not remove unused channels until you close and reopen a script, so the Channel Count does not decrease when you remove Read nodes from the Node Graph.

## Color Management

### OpenColorIO config

<table>
<thead>
<tr>
<th>OpenColorIO config file</th>
<th>Sets the OpenColorIO configuration to use, if you don't intend to use the nuke-default settings. If you select custom from the dropdown, enter the file path of the configuration file or click Choose to use the browser.</th>
</tr>
</thead>
</table>

**Note:** Nuke also includes an environment variable method for setting a config file. See Environment Variables in the Nuke Online Help for more information.

### Default Color Transform

<table>
<thead>
<tr>
<th>Working space</th>
<th>Sets the colorspace files should be converted to, on import, and from, during render - it's the colorspace used by Nuke under the hood.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Viewer</td>
<td>Sets the default LUT applied to Viewers.</td>
</tr>
<tr>
<td>Thumbnails</td>
<td>Sets the default LUT applied to thumbnails when ever they are generated.</td>
</tr>
<tr>
<td>8 bit files</td>
<td>Sets the default LUT applied to the specified ingested file type.</td>
</tr>
<tr>
<td>16 bit files</td>
<td></td>
</tr>
<tr>
<td>Log files</td>
<td></td>
</tr>
<tr>
<td>Floating point files</td>
<td></td>
</tr>
</tbody>
</table>
## Color Management

| color management | Sets whether Nuke uses the LUTs read from the configuration specified or the Nuke native LUTs during export. Selecting OCIO makes the relevant OCIO LUTs available to the Read and Write nodes in scripts on a per project basis. All configurations except nuke-default automatically switch this control to OCIO. |

### General - these preferences only apply to new scripts and projects. To affect the current project, use the Project Settings.

### Project

| project directory | Sets the project directory used by new projects. You can change the project directory for the current project in the Project Settings. |
| Hrox Directory | Click to set the project directory to the location of the .hrox file using the \[python {nuke.script_directory()\}] expression. |
| export directory | Sets the directory used by timeline exports:  
  - **Use Project Directory** - use the directory specified by the project directory preference.  
  - **Use Custom Directory** - use the directory specified by the custom export directory control. |
| custom export directory | Sets the export directory used by new projects when the export directory control is set to Use Custom Directory. |

### Sequence

| output resolution | Use this to set the output resolution in the Timeline environment for new projects. By default, clips in the sequence are not reformatted to fit this format, they retain the source clip resolution. You can adjust the reformatting for new clips added to a sequence using the Clip settings below or by selecting clips in the timeline and adjusting the settings in the Properties tab. |
## Appendix A: Preferences | Project Defaults

### General
- These preferences only apply to new scripts and projects. To affect the current project, use the **Project Settings**.

<table>
<thead>
<tr>
<th>Preference</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>frame rate</strong></td>
<td>Select the frame rate for new projects in the Timeline environment.</td>
</tr>
<tr>
<td><strong>start timecode</strong></td>
<td>Use this to define the start timecode for new projects. For shots, this overrides the timecode defined in the media.</td>
</tr>
<tr>
<td><strong>time display</strong></td>
<td>You can use this to select the display format for times. You can select either <strong>Timecode</strong> or <strong>Frames</strong>.</td>
</tr>
<tr>
<td><strong>drop frame</strong></td>
<td>Use this to choose whether timecodes from this sequence are displayed in drop frame times or not.</td>
</tr>
</tbody>
</table>

Drop Frame is a timecode display option that leaves out two frames from the 30 fps timecode sequence every minute (except every 10th minute) so that long running NTSC sequences are accurate to a real-time clock (NTSC frame rate is 3000/1001, or approximately 0.01% slower than 30fps).

**Note:** Enabling Drop Frame is a Timecode display feature only - the source media remains a continuous stream of frames.

### Clip

<table>
<thead>
<tr>
<th>Preference</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>clip reformat</strong></td>
<td>Sets how new clips added to a sequence are formatted. The default, <strong>None</strong>, displays shots at the source clip resolution, whereas <strong>To Sequence Resolution</strong> reformats the shot to the output resolution using the resize type control.</td>
</tr>
<tr>
<td><strong>resize type</strong></td>
<td>When <strong>clip reformat</strong> is <strong>To Sequence Resolution</strong>, determines how the shot is resized to fit the output resolution. This control is disabled when <strong>clip reformat</strong> is set to <strong>None</strong>.</td>
</tr>
<tr>
<td><strong>center</strong></td>
<td>When <strong>clip reformat</strong> is <strong>To Sequence Resolution</strong>, disabling <strong>center</strong> places the clip at the bottom-left of the output resolution, rather than the center.</td>
</tr>
</tbody>
</table>

### Poster Frame

<table>
<thead>
<tr>
<th>Preference</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>poster frame</strong></td>
<td>Sets a preset poster frame for all new source clips or select <strong>Custom</strong> to select a relative frame offset as the poster frame:</td>
</tr>
</tbody>
</table>

- **First** - new source clips display the first frame of the file on disk as the
### General
- These preferences only apply to new scripts and projects. To affect the current project, use the Project Settings.

<table>
<thead>
<tr>
<th>poster frame.</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>• <strong>Middle</strong> - new source clips display the middle frame of the file(s) on disk as the poster frame.</td>
<td></td>
</tr>
<tr>
<td>• <strong>Last</strong> - new source clips display the last frame of the file(s) on disk as the poster frame.</td>
<td></td>
</tr>
<tr>
<td>• <strong>Custom</strong> - the poster frame number is derived from the number of frames in the clip starting at 0. Use the frame offset control to set the relative offset.</td>
<td></td>
</tr>
</tbody>
</table>

For example, a `.dpx` sequence `myClip.####.dpx 1001-1500` can have a custom poster frame between 0 and 499.

<table>
<thead>
<tr>
<th>frame offset</th>
<th>When poster frame is set to <strong>Custom</strong>, enter the relative frame offset from the first available frame in the file(s) on disk.</th>
</tr>
</thead>
</table>

### Views
- These preferences only apply to new scripts and projects. To affect the current project, use the Project Settings.

| Views |
| --- | |
| **Views** | |
| +/- | Click to add and remove views from the views list. |
| ↑ ↓ | Click to move views up and down the views list. Moving a view changes the position of the corresponding Viewer button. |
| View | Lists the views in the script or project and the color associated with the view when **Use color in UI** is enabled. |
| hero | Sets the principal view selected on script or project load. |
| Set up views for stereo | Click to automatically add **left** and **right** views to scripts or projects. |
| Use colors in UI? | When enabled, the colors specified in the views matrix are applied to the interface. For example, if you click **Set up views for stereo** and enable this control, any UI items representing the left and right views are colored red and green. |
## Performance

### Caching

<table>
<thead>
<tr>
<th><strong>Timeline Disk Caching</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>directory path</strong></td>
</tr>
<tr>
<td><strong>limit to (GB)</strong></td>
</tr>
<tr>
<td><strong>EXR compression</strong></td>
</tr>
<tr>
<td><strong>clear cache</strong></td>
</tr>
</tbody>
</table>

### Comp Disk Caching

| **temp directory**       | The comp disk cache saves all recent images displayed in the Viewer for fast playback. Using this control, you can specify where you want Nuke to save these images. Select a local disk (for example, C:/temp), preferably with the fastest access time available. It's also important to leave enough space for the maximum disk cache size (defined below). |
| **comp disk cache size (GB)** | Specifies the size of Nuke's disk cache, independent of the playback cache. It is the maximum amount of disk that Nuke can allocate for caching. When the limit is reached, Nuke attempts to free disk space before using |
### Caching

| rotopaint cache size (GB) | Specifies the size of Nuke's RotoPaint tile cache. It stores tiles for the output image of each RotoPaint node, enabling you to paint on top of existing RotoPaint without having to re-render the strokes underneath. If you run out of RotoPaint disk cache, response times may suffer when painting onto tiles containing lots of strokes. |

| Memory Caching |

| playback cache size (% of system RAM) | Specifies the percentage of system RAM used for the timeline Viewer playback cache. The entire amount is allocated, even if you've only got a few frames in the Viewer. Recently used frames are retained in the memory to avoid relying on the disk buffering system. The cache is freed when you switch to the compositing Viewer and reallocated when you switch back to the timeline Viewer. **Tip:** On low-end machines, minimizing this may improve application responsiveness at the expense of smooth playback. |

| free timeline playback RAM cache when switching to the node graph | When enabled, any frames cached to RAM (the white bar in the timeline Viewer) are discarded when you switch to the Node Graph within Nuke Studio, freeing the RAM for use in Nuke. **Note:** When you switch back to the timeline, the cached files are re-cached, which can be time consuming. |
## Caching

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>comp cache size (% of system memory)</td>
<td>Specifies the percentage of system memory available for comp caching. It is independent of the playback cache and is the maximum amount of memory that Nuke can use for caching. When the limit is reached, Nuke attempts to free memory before using any more.</td>
</tr>
<tr>
<td>comp playback cache size (% of comp cache)</td>
<td>Specifies the percentage of the comp cache available for comp playback. This cache holds data displayed in Nuke's compositing Viewer, the result from the node tree you're viewing. Results from further up the tree are sometimes cached inside the comp cache as well, such as the output from a node which is needed by more than one downstream node.</td>
</tr>
<tr>
<td>comp paint cache size (% of comp cache)</td>
<td>Specifies the percentage of the comp cache available for comp paint. The comp playback cache and comp paint cache are limits for the total memory that is used, so it's possible to have a combined size greater than 100%. How the memory is shared between the two depends on what you're doing, and you won't necessarily be filling both up at the same rate. For example, if you're doing a lot of paint work, you might want to allow the paint cache to fill up more than 50% of the available memory if the playback cache isn't using its full share.</td>
</tr>
</tbody>
</table>

## Audio Waveforms

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>waveform memory (MB)</td>
<td>Sets the amount of memory available for storing timeline audio waveforms.</td>
</tr>
</tbody>
</table>

## Application in Background

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>pause timeline Viewer when the application goes to the background</td>
<td>When enabled, pause timeline Viewer caching when the application is in the background.</td>
</tr>
<tr>
<td>clear timeline Viewer cache when the application goes to the background</td>
<td>When enabled, the timeline Viewer cache is cleared when the application goes into the background.</td>
</tr>
</tbody>
</table>

**Note:** This preference is only available when pause timeline Viewer when the application goes to the background is enabled.
## Caching

<table>
<thead>
<tr>
<th>Preference</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>pause comp Viewer when the application goes to the background</td>
<td>When enabled, pause comp Viewer caching when the application is in the background.</td>
</tr>
<tr>
<td>clear comp Viewer cache when the application goes to the background</td>
<td>When enabled, the comp Viewer cache is cleared when the application goes into the background.</td>
</tr>
</tbody>
</table>

**Note:** This preference is only available when pause comp Viewer when the application goes to the background is enabled.

## Undo Caching

<table>
<thead>
<tr>
<th>Preference</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>undo history size</td>
<td>Allows you to set the amount of RAM, in MB, to use for the undo history. If this limit is exceeded, older items are discarded.</td>
</tr>
<tr>
<td>minimum undo events</td>
<td>Use this to set the amount of undo events. This setting always applies, even if it breaches the undo history size limit.</td>
</tr>
</tbody>
</table>

## Expression

### Expressions Re-Evaluation

<table>
<thead>
<tr>
<th>Mode</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Always</strong></td>
<td>expressions in the script are re-evaluated after every change to the Node Graph. This can cause performance issues in large, expression-heavy scripts.</td>
</tr>
<tr>
<td><strong>Lazy</strong></td>
<td>expressions are only re-evaluated when required for GUI or render updates. This option can speed up interactive performance in large, expression-heavy scripts.</td>
</tr>
</tbody>
</table>
# Hardware

## Audio

| audio device | The **audio device** control allows you to select an existing audio device for playout from a list of automatically detected devices. You can disable playout on a device by selecting **Disabled**. |

## RED Rocket

| use red rocket | You can select the **use red rocket** checkbox to speed up decoding RED media. If you’re using R3D Rocket graphics card, note that using it is likely to only be considerably faster when you're reading in at full resolution. If you're reading in at half resolution, for instance, using without the R3D Rocket card enabled may be faster. This is because the R3D Rocket graphics card is designed to be fast when reading in multiple frames at the same time. This is not how it works internally, and therefore reads with the R3D Rocket card disabled may sometimes be faster when working in lower resolutions (< 4K widths). Also, note that the R3D Rocket card always produces better results than when downsampling. Also, the R3D Rocket card can only be used by one application at a time, so if you are viewing multiple scripts at once, you may only be able to use the R3D Rocket card in one. |

## GPU

| expand 3 to 4 channels | You can use this to expand images cached for playback from 3 to 4 color channels per pixel. Some graphics hardware performs better at loading images to video memory with 4 channels per pixel, than it does with 3. Enabling this option improves playback performance on such hardware, at the expense of reducing the number of frames that it's possible to cache. If you are seeing poor playback performance, enabling this option may help. However, if you are seeing acceptable playback performance with this option disabled, then leaving it disabled increases the number of frames that may be cached for smooth playback. |

| GPU texture cache | Use this to set the maximum amount of GPU memory to use for caching |

---

**Note:** You must restart Nuke for this option to take effect.
## Hardware

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>size (MB)</td>
<td>textures.</td>
</tr>
<tr>
<td>default blink device</td>
<td>You can use this to set the default blink device to use the CPU, or choose which GPU to use for GPU accelerated plug-ins, such as, ZDefocus, Convolve, Denoise, BlinkScript, Kronos, MotionBlur, and so on. Any changes to this setting only take effect after you have restarted the application.</td>
</tr>
<tr>
<td>enable multi_GPU support</td>
<td>If you have multiple GPUs of the same type installed, you can enable this preference to share work between the available GPUs for extra processing speed. This is a global preference and is applied to all GPU enabled nodes. See Windows, Mac OS X and macOS, or Linux for more information on the GPUs Nuke supports.</td>
</tr>
</tbody>
</table>

## Localization

### System

<table>
<thead>
<tr>
<th>Mode</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>mode</td>
<td>Sets the overall localization mode:</td>
</tr>
<tr>
<td>on</td>
<td>checks for updates to source clips and Read nodes with localization policy set to On or From auto-localize path and localizes those files automatically.</td>
</tr>
<tr>
<td>manual</td>
<td>checks for updates to source clips and Read nodes with localization policy set to On Demand and prompts you to update them manually.</td>
</tr>
<tr>
<td>off</td>
<td>no source clips or Read nodes are localized, regardless of the their localization policy.</td>
</tr>
</tbody>
</table>

**Note:** The current localization mode is displayed in the status bar at the bottom-right of the interface.

| Read source files when localized files are out of date | When enabled, source clips and Read nodes referencing out of date localized files automatically revert to reading the entire sequence from the original source files. Source files are read in the following circumstances: |
## Localization

| With Localization mode set to **on**: | A localized source clip or Read node with localization policy set to **on demand** is detected to be out of date. |
| | A localized source clip or Read node with localization policy set to **on** or from auto-localize path is detected to be out of date and it is queuing to be automatically localized. |
| With Localization mode set to **manual**: | A localized source clip or Read node with localization policy set to **on**, on demand, or from auto-localize path is detected to be out of date. |

When disabled, the out of date localized files are read until you update them manually.

| hide out of date progress bar | When **read source files when localized files are out of date** is enabled, you can enable this control to hide the progress/status bar on Read nodes that are reading from the original source files. |
| pause localization on script/project open | When enabled, localization does not start automatically when you open a script or project. Enabling this option can help to open scripts and projects more quickly. |

### Inputs

| localization policy | Sets the localization policy for all new source clips and Read nodes: |
| | • **on** - always localize source clips and Read nodes with this policy automatically. |
| | • **from auto-localize path** - localize these source clips and Read nodes automatically if they reside in the **auto-localize from** directory. |
| | • **on demand** - only localize these source clips and Read nodes when you manually update them. |
| | • **off** - never localize these source clips or Read nodes. |

### Paths

| auto-localize from | Enter the location of the files you need automatically localized, unless otherwise specified in the Read node's **cache locally** control or in the bin right-click, Localization Policy menu. Commonly this would be your |
### Localization

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>localize to</td>
<td>Enter the file path where all the localized files are automatically stored. Localizing files allows for faster reloading for files that are stored in a location that is slow to access (such as a network drive).</td>
</tr>
</tbody>
</table>

On Windows, files saved to the `localize to` directory replace `\` (double backslashes) and `:` (colon drive signifiers) with underscores so that the file path works as expected between operating systems. For example:

- `\\windowspath\to\my\network\file.dpx` is saved as `_windowspath\to\my\network\file.dpx`
- `t:\my\network\path\file.dpx` is saved as `t\_my\network\path\file.dpx`

### Storage

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>limit to (GB)</td>
<td>This allows you to set the maximum amount of space (in GB) to use for localized files. Set to zero for unlimited size. Values lower than zero, leave that amount of space free. The <strong>currently in use</strong> and <strong>currently free</strong> fields display how much free storage remains from the total specified.</td>
</tr>
</tbody>
</table>

If this limit is reached during localization, a dialog displays options to free up storage space.

### Network

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>check for updated files every</td>
<td>When files are localized, specifies the time interval (in minutes) before Nuke checks for updated versions of the files.</td>
</tr>
</tbody>
</table>

### Appearance

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>progress bar</td>
<td>Sets the colors used to represent the localization states of source clips and Read nodes.</td>
</tr>
</tbody>
</table>

### Threads/Processes

- **Playback**
## Threads/Processes

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>default number of threads per reader</td>
<td>Sets the number of threads to use per reader. If your source files are located on high performance local drives, increasing the number threads can significantly improve read times. CPU intensive operations, such as .jpg decoding, can also be improved by increasing the number of threads per reader.</td>
</tr>
<tr>
<td>override number of threads per reader</td>
<td>Allows you to override the default number of decode threads used, dependent on file format. Use the plus button to add an entry to the table and then select the file format using the dropdown menu. Double click the Number of threads column to set the required number of decode threads for that format.</td>
</tr>
<tr>
<td>OpenEXR helper threads to use</td>
<td>Sets the number of helper threads to use for OpenEXR only. The default, zero, automatically sets the number of helper threads used.</td>
</tr>
<tr>
<td>Arri helper threads to use</td>
<td>Sets the number of helper threads to use for ARRI only. The default, zero, automatically sets the number of helper threads used.</td>
</tr>
</tbody>
</table>

**Note:** The OpenEXR and ARRI helper thread preferences are independent of the threads per reader and override table per format settings.

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>QuickTime decoders to use</td>
<td>Sets the number of background processes available to handle QuickTime file I/O. You must restart the application for this preference change to take effect.</td>
</tr>
</tbody>
</table>

**Note:** Using too many decoders can affect performance, depending on the available hardware.

**Note:** You must restart Nuke for this setting to take effect.

## Rendering

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>render using frame server (Nuke)</td>
<td>When enabled, the Frame Server is always used for rendering.</td>
</tr>
</tbody>
</table>
### Threads/Processes

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>frame server render timeout</strong></td>
<td>Allows you to increase the number of minutes a render process can stay unresponsive before ending. If you're experiencing Render application timed out messages with process-heavy scripts, you can try increasing this value.</td>
</tr>
<tr>
<td><strong>focus background renders</strong></td>
<td>When enabled, rendering using the Frame Server automatically opens the Background Renders panel, or if it is already open, shifts focus to the panel.</td>
</tr>
<tr>
<td><strong>frame server processes to run</strong></td>
<td>Set the number of slave Nuke processes to run for the frame server.</td>
</tr>
<tr>
<td><strong>export renders</strong></td>
<td>You can select from several render options:</td>
</tr>
<tr>
<td>- limit renderer (more responsive ui)</td>
<td>Select this option to make the user interface more responsive during transcodes. It tells Nuke to use 2 threads to transcode and to use 25% of RAM to cache. Using this option is likely to result in slower transcodes.</td>
</tr>
<tr>
<td>- no renderer limits (fastest transcoding)</td>
<td>Select this option to ensure that transcodes happen as quickly as possible. This option may result in a less responsive user interface during transcodes.</td>
</tr>
<tr>
<td>- customize render limits</td>
<td>Select this option to manually configure the number of threads used and cache memory available when transcoding files.</td>
</tr>
<tr>
<td><strong>number of threads</strong></td>
<td>Sets the number of threads that Nuke uses when transcoding. Lower numbers allow the Timeline environment's interface to be more responsive. Higher numbers allow faster transcodes. This setting is passed to Nuke using the -m option.</td>
</tr>
<tr>
<td><strong>cache memory (GB)</strong></td>
<td>Use this to set the number of gigabytes of RAM that Nuke uses for its cache.</td>
</tr>
</tbody>
</table>

**Note:** Local Frame Server processes use ports 5558-5662.

**Note:** You must restart Nuke for this setting to take effect.

**Note:** You must restart Nuke for this setting to take effect.
Threads/Processes

settings. Lower numbers may improve the Timeline environment's interface responsiveness, while higher numbers may improve the speed of the transcodes. This setting is passed to Nuke with the -c option.

background renders

Sets when background renders occur:

• **don't auto-start background renders** – Comps on the timeline are not rendered in the background automatically.

• **start background renders on Comp save** – Comps on the timeline are rendered in the background automatically when they are saved.

• **start background renders on Comp create, Comp save and Comp version change** – Comps on the timeline are rendered in the background automatically when first created, when saved, and when a new version is selected.

Downsize Filtering

8-bit images
10-, 12- and 16-bit integer images
16-bit float images
32-bit images

Customizes the downsize filtering behavior by bit-depth. The default (1x) retains the original image size. You can select 2x to halve the image size, or 4x to quarter the image size.

The Viewer image quality dropdown affects the decode rate and resolution of clips displayed in the Viewer. Lower resolutions decode faster and vice versa.

Behaviors

Documentation

documentation source

Sets the help source for the **Properties ?** button:

• **local** – use Nuke's built-in HTML help server. Nuke's local help server also searches the NUKE_PATH, .nuke, and .nuke/Documentation directories for HTML files with the same name as the requested node, such as Blur.html.

• **foundry** – This uses Foundry's Online Help, the most up-to-date version of the documentation.

• **custom** – Select this to point to your own help server.
### Documentation

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>auto port</td>
<td>When enabled, assign a free port automatically.</td>
</tr>
<tr>
<td>local port</td>
<td>Specify a local documentation server port manually. This is usually &gt;= 1024. You can also set this to 0 to automatically assign the port.</td>
</tr>
<tr>
<td>range</td>
<td>Specify a range of ports to attempt with the local documentation server.</td>
</tr>
</tbody>
</table>

### File Handling

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>scan for file sequence range on drop into Bin view</td>
<td>When enabled, identify and import the file range of media that is dropped into the bin. When disabled, no range is detected and only a single frame is ingested. (This does not affect container formats, such as .mov and .r3d.)</td>
</tr>
<tr>
<td>automatically rescan versions when moving off end of the version list</td>
<td>When enabled, incrementing a source clip or shot’s version past the end of the previously discovered versions list, forces a rescan to update the versions list. See Using Versions for more information.</td>
</tr>
<tr>
<td>frame number style</td>
<td>Sets the sequence display mode to be used in the file browser.</td>
</tr>
<tr>
<td>assume fixed width frame numbers in file sequences</td>
<td>When enabled, assume frames have a fixed width number. With this selected, frame numbers need to be padded with zeros to a fixed length, otherwise frame numbers without preceding zeros are assumed to belong to sequences with no padding. This is important as the sequence identifier specifies a unique file name for each and every frame. For example:</td>
</tr>
<tr>
<td></td>
<td><strong>Files</strong></td>
</tr>
<tr>
<td></td>
<td>sequence 1.18.exr, sequence 1.19.exr, and sequence 1.20.exr</td>
</tr>
<tr>
<td>default red clip video decode mode</td>
<td>Sets the default red clip decode mode for new projects. You can choose from FullPremium, HalfPremium, HalfGood, QuarterGood, EighthGood, or SixteenthGood.</td>
</tr>
</tbody>
</table>

---

**NUKE USER GUIDE**

1958
### File Handling

**Note:** Changing this preference does not change the default decode setting for existing projects.

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>alembic files</td>
<td></td>
</tr>
<tr>
<td>always load abc files as all-in-one</td>
<td>When enabled, all <code>.abc</code> files are imported as a single node, without displaying the Alembic import scenegraph.</td>
</tr>
</tbody>
</table>

### Nodes

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>new Merge nodes connect A input</td>
<td>When enabled, inserting a new Merge node automatically connects the <strong>A</strong> input.</td>
</tr>
<tr>
<td>autokey roto shapes</td>
<td>When enabled, keyframes are added automatically to Roto shapes when they are adjusted.</td>
</tr>
<tr>
<td>when Viewer is closed delete its node</td>
<td>When enabled, Viewer nodes are deleted when you close the associated Viewer.</td>
</tr>
</tbody>
</table>

### Tab Search Menu

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
</table>
| Weighting | When enabled, nodes in the Tab menu are weighted so that more commonly used nodes appear at the top of the list.  
When disabled, the node list is ordered as it appears in the nodes toolbar or alphabetically if you start typing. |
| Favorites | When enabled, nodes that you have marked as favorite in the Tab menu appear at the top of the list.  
When disabled, the node list is ordered as it appears in the nodes toolbar or alphabetically if you start typing. |
<p>| Clear Weighting | Click to reset any weighting information collected by Nuke. |
| Clear Favorites | Click to reset any nodes marked as favorite. |</p>
<table>
<thead>
<tr>
<th><strong>OFX Plug-ins</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>allow trial mode in OFX plugins</td>
<td>When enabled, OFX plug-ins that offer a trial mode render in that mode, if a license cannot be found. When disabled, OFX plug-ins that can't get a license appear in an error state.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Positions</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>show menus with previous item under the cursor</td>
<td>When enabled, opening contextual menus positions them with the most recently used item under the pointer.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Scripting</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>script command dialog defaults to Tcl</td>
<td>When enabled, the dialog that appears when you press X in the Node Graph defaults to Tcl, rather than Python.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Startup</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>startup workspace</td>
<td>Sets which workspace to display on startup. You can choose from Compositing, Conforming, Editing, Finishing, Reviewing, and Timeline. You can also choose to save a customized workspace, which would also be available from this list.</td>
</tr>
<tr>
<td>show splash screen at startup</td>
<td>When enabled, display the splash screen on startup.</td>
</tr>
<tr>
<td>show startup dialog</td>
<td>When enabled, display the dialog on startup.</td>
</tr>
<tr>
<td>restore workspace when opening projects</td>
<td>When enabled, restore the selected saved workspace when opening projects.</td>
</tr>
</tbody>
</table>

**Analytics**
### Startup

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>submit usage statistics</td>
<td>When enabled, certain usage statistics are collected from the machine on which you license Nuke, NukeX, Nuke Studio, Hiero, and HieroPlayer. When disabled, we don't collect any usage data from your machine.</td>
</tr>
</tbody>
</table>

**Note:** The port number used to communicate with Foundry is 443, the same one used for uploading crash reports.

### Timecode

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>R3D file timecode</td>
<td>Sets the source timecode for RED files. You can choose from Default From File, Absolute Timecode, or Edge Timecode.</td>
</tr>
<tr>
<td>other media timecode</td>
<td>Sets the timecode source for file-per-frame media (such as .dpx). You can choose from File Header or Frame Number. If File Header is selected and a timecode exists, then the timecode is used. Otherwise it defaults back to using the frame number from the file name.</td>
</tr>
<tr>
<td>max valid timebase (fps)</td>
<td>Sets the maximum image header timebase above which the value is clamped.</td>
</tr>
<tr>
<td></td>
<td>Image files are often created with application specific timebase values in the header description. This can lead to reading in spuriously high frame rates and the clamp aims to prevent this from happening.</td>
</tr>
<tr>
<td></td>
<td>If your clips do have extremely high frame rates, increase this value as necessary to avoid clamping.</td>
</tr>
<tr>
<td>EDL style spreadsheet timecodes</td>
<td>When disabled, the srcOut and dstOut values in the spreadsheet use the film convention, representing the last frame of the cut.</td>
</tr>
<tr>
<td></td>
<td>When enabled, the srcOut and dstOut values in the spreadsheet use the video convention, representing the frame directly after the cut.</td>
</tr>
</tbody>
</table>
# Panels

<table>
<thead>
<tr>
<th>Appearance</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Font</td>
<td>Change the type, weight, angle, and size of the font used on Nuke’s user interface.</td>
</tr>
</tbody>
</table>

**UI Colors** - right-click on any color button and select *Set color to default* to revert changes.

<table>
<thead>
<tr>
<th>Background</th>
<th>Change the background color of most user interface elements (menus, toolbars, panes, properties panels, Viewers, and pop-up dialogs).</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base</td>
<td>Change the color of input fields, the input pane of the Script Editor, and the left side of the Curve Editor.</td>
</tr>
<tr>
<td>Highlight</td>
<td>Change the color of the highlighting that appears when you hover the cursor over a control, select a file or folder in the File Browser, or scrub to a new frame on the timeline.</td>
</tr>
<tr>
<td>Highlighted Text</td>
<td>Change the color of any highlighted text (for example, text you select in node properties).</td>
</tr>
<tr>
<td>Label</td>
<td>Change the color of labels and text on the application interface. Note that this does not set the color of the labels on nodes in the Node Graph.</td>
</tr>
<tr>
<td>Button</td>
<td>Change the color of buttons and dropdown menus.</td>
</tr>
<tr>
<td>Animated</td>
<td>Change the color that indicates a control has been animated.</td>
</tr>
<tr>
<td>Keyframe</td>
<td>Change the color that indicates a keyframe has been set.</td>
</tr>
<tr>
<td>Disk cached frames</td>
<td>Change the color of the disk cached frames on the Viewer timeline.</td>
</tr>
<tr>
<td>RAM cached frames</td>
<td>Change the color of the RAM cached frames on the Viewer timeline.</td>
</tr>
<tr>
<td>Playhead</td>
<td>Change the color of the frame marker on the Viewer timeline.</td>
</tr>
<tr>
<td>In/Out Markers</td>
<td>Change the color of the in and out frame markers on the Viewer timeline.</td>
</tr>
</tbody>
</table>

**Curve Editor / Dope Sheet** - right-click on any color button and select *Set color to default* to revert changes.

| no. of curves visible | Sets the maximum number of curves visible in the Curve Editor. |
### Appearance

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>background</td>
<td>Change the background color of the Dope Sheet tab.</td>
</tr>
<tr>
<td>unselected key</td>
<td>Change the color used for an unselected key on the Dope Sheet.</td>
</tr>
<tr>
<td>part-selected key</td>
<td>Change the color used for a part-selected key on the Dope Sheet.</td>
</tr>
<tr>
<td>selected key</td>
<td>Change the color used for a selected key on the Dope Sheet.</td>
</tr>
<tr>
<td>timeline</td>
<td>Change the color used for the timeline on the Dope Sheet.</td>
</tr>
<tr>
<td>control text</td>
<td>Change the color used for the control text on the Dope Sheet. These indicate the frame number of a key when you select one.</td>
</tr>
<tr>
<td>control text shadow</td>
<td>Change the color used for the shadow of the control text on the Dope Sheet.</td>
</tr>
<tr>
<td>time label</td>
<td>Change the color used for the time labels on the Dope Sheet. These indicate frame numbers.</td>
</tr>
<tr>
<td>current frame</td>
<td>Change the color used for the current frame on the Dope Sheet. This is a vertical line that indicates the current frame on the timeline.</td>
</tr>
<tr>
<td>project frame range</td>
<td>Change the color used for the project frame range on the Dope Sheet. These are two vertical lines indicate your frame range.</td>
</tr>
</tbody>
</table>

### Control Panels

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>max nodes in properties bin</td>
<td>Use this to set the maximum number of panels that can be open in the Properties pane.</td>
</tr>
<tr>
<td>reopen acts like new panel</td>
<td>When this is enabled, double-clicking a node that has been open before, places the panel in the same place as a new panel. If this is disabled, the panel appears in its previous position.</td>
</tr>
<tr>
<td>expand / collapse panels in Properties bin to match selection</td>
<td>If this is enabled, the node selection in the Node Graph determines which control panels are expanded (all unselected nodes automatically have their panels collapsed). This does not apply to floating control panels.</td>
</tr>
<tr>
<td>input button action</td>
<td>Use this to define node input button action, which is located in the top-left of the node properties panel. For example, you can set this to center a</td>
</tr>
</tbody>
</table>
### Control Panels

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>selected input of a node in the Node Graph.</td>
<td>Use this to set the maximum number of channels or layers that are displayed in a single sub-menu of the main channel control.</td>
</tr>
</tbody>
</table>

### Color Panel

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>in-panel color picker</td>
<td>Sets the type of color picker displayed when you click a color picker button in the properties panel:</td>
</tr>
<tr>
<td>floating color picker</td>
<td>- opens a color wheel and sliders in the properties panel.</td>
</tr>
<tr>
<td>floating color picker</td>
<td>- opens a color wheel and sliders in a floating panel.</td>
</tr>
</tbody>
</table>

**Tip:** Holding Ctrl/Cmd and clicking the color picker button opens the alternate color picker to the one specified in the Preferences.

### File Browser

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>start file browser from most recently used directory</td>
<td>When enabled, new file browsers open at the last location used. When disabled, new file browsers open at the current working directory.</td>
</tr>
</tbody>
</table>

### Node Colors

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>autocolor</td>
<td>Deselect this checkbox to ignore individual node/soft effect color settings and, instead always use the All Others color.</td>
</tr>
<tr>
<td>Shade Nodes</td>
<td>Select this checkbox to apply a slight gradient shading to nodes.</td>
</tr>
<tr>
<td>&lt;node name or type&gt;</td>
<td>The nodes listed here have been given a default color. You can change this by clicking the assigned color to open the color menu, and selecting a new one.</td>
</tr>
</tbody>
</table>
## Node Colors

| All Others | Use this to select the color to use as default for all nodes not otherwise specified above, or all nodes if autocolor is disabled. |
| Text | Use this to select the color used for node label text. |
| Selected | Use this to choose the highlight color applied to any selected nodes. |
| Selected Input | Use this to choose the color used for node inputs' label text when selected. |
| GL Color | Use this to select the color to draw nodes' Viewer controls in. For example, the Ramp node's gradient control. |

## Node Graph

| autolabel | When disabled, nodes only show the filename or node name - most of the code in autolabel.py is disregarded. For example, the Blur node does not display the affected channels when this control is disabled. |
| highlight running operators | When enabled, highlight any nodes whose output is currently being calculated. |
| postage stamp mode | When displaying a thumbnail render of the node's output on its surface (either using the PostageStamp node or the postage stamp control on the Node tab of each node), you can select one of two modes:  
  - **Current frame** - The postage stamp is always updated to match the current frame.  
  - **Static frame** - The postage stamp displays a fixed frame. To specify the |
### Node Graph

- **Frame to use**, open the node's controls, go to the **Node** tab, and adjust **static frame**.

  - **Note**: If the frame number you use is outside the frame range for the node, it is clamped to the first or last frame in the range accordingly.

<table>
<thead>
<tr>
<th>Node name background</th>
<th>When a node is selected and the node’s name is too long to fit inside the node, a background is drawn behind the name to improve legibility. Use this control to set the intensity of the background, from 0 (no background) to 1 (fully opaque background).</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Label font</strong></td>
<td><strong>Sets the font for labels. You can use the B and I, to the right of the font dropdown, to bold or italicize the selected label font.</strong></td>
</tr>
<tr>
<td><strong>Tile size (WxH)</strong></td>
<td><strong>Sets the size of nodes in the Node Graph using the width and height.</strong></td>
</tr>
<tr>
<td><strong>Snap to node</strong></td>
<td><strong>When enabled, nodes snap into positions (while dragging them) that line up horizontally and vertically with their inputs and outputs.</strong></td>
</tr>
<tr>
<td><strong>Grid size (WxH)</strong></td>
<td><strong>Sets the grid size using width and height.</strong></td>
</tr>
<tr>
<td><strong>Snap to grid</strong></td>
<td><strong>When enabled, nodes snap into positions (while dragging them) that line them up with the grid.</strong></td>
</tr>
<tr>
<td><strong>Show grid</strong></td>
<td><strong>When enabled, display the grid using the overlay color.</strong></td>
</tr>
<tr>
<td><strong>Snap threshold</strong></td>
<td><strong>When <strong>snap to grid</strong> is enabled, use this to set the maximum number of pixels to jump by, when snapping nodes to the grid or other nodes.</strong></td>
</tr>
</tbody>
</table>

### Colors

| **Node Graph** | **Sets the color of the Node Graph background.** |
| **Overlay**    | **Sets the color of the selection marquee when you lasso nodes.** |
| **Elbow**      | **Sets the color of the dots created when you ‘elbow’ a connection pipe by** |
## Node Graph

holding **Ctrl/Cmd**.

## Bounding Box Warning

<table>
<thead>
<tr>
<th><strong>Highlight</strong></th>
<th>When the bounding box warning is enabled, sets the color of the warning displayed on nodes that exceed the threshold.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Line</strong></td>
<td>When the bounding box warning is enabled, sets the color of the dotted line around the <strong>Highlight</strong> color on nodes that exceed the threshold.</td>
</tr>
</tbody>
</table>

**enable**

When enabled, nodes that force the bounding box past the format size are marked in the Node Graph:

- **red rectangle with dotted stroke** - the indicated node creates a bounding box greater than the format.
- **dotted stroke without the red rectangle** - the bounding box size is greater than the format at the indicated node, but the bounding box size has been set by an upstream node.

**threshold %**

Sets the threshold past which the bounding box warning is displayed.

## Arrow

<table>
<thead>
<tr>
<th><strong>&lt;directional arrows&gt;</strong></th>
<th>You can select one of the directional arrows (up, down, left, or right) to change the color it is displayed in. Click the arrow to open the color menu and select a new color.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>deep arrows</strong></td>
<td>Sets the color of arrows carrying deep data. Click the button to open the color menu and select a new color.</td>
</tr>
<tr>
<td><strong>expression arrows</strong></td>
<td>Sets the color of expression arrows, if enabled. Select the <strong>enable</strong> checkbox to display expression arrows.</td>
</tr>
<tr>
<td><strong>link knob arrows</strong></td>
<td>Sets the color of arrows indicating that a node contains linked knobs, that is, knobs that are in use in another node's Properties panel. Select the <strong>enable</strong> checkbox to display link knob arrows.</td>
</tr>
<tr>
<td><strong>clone arrows</strong></td>
<td>Sets the color of clone arrows, if enabled. Select the <strong>enable</strong> checkbox to display clone arrows.</td>
</tr>
<tr>
<td><strong>&lt;arrow components&gt;</strong></td>
<td>Sets arrow and arrow head, lengths and widths. You can also use the numeric field next to each component to enter a specific value.</td>
</tr>
<tr>
<td><strong>Node Graph</strong></td>
<td></td>
</tr>
<tr>
<td>----------------</td>
<td>---------------------------------------------------------------</td>
</tr>
<tr>
<td>allow picking of connected arrow heads</td>
<td>Select this checkbox to be able to pick up and move connected arrow heads.</td>
</tr>
<tr>
<td>allow picking of arrow elbows to create Dots</td>
<td>When enabled, press Ctrl (Cmd on a Mac) on the Node Graph to display yellow “elbows” on the Node Graph arrows and then click on these to insert Dot nodes. If you Ctrl/Cmd+Shift+click on an elbow, the new Dot node is branched off to a new arrow rather than inserted in the existing arrow. When disabled, adding Dot nodes in this manner is not possible.</td>
</tr>
<tr>
<td>drag-to-insert only work near middle of arrows</td>
<td>Select this checkbox to restrict the arrow hotspot, for inserting nodes, to the middle of the arrow.</td>
</tr>
<tr>
<td>size of dots</td>
<td>Use this slider to set the size of the Dot nodes. You can also enter a specific value in the numeric field to the left of the slider.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Project Items</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>shade project items</td>
<td>When enabled, additional shading is applied to source clips and shots in the Project bin and timeline.</td>
</tr>
</tbody>
</table>
### Project Items

#### Item Labels

<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>project bin</td>
<td>Click to change the color of labels in the <strong>Project</strong> and timeline panels. A color wheel displays allowing you to select the required color.</td>
</tr>
<tr>
<td>timeline</td>
<td></td>
</tr>
<tr>
<td>auto-adjust contrast</td>
<td>When enabled, label colors are automatically adjusted if a potential color-clash is detected.</td>
</tr>
</tbody>
</table>

#### Item States

<table>
<thead>
<tr>
<th>State</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>offline</td>
<td>Click the buttons to change the color of shots and comps in the timeline panel. A color wheel displays allowing you to select the required color.</td>
</tr>
<tr>
<td>error</td>
<td></td>
</tr>
<tr>
<td>freeze</td>
<td></td>
</tr>
<tr>
<td>comp not rendered</td>
<td></td>
</tr>
<tr>
<td>comp out of date</td>
<td></td>
</tr>
<tr>
<td>comp rendered</td>
<td></td>
</tr>
</tbody>
</table>

#### Item Colors

<table>
<thead>
<tr>
<th>Display Location</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>display in project panel</td>
<td>When enabled, the specified item colors, or defaults, are displayed in the <strong>Project</strong> panel.</td>
</tr>
<tr>
<td>display in sequence panel</td>
<td>When enabled, the specified item colors, or defaults, are displayed in the timeline panel.</td>
</tr>
<tr>
<td>spreadsheet color rows</td>
<td>When enabled, the specified item colors applied to rows in the spreadsheet.</td>
</tr>
</tbody>
</table>
## Project Items

<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>project</td>
<td>Click the buttons to change the color of items in the <strong>Project</strong> and timeline panels. A color wheel displays allowing you to select the required color.</td>
</tr>
<tr>
<td>bin</td>
<td></td>
</tr>
<tr>
<td>sequence</td>
<td></td>
</tr>
<tr>
<td>source</td>
<td></td>
</tr>
<tr>
<td>audio</td>
<td></td>
</tr>
<tr>
<td>comp</td>
<td></td>
</tr>
<tr>
<td>file types</td>
<td>Allows you to add custom color-coding by file extension. Click the button and then select a file type from the <strong>extension</strong> dropdown. Double-click the color swatch to display a color wheel allowing you to select the required color. Any source clip or shot with that extension is then colored accordingly in the interface.</td>
</tr>
</tbody>
</table>

![Color wheel and file types example]

## Scopes

<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>black point</td>
<td>Use the slider or the entry box to select the black point value.</td>
</tr>
<tr>
<td>white point</td>
<td>Use the slider or the entry box to select the white point value.</td>
</tr>
<tr>
<td>luma/chroma encoding</td>
<td>Use this to select the video standard to use when converting from RGB to luma and chroma for scope display.</td>
</tr>
<tr>
<td>Include viewer color</td>
<td>Select this checkbox to include applied Viewer color transforms (gain,</td>
</tr>
</tbody>
</table>
### Scopes

<table>
<thead>
<tr>
<th>Transform</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>gamma, and LUT</td>
<td>in scope data. If this checkbox is disabled, all Viewer transforms are ignored.</td>
</tr>
</tbody>
</table>

**Note:** If disabled, rendering may become slow as image calculation may be needed.

**Force full frame**

Select this checkbox so that the Viewer always requests full frame data when a scope is displaying data for that Viewer. If this checkbox is disabled, the scopes only display data for the current area requested by the Viewer, rather than the whole image.

### Script Editor

<table>
<thead>
<tr>
<th>Font</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Font</td>
<td>Use this to select the font to use in the Script Editor.</td>
</tr>
</tbody>
</table>

**Note:** This control also changes the font in the BlinkScript Kernel Source field.

**Indent**

You can use this control to set the indent value to use when scripting.

**Save and restore script editor history**

Disable this checkbox if you prefer that the contents of the Script Editor is not saved and restored between sessions of Nuke.

**Echo python comments to output window**

Select this checkbox to print any Python commands executed by Nuke itself to the Script Editor output window.

**Note:** Note that not everything you do results in a command being echoed, because many of Nuke's internal functions are not executed using Python commands.

**Clear input window on successful script execution**

Disable this checkbox if you want the most recent script to remain in the input window after execution.
### Timeline

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>show frame end marker</td>
<td>When enabled, an extra line is drawn on the timeline to the right of the playhead, indicating the end of the current frame.</td>
</tr>
<tr>
<td>visible range follows playhead</td>
<td>When enabled, the timeline scrolls with the playhead, constantly updating the view. When disabled, the playhead is allowed to move off screen.</td>
</tr>
</tbody>
</table>

### Audio Tracks

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>half waveforms</td>
<td>When enabled, audio tracks on the timeline display only the rectified waveform. When disabled, the full waveform is displayed.</td>
</tr>
</tbody>
</table>

### Viewer (Comp)

#### Defaults

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>new Viewers go to own window</td>
<td>When enabled, new Viewers are placed in their own window instead of docking in existing windows.</td>
</tr>
<tr>
<td>prevent auto zoom for new Viewers</td>
<td>When enabled, new Viewers are not auto-zoomed to the current Viewer's zoom level.</td>
</tr>
<tr>
<td>apply LUT to color channels only</td>
<td>When this is checked, look-up tables (LUTs) are only applied to the red, green, and blue channels.</td>
</tr>
<tr>
<td></td>
<td>When this is NOT checked, LUTs are applied to all channels.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>viewer buffer bit depth (byte)</th>
<th>Use this to select the OpenGL buffer depth, and enable use of the GPU for the Viewer process and input process.</th>
</tr>
</thead>
<tbody>
<tr>
<td>• <strong>byte</strong></td>
<td>Converts to 8-bit with error diffusion.</td>
</tr>
<tr>
<td>• <strong>half-float</strong></td>
<td>Converts to 16-bit (half) float. In this mode, the GPU can be used to apply Viewer effects like gamma and the LUT in a Viewer process.</td>
</tr>
<tr>
<td>• <strong>float</strong></td>
<td>Uses a full 32-bit floating point texture. This may be slow on selected cards. In this mode, the GPU can be used to apply Viewer effects like gamma and the LUT in a Viewer process.</td>
</tr>
</tbody>
</table>

You can choose a default value for this setting in the Preferences, or by using `knobDefault()` in a startup script.
### Viewer (Comp)

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>use GPU for Viewer when possible</td>
<td>When this is checked, the Viewer applies its effects (such as the Viewer Process node) in the GPU when possible. However, in some cases, like when monitor output is enabled or <em>gl buffer depth</em> is set to <em>byte</em> in the Viewer settings, effects (such as gain and gamma) must still be computed on the CPU.</td>
</tr>
<tr>
<td>use GPU for inputs when possible</td>
<td>Normally, the Viewer only attempts to run its own effects (such as the Viewer Process node) on the GPU. However, when this is checked, any nodes connected to the Viewer are also computed on the GPU when possible. Note that this cannot be done for all nodes because not all nodes have a GPU implementation.</td>
</tr>
<tr>
<td>disable GPU Viewer dithering</td>
<td>Check this to disable dithering in the Viewer when you're using half-float depth. Uncheck to allow dithering at all times.</td>
</tr>
<tr>
<td>no incomplete stereo for new viewers</td>
<td>When this is checked, the Viewer only displays one view of a stereo project until both views have been rendered. This is to prevent disorienting effects when watching the results.</td>
</tr>
<tr>
<td></td>
<td>When this is not checked, the Viewer displays both stereo views, even if the render of either is incomplete.</td>
</tr>
</tbody>
</table>

**Note:** When the Viewer AB mode is changed to wipe or stack, the state of the GPU acceleration controls is stored, GPU acceleration is turned off, and GPU acceleration is disabled.

When the Viewer AB mode is changed back to default, GPU acceleration is re-enabled and the state of the GPU acceleration controls is restored.

### Settings

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>flip stereo interlaced views</td>
<td>Select this checkbox to flip the left and right stereo views when the Viewer is set to <em>Stereo Mode &gt; Interlaced</em> in the right-click/context menu.</td>
</tr>
</tbody>
</table>
Appendix A: Preferences | Panels

Viewer (Comp)
texture size

Use this to define the OpenGL texture size in Nuke's Viewers. It affects the
2D Viewer preview texture size, such as the preview displayed when
dragging transform or SplineWarp handles, and all textures in the 3D
Viewer. You can choose from 256x256, 512x512, 1024x1024, and
2048x2048.

Note: The larger the texture size, the sharper it is, but larger
texture sizes need more time and memory. Setting this
preference to a high value can impact 3D Viewer performance.
texture mode

Use this to choose how textures are handled in the Viewer. You can choose
from the following options:
• Multiframe – Cache each frame of a texture; this gives animated textures
and fast playback after each frame is cached, but uses a lot more
memory. It also enables you to have multiple frames of a texture visible
at one time (for example, on particles).
• Classic – Textures are not updated in the Viewer during playback; this
gives the fastest playback speed.

Viewer (Monitor Out)
use video legal range
for monitor out

When enabled, automatically limit monitor output to the legal range when
swapping between the Timeline environment and Compositing
environment.

Viewer (Sequence) or (Flipbook)
playback mode

Use this to set the Viewer playback mode:
• Play All Frames - the default setting, plays all frames in real-time
(dependent on hardware).
• Skip Frames - plays frames in real-time skipping where necessary to

USER GUIDE
1974


### Viewer (Sequence) or (Flipbook)

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>maintain the frame rate.</td>
<td></td>
</tr>
<tr>
<td><strong>Play All Frames, Buffering</strong></td>
<td>plays all frames by buffering and playing frames back as they become available.</td>
</tr>
<tr>
<td>guides</td>
<td>You can use this to choose to show overlays in the image area. You can choose from:</td>
</tr>
<tr>
<td><strong>Title Safe</strong></td>
<td>Indicates where text should be entered to be visible.</td>
</tr>
<tr>
<td><strong>Action Safe</strong></td>
<td>Indicates the area in which to place actions so that they are visible.</td>
</tr>
<tr>
<td><strong>Format</strong></td>
<td>Displays the size of the format over the Viewer.</td>
</tr>
<tr>
<td>fullscreen display</td>
<td>Use this to select which display to use for fullscreen mode. This setting takes effect the next time fullscreen mode is entered.</td>
</tr>
<tr>
<td>see through missing media</td>
<td>Select this checkbox to see through missing media in the timeline, displaying the first displayable media in the underlying tracks.</td>
</tr>
<tr>
<td>background</td>
<td>Use this to select the Viewer background. You can select black, or gray (using the slider to determine the grayscale), or checkerboard (using the slider to determine the size of the squares).</td>
</tr>
<tr>
<td>frame increment</td>
<td>Use this to set the default number of frames skipped by the Viewer skip controls, and the timeline Nudge More commands.</td>
</tr>
<tr>
<td>filtering mode</td>
<td>Use this to determine the filtering used during rendering in the Timeline environment. You can select <strong>Auto, Nearest neighbour</strong>, or <strong>Linear</strong>.</td>
</tr>
<tr>
<td></td>
<td><strong>Auto</strong> uses the same automatic selection as in the Compositing environment. This does not affect exports or rendering in the Compositing environment.</td>
</tr>
</tbody>
</table>

### Audio

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>default latency adjustment (ms)</td>
<td>Use this to adjust the default timing offset (in milliseconds) between audio and video to apply to new Viewers. Positive values make audio play earlier relative to video; negative values make audio play later. To convert from video frames to ms, divide 1000 ms by the video frame rate. For example:</td>
</tr>
<tr>
<td></td>
<td>• at 25fps, a video frame is 1000/25 = 40ms, or</td>
</tr>
<tr>
<td></td>
<td>• a 1.5 video frame delay = 1.5 \times 40ms = 60ms.</td>
</tr>
<tr>
<td>default volume</td>
<td>Use the slider or numeric field to set the default volume.</td>
</tr>
</tbody>
</table>
### Viewer Handles

#### Colors

<table>
<thead>
<tr>
<th>2D</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>bg</strong></td>
<td>Change the background color of the 2D Viewer.</td>
</tr>
<tr>
<td><strong>fg</strong></td>
<td>Change the color of borders and text in the 2D Viewer.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>3D</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>bg</strong></td>
<td>Change the background color of the 3D Viewer.</td>
</tr>
<tr>
<td><strong>fg</strong></td>
<td>Change the color of borders and text in the 3D Viewer.</td>
</tr>
<tr>
<td><strong>sel</strong></td>
<td>Change the color of the selected vertices or faces of an object in the 3D viewer.</td>
</tr>
</tbody>
</table>

#### Splines

| line width | Sets the width of splines drawn in RotoPaint and SplineWarp. |
| draw shadows | When enabled, shadows are added to splines drawn in RotoPaint and SplineWarp to make the overlay easier to see. |

#### general

| **Expression color** | Change the default color of the control points when an expression is set. |
| **Focus color** | Change the default color of the control points when focused. |

#### roto

| **Points** | Change the default color of the points on RotoPaint shapes and strokes. |
| **Curves** | Change the default color of the roto shape and stroke curves in RotoPaint and Roto. |
| **Transform** | Change the default color of the RotoPaint transform jack. |
| **Locked** | Change the default color of RotoPaint points and curves when locked or otherwise unmodifiable. |

#### splinewarp

| **A Sourcecolor** | Change the default color of SplineWarp's A source curves. |
| **B Sourcecolor** | Change the default color of SplineWarp's B source curves. |
| **draw source stippled** | Check to change source curves from solid to stippled. |
| **A Destinationcolor** | Change the default color of SplineWarp's A source curves |
### Viewer Handles

<table>
<thead>
<tr>
<th></th>
<th>Destination curves.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>B Destinationcolor</strong></td>
<td>Change the default color of SplineWarp's B destination curves.</td>
</tr>
<tr>
<td><strong>draw destination stippled</strong></td>
<td>Check to change destination curves from solid to stippled.</td>
</tr>
<tr>
<td><strong>Correspondence color</strong></td>
<td>Change the default color of the SplineWarp correspondence lines.</td>
</tr>
<tr>
<td><strong>Boundary color</strong></td>
<td>Change the default color of the SplineWarp boundary curves.</td>
</tr>
<tr>
<td><strong>Hard boundary color</strong></td>
<td>Change the default color of the SplineWarp hard boundary curves.</td>
</tr>
</tbody>
</table>

**Note:** To set a color back to its default, right-click on the button and select **Set Color to Default**.

### Controls

<table>
<thead>
<tr>
<th>Control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>middle button pans</td>
<td>Check this to use the middle-mouse button to pan in the Viewer, Node Graph, the Curve Editor, and the Dope Sheet.</td>
</tr>
<tr>
<td>left-middle to zoom</td>
<td>Check this to use the left and the middle-mouse button together to zoom in the Viewer, Node Graph, the Curve Editor, and the Dope Sheet.</td>
</tr>
<tr>
<td>show transform preview</td>
<td>Check this to disable the OpenGL preview when manipulating the handles of 2D nodes, such as Transform and CornerPin.</td>
</tr>
<tr>
<td>3D control type</td>
<td>Select the navigation control scheme you want to use in the 3D Viewer: Nuke, Maya, Houdini, Lightwave, or Modo.</td>
</tr>
<tr>
<td>2D handle size</td>
<td>Adjust the size of the square control handles that appear on the Viewer for some operations, such as transformations, warps, and Bezier and B-spline shapes.</td>
</tr>
</tbody>
</table>
### Viewer Handles

By default, this value is set to 5. You can also set the pickable area size of the square control handles in the numeric field or slider to the right of the **2D handle size** control.

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>3D handle size</strong></td>
<td>Adjust the size of the square control handles that appear when you’re, for instance, selecting vertices on a 3D object in the 3D view. By default, this value is set to 5. You can also set the pickable area size of the square control handles in the numeric field or slider to the right of the <strong>3D handle size</strong> control.</td>
</tr>
<tr>
<td><strong>icon size</strong></td>
<td>Adjust the size of the 2D transformation overlay, 3D camera, 3D object normals, and 3D axis on the Viewer. By default, this value is set to 50.</td>
</tr>
<tr>
<td><strong>icon scaling</strong></td>
<td>Adjust how much the scale of display affects the size of the 2D transformation overlay, 3D camera, and 3D axis. When this is set to 0, these controls are always drawn the same size, regardless of the zoom level. When the value is set to 1, the controls scale with the displayed image or 3D scene when you zoom in or out. Intermediate values mix this so that the controls do scale, but not as much as the image does. This gives an optical illusion that you are zooming in or out without making the controls unusably small or large.</td>
</tr>
<tr>
<td><strong>object interaction speed</strong></td>
<td>Set how fast mouse movements rotate and translate 3D axis and cameras. The lower the value, the finer the movements. The default value is 1.</td>
</tr>
<tr>
<td><strong>camera interaction speed</strong></td>
<td>Set how fast mouse movements tumble and roll the 3D view in the Viewer. The lower the value, the finer the movements. The default value is 1.</td>
</tr>
</tbody>
</table>
Appendix B: Keyboard Shortcuts

Keyboard shortcuts, or hotkeys, provide quick access to the features of Nuke. The following tables show these keystrokes.

Conventions

The following conventions apply to instructions for mouse-clicks and key presses.

• LMB means click or press the left mouse button.
• MMB means click or press the middle mouse button
• RMB means click or press the right mouse button.
• When you see the word “drag” after a mouse button abbreviation (i.e., “MMB drag”), this tells you to press and hold the mouse button while dragging the mouse pointer.
• Keystroke combinations with the Ctrl, Alt, and Shift keys tell you to press and hold the key and then type the specified letter.

For example, “Press Ctrl+S” means hold down the Ctrl key, press S, and then release both keys.

Note: On Mac, replace the Ctrl key with the Cmd key.

Note: Keystrokes in the tables appear in upper case, but you do not type them as upper case. If the Shift+modifier does not appear before the letter, just press the letter key alone.

Note: This section assumes you are using the default keyboard and mouse-button assignments. If the mouse buttons do not work for you as described here, try resetting the mouse control type back to the standard Nuke setting (Preferences > Panels > Viewer Handles > 3D control type > Nuke).
### Global

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Backspace/Delete</td>
<td>Delete selected clips or folders</td>
</tr>
<tr>
<td>F12</td>
<td>Clear buffers and playback cache</td>
</tr>
<tr>
<td>MMB drag</td>
<td>Virtual slider (number fields)</td>
</tr>
<tr>
<td>Space bar (short press)</td>
<td>Expand the focused panel to the full window</td>
</tr>
<tr>
<td>Space bar (long press)</td>
<td>Raise the right-click menu</td>
</tr>
<tr>
<td>Alt+S</td>
<td>Make the application or floating window fullscreen.</td>
</tr>
<tr>
<td>Alt+`</td>
<td>Show Curve Editor.</td>
</tr>
<tr>
<td>Ctrl+A</td>
<td>Select all</td>
</tr>
<tr>
<td>Ctrl+C</td>
<td>Copy selected item(s)</td>
</tr>
<tr>
<td>Ctrl+D</td>
<td>Duplicate selected item(s)</td>
</tr>
<tr>
<td>Ctrl+F#</td>
<td>Save current window layout. The # represents a function key number, F1</td>
</tr>
<tr>
<td>Ctrl+LMB on panel name</td>
<td>Float panel</td>
</tr>
<tr>
<td>Ctrl+N</td>
<td>Create a new project or script, depending on environment</td>
</tr>
<tr>
<td>Ctrl+O</td>
<td>Open a project or script, depending on environment</td>
</tr>
<tr>
<td>Ctrl+Q</td>
<td>Exit the application</td>
</tr>
<tr>
<td>Ctrl+S</td>
<td>Save current project or script, depending on environment</td>
</tr>
<tr>
<td>Ctrl+T</td>
<td>Cycle through tabs in the current pane. Note that this does not work if</td>
</tr>
<tr>
<td></td>
<td>the focus is on the input pane of the Script Editor</td>
</tr>
<tr>
<td>Ctrl+V</td>
<td>Paste the contents of the clipboard</td>
</tr>
<tr>
<td>Keystroke(s)</td>
<td>Action</td>
</tr>
<tr>
<td>-------------------</td>
<td>------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Ctrl+W</td>
<td>Close the current project or script, dependent on environment</td>
</tr>
<tr>
<td>Ctrl+X</td>
<td>Cut selected item(s)</td>
</tr>
<tr>
<td>Ctrl+Z</td>
<td>Undo last action</td>
</tr>
<tr>
<td>Shift+Esc</td>
<td>Close the current tab</td>
</tr>
<tr>
<td>Shift+F1..F6</td>
<td>Change workspace</td>
</tr>
<tr>
<td>Shift+S</td>
<td>Open the Preferences dialog</td>
</tr>
<tr>
<td>Alt+Shift+1..6</td>
<td>Open a recent project or script, depending on environment</td>
</tr>
<tr>
<td>Ctrl+Shift+A</td>
<td>Select none</td>
</tr>
<tr>
<td>Ctrl+Alt+`</td>
<td>Goto next pane</td>
</tr>
<tr>
<td>Ctrl+Shift+[</td>
<td>Goto next tab</td>
</tr>
<tr>
<td>Ctrl+Shift+]</td>
<td>Goto previous tab</td>
</tr>
<tr>
<td>Ctrl+Shift+S</td>
<td>Save current project or script, depending on environment, and specify name (Save As)</td>
</tr>
<tr>
<td>Ctrl+Shift+Z</td>
<td>Redo last action</td>
</tr>
<tr>
<td>Ctrl+Alt+Shift+`</td>
<td>Goto previous pane</td>
</tr>
</tbody>
</table>

**Nuke Studio's Timeline Viewer**

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>-</td>
<td>Zoom Out</td>
</tr>
<tr>
<td>+</td>
<td>Zoom In</td>
</tr>
<tr>
<td>A</td>
<td>Alpha</td>
</tr>
<tr>
<td>B</td>
<td>Blue</td>
</tr>
<tr>
<td>Keystroke(s)</td>
<td>Action</td>
</tr>
<tr>
<td>-------------</td>
<td>----------------------------------</td>
</tr>
<tr>
<td>C</td>
<td>Razor selected</td>
</tr>
<tr>
<td>E</td>
<td>Clipping Warning</td>
</tr>
<tr>
<td>End</td>
<td>Go to End</td>
</tr>
<tr>
<td>F</td>
<td>Zoom to Fit</td>
</tr>
<tr>
<td>F12</td>
<td>Clear playback cache</td>
</tr>
<tr>
<td>G</td>
<td>Green</td>
</tr>
<tr>
<td>H</td>
<td>Zoom to Fill</td>
</tr>
<tr>
<td>Home</td>
<td>Go to Start</td>
</tr>
<tr>
<td>I</td>
<td>Mark In point</td>
</tr>
<tr>
<td>J</td>
<td>Play Backward</td>
</tr>
<tr>
<td>K</td>
<td>Pause</td>
</tr>
<tr>
<td>L</td>
<td>Play Forward</td>
</tr>
<tr>
<td>Left Arrow</td>
<td>Frame Backwards</td>
</tr>
<tr>
<td>O</td>
<td>Mark Out point</td>
</tr>
<tr>
<td>PgDown</td>
<td>Next Layer</td>
</tr>
<tr>
<td>PgUp</td>
<td>Previous Layer</td>
</tr>
<tr>
<td>Q</td>
<td>Show Overlays</td>
</tr>
<tr>
<td>R</td>
<td>Red</td>
</tr>
<tr>
<td>Return</td>
<td>Swap A/B Inputs</td>
</tr>
<tr>
<td>Right Arrow</td>
<td>Frame Forwards</td>
</tr>
<tr>
<td>V</td>
<td>Display the version selector</td>
</tr>
<tr>
<td>W</td>
<td>Wipe</td>
</tr>
<tr>
<td>Y</td>
<td>Luma</td>
</tr>
<tr>
<td>Keystroke(s)</td>
<td>Action</td>
</tr>
<tr>
<td>------------------------------</td>
<td>---------------------------------------------</td>
</tr>
<tr>
<td>Alt+Down Arrow</td>
<td>Version down the selected clip</td>
</tr>
<tr>
<td>Alt+I</td>
<td>Clear In Point</td>
</tr>
<tr>
<td>Alt+O</td>
<td>Clear Out Point</td>
</tr>
<tr>
<td>Alt+Shift+Left Arrow</td>
<td>Previous Tag</td>
</tr>
<tr>
<td>Alt+Shift+Right Arrow</td>
<td>Next Tag</td>
</tr>
<tr>
<td>Alt+U</td>
<td>Clear In/Out Points</td>
</tr>
<tr>
<td>Alt+Up Arrow</td>
<td>Version up the selected clip</td>
</tr>
<tr>
<td>Ctrl+/</td>
<td>Show Timeline Editor</td>
</tr>
<tr>
<td>Ctrl+F</td>
<td>Full Screen</td>
</tr>
<tr>
<td>Ctrl+LMB</td>
<td>Color picker (single pixel)</td>
</tr>
<tr>
<td>Ctrl+RMB</td>
<td>Deselect sampled pixels</td>
</tr>
<tr>
<td>Ctrl+S</td>
<td>Save current project</td>
</tr>
<tr>
<td>Shift+C</td>
<td>Razor all under playhead</td>
</tr>
<tr>
<td>Shift+I</td>
<td>Go to In Point</td>
</tr>
<tr>
<td>Shift+Left Arrow</td>
<td>Skip Backwards</td>
</tr>
<tr>
<td>Shift+O</td>
<td>Go to Out Point</td>
</tr>
<tr>
<td>Shift+Right Arrow</td>
<td>Skip Forwards</td>
</tr>
<tr>
<td>Ctrl+Alt+Down Arrow</td>
<td>Go to min version</td>
</tr>
<tr>
<td>Ctrl+Alt+Up Arrow</td>
<td>Go to max version</td>
</tr>
<tr>
<td>Ctrl+Shift+1</td>
<td>Zoom to Actual Size</td>
</tr>
<tr>
<td>Ctrl+Shift+2</td>
<td>Zoom to Half Size</td>
</tr>
<tr>
<td>Ctrl+Shift+F</td>
<td>Full Quality 1:1</td>
</tr>
<tr>
<td>Ctrl+Shift+LMB</td>
<td>Color picker (region of pixels)</td>
</tr>
</tbody>
</table>
## Keystroke(s) | Action
--- | ---
Ctrl+Shift+P | Ignore Pixel Aspect
Ctrl+Shift+S | Save current project and specify name (Save As)

### Nuke Studio's Timeline

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>, (comma)</td>
<td>Nudge selected shot(s) left, where space is available</td>
</tr>
<tr>
<td>. (period)</td>
<td>Nudge selected shot(s) right, where space is available</td>
</tr>
<tr>
<td>1</td>
<td>Display in the A input buffer</td>
</tr>
<tr>
<td>2</td>
<td>Display in the B input buffer</td>
</tr>
<tr>
<td>Alt and drag</td>
<td>Ripple and duplicate the dragged shot</td>
</tr>
<tr>
<td>Alt then drag</td>
<td>Duplicate the dragged shot</td>
</tr>
<tr>
<td>D</td>
<td>Enable or disable the selected shot(s)</td>
</tr>
<tr>
<td>Down Arrow</td>
<td>Next Edit</td>
</tr>
<tr>
<td>drag then Alt</td>
<td>Activate Ripple mode while dragging shot</td>
</tr>
<tr>
<td>E</td>
<td>Cycles between the Slip Clip and Slide Clip tools</td>
</tr>
<tr>
<td>Enter (numeric keypad)</td>
<td>Edit playhead time</td>
</tr>
<tr>
<td>F12</td>
<td>Clear playback cache</td>
</tr>
<tr>
<td>F5</td>
<td>Render all Comp containers</td>
</tr>
<tr>
<td>F7</td>
<td>Render selected Comp containers</td>
</tr>
<tr>
<td>LMB</td>
<td>Select a clip including any linked tracks</td>
</tr>
<tr>
<td>M</td>
<td>Insert the contents of a source Viewer into the timeline at the current playhead position overwriting existing shots</td>
</tr>
<tr>
<td>Keystroke(s)</td>
<td>Action</td>
</tr>
<tr>
<td>----------------------</td>
<td>------------------------------------------------------------------------</td>
</tr>
<tr>
<td>N</td>
<td>Insert the contents of a source Viewer into the timeline at the current playhead position and ripple existing shots downstream to accommodate the change</td>
</tr>
<tr>
<td>Numeric keypad arrows</td>
<td>Change selection between shots</td>
</tr>
<tr>
<td>Q</td>
<td>Cycles between the available move tools: Multi, Move/Trim, and Select</td>
</tr>
<tr>
<td>R</td>
<td>Cycles between the available edit tools: Ripple, Roll, and Retime</td>
</tr>
<tr>
<td>T</td>
<td>Cycles between the available razor tools: Razor, Razor All, and Join</td>
</tr>
<tr>
<td>U</td>
<td>Mark clip</td>
</tr>
<tr>
<td>Up Arrow</td>
<td>Previous Edit</td>
</tr>
<tr>
<td>V</td>
<td>Display the version selector</td>
</tr>
<tr>
<td>W</td>
<td>Cycles between the available selection tools</td>
</tr>
<tr>
<td>Shift+drag clip</td>
<td>Disable snap to transition when dragging clips</td>
</tr>
<tr>
<td>Alt+, (comma)</td>
<td>Nudge selected shot(s) up, overwriting any clips on the tracks above</td>
</tr>
<tr>
<td>Alt+, (period)</td>
<td>Nudge selected shot(s) down, overwriting any clips on the tracks below</td>
</tr>
<tr>
<td>Alt+D</td>
<td>Display metadata for the selected track item(s)</td>
</tr>
<tr>
<td>Alt+Down Arrow</td>
<td>Version down the selected shot</td>
</tr>
<tr>
<td>Alt+LMB</td>
<td>Select a clip, ignoring linked tracks (for example, audio only)</td>
</tr>
<tr>
<td>Alt+Up Arrow</td>
<td>Version up the selected shot</td>
</tr>
<tr>
<td>Ctrl+numeric keypad arrows</td>
<td>Nudge selected shot(s), where space is available</td>
</tr>
<tr>
<td>Ctrl+A</td>
<td>Select all shots</td>
</tr>
<tr>
<td>Ctrl+I</td>
<td>Import files</td>
</tr>
<tr>
<td>Ctrl+Return</td>
<td>Open selected shot in the Viewer</td>
</tr>
<tr>
<td>Ctrl+T</td>
<td>Add a dissolve between two selected shots</td>
</tr>
<tr>
<td>Keystroke(s)</td>
<td>Action</td>
</tr>
<tr>
<td>-------------------</td>
<td>------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Shift+, (comma)</td>
<td>Nudge selected shot(s) left by the increment amount set under the Viewer, where space is available</td>
</tr>
<tr>
<td>Shift+. (period)</td>
<td>Nudge selected shot(s) right by the increment amount set under the Viewer, where space is available</td>
</tr>
<tr>
<td>Shift+Backspace</td>
<td>Ripple delete</td>
</tr>
<tr>
<td>Shift+U</td>
<td>Mark selection</td>
</tr>
<tr>
<td>Alt+Shift+/</td>
<td>Rename shots</td>
</tr>
<tr>
<td>Ctrl+Alt+A</td>
<td>Select all in track</td>
</tr>
<tr>
<td>Alt+Shift+Down Arrow</td>
<td>Go to min version</td>
</tr>
<tr>
<td>Alt+Shift+Up Arrow</td>
<td>Go to max version</td>
</tr>
<tr>
<td>Ctrl+Alt+Return</td>
<td>Open selected shot in a new Viewer</td>
</tr>
<tr>
<td>Ctrl+Shift+A</td>
<td>Deselect all shots</td>
</tr>
<tr>
<td>Ctrl+Shift+E</td>
<td>Open the Export dialog</td>
</tr>
<tr>
<td>Ctrl+Shift+I</td>
<td>Import folders</td>
</tr>
<tr>
<td>LMB then Shift+LMB</td>
<td>Select all clips between the left-clicks</td>
</tr>
<tr>
<td>Shift+Alt+LMB</td>
<td>Ignore linked tracks during selection</td>
</tr>
<tr>
<td>Ctrl+Alt+Shift+I</td>
<td>Import EDL, XML, or AAF</td>
</tr>
</tbody>
</table>

**2D Compositing Viewer**

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>-</td>
<td>Zoom Out</td>
</tr>
<tr>
<td>Keystroke(s)</td>
<td>Action</td>
</tr>
<tr>
<td>-------------</td>
<td>---------------------------------------------</td>
</tr>
<tr>
<td>, (comma)</td>
<td>Decrease Gain</td>
</tr>
<tr>
<td>; (semi-colon)</td>
<td>Previous view (multi-view)</td>
</tr>
<tr>
<td>. (period)</td>
<td>Increase Gain</td>
</tr>
<tr>
<td>' (apostrophe)</td>
<td>Next view (multi-view)</td>
</tr>
<tr>
<td>[</td>
<td>Toggle left toolbar</td>
</tr>
<tr>
<td>]</td>
<td>Toggle right toolbar</td>
</tr>
<tr>
<td>` (backtick)</td>
<td>Toggle floating viewers</td>
</tr>
<tr>
<td>+</td>
<td>Zoom In</td>
</tr>
<tr>
<td>1, 2, etc.</td>
<td>Set Viewer inputs for A buffer</td>
</tr>
<tr>
<td>A</td>
<td>Toggles between the Alpha channel and RBG</td>
</tr>
<tr>
<td>B</td>
<td>Toggles between the Blue channel and RBG</td>
</tr>
<tr>
<td>Down Arrow</td>
<td>Switch to the previous input for the A buffer</td>
</tr>
<tr>
<td>End</td>
<td>Go to the out point in the time slider</td>
</tr>
<tr>
<td>F or MMB</td>
<td>Zoom to Fit</td>
</tr>
<tr>
<td>G</td>
<td>Toggles between the Green channel and RBG</td>
</tr>
<tr>
<td>H</td>
<td>Zoom to Fill</td>
</tr>
<tr>
<td>Home</td>
<td>Go to the in point in the time slider</td>
</tr>
<tr>
<td>I</td>
<td>Mark In point</td>
</tr>
<tr>
<td>J</td>
<td>Play Backward</td>
</tr>
<tr>
<td>K</td>
<td>Pause</td>
</tr>
<tr>
<td>L</td>
<td>Play Forward</td>
</tr>
<tr>
<td>Left Arrow</td>
<td>Step backward one frame</td>
</tr>
<tr>
<td>M</td>
<td>Toggles between the Matte channel and RBG</td>
</tr>
<tr>
<td>Keystroke(s)</td>
<td>Action</td>
</tr>
<tr>
<td>------------------------</td>
<td>-------------------------------------------------------------</td>
</tr>
<tr>
<td>Numeric pad left/right arrow keys</td>
<td>Nudge one frame backward or forward</td>
</tr>
<tr>
<td>O</td>
<td>Mark Out point</td>
</tr>
<tr>
<td>P</td>
<td>Pause Viewer refreshing</td>
</tr>
<tr>
<td>PgDown</td>
<td>Next layer</td>
</tr>
<tr>
<td>PgUp</td>
<td>Previous layer</td>
</tr>
<tr>
<td>Q</td>
<td>Toggle overlays</td>
</tr>
<tr>
<td>R</td>
<td>Toggles between the Red channel and RBG</td>
</tr>
<tr>
<td>Return</td>
<td>Swap A/B input buffers</td>
</tr>
<tr>
<td>Right Arrow</td>
<td>Step forward one frame</td>
</tr>
<tr>
<td>S</td>
<td>Open Viewer settings</td>
</tr>
</tbody>
</table>

**Note:** You can't use the S keyboard shortcut to open Viewer properties when Roto or RotoPaint properties are open.

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tab</td>
<td>Toggle 2D/3D</td>
</tr>
<tr>
<td>U</td>
<td>Update the Viewer</td>
</tr>
<tr>
<td>Up Arrow</td>
<td>Switch to the next input for the A buffer</td>
</tr>
<tr>
<td>W</td>
<td>Toggle wipe tool</td>
</tr>
<tr>
<td>Y</td>
<td>Toggles between the Luminance channel and RBG</td>
</tr>
<tr>
<td>Alt+# (1, 2, etc.)</td>
<td>Zoom out by 100%, 200% and so on</td>
</tr>
<tr>
<td>Alt+G</td>
<td>Go to a specific frame</td>
</tr>
<tr>
<td>Alt+I</td>
<td>Clear In point</td>
</tr>
<tr>
<td>Alt+LMB</td>
<td>Pan</td>
</tr>
<tr>
<td>Keystroke(s)</td>
<td>Action</td>
</tr>
<tr>
<td>---------------------------</td>
<td>------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Alt+MMB and drag</td>
<td>Zoom in/Out at the pointer location</td>
</tr>
<tr>
<td>Alt+O</td>
<td>Clear Out point</td>
</tr>
<tr>
<td>Alt+P</td>
<td>Toggle Input Process</td>
</tr>
<tr>
<td>Alt+U</td>
<td>Clear In and Out points</td>
</tr>
<tr>
<td>Alt+W</td>
<td>ROI - Activate new region</td>
</tr>
<tr>
<td>Alt+Z</td>
<td>Toggle lock/unlock zoom level</td>
</tr>
<tr>
<td>Ctrl+# (1, 2, etc.)</td>
<td>Zoom-in by 100%, 200% and so on</td>
</tr>
<tr>
<td>Ctrl+Left Arrow</td>
<td>Move the playhead backwards by 1/2 the distance from the playhead to</td>
</tr>
<tr>
<td></td>
<td>the in point (subsequent key strokes recalculate the distance before</td>
</tr>
<tr>
<td></td>
<td>applying the move)</td>
</tr>
<tr>
<td>Ctrl+LMB</td>
<td>Color picker (single pixel)</td>
</tr>
<tr>
<td>Ctrl+P</td>
<td>Toggle Proxy</td>
</tr>
<tr>
<td>Ctrl+Right Arrow</td>
<td>Move the playhead forwards by 1/2 the distance from the playhead to</td>
</tr>
<tr>
<td></td>
<td>the out point (subsequent key strokes recalculate the distance before</td>
</tr>
<tr>
<td></td>
<td>applying the move)</td>
</tr>
<tr>
<td>Ctrl+RMB</td>
<td>Deselect sampled pixels</td>
</tr>
<tr>
<td>Ctrl+S</td>
<td>Save current script</td>
</tr>
<tr>
<td>Ctrl+U</td>
<td>Toggle previewing output on an external broadcast video monitor</td>
</tr>
<tr>
<td>LMB + MMB and drag</td>
<td>Zoom in and out with the click point set as the center of the Viewer</td>
</tr>
<tr>
<td>MMB and RMB</td>
<td>Toggle between the last specified range (MMB and drag) and <strong>Visible</strong></td>
</tr>
<tr>
<td>MMB and drag (time slider)</td>
<td>Zooms the time slider to the range specified by the drag operation</td>
</tr>
<tr>
<td>MMB and drag (Viewer)</td>
<td>Pans in the Viewer</td>
</tr>
<tr>
<td>Keystroke(s)</td>
<td>Action</td>
</tr>
<tr>
<td>------------------------------</td>
<td>------------------------------------------------------------------------</td>
</tr>
<tr>
<td>MMB scroll</td>
<td>Zoom in and out of the time slider</td>
</tr>
<tr>
<td>Shift+[</td>
<td>Toggle top toolbar</td>
</tr>
<tr>
<td>Shift+]</td>
<td>Toggle bottom toolbar</td>
</tr>
<tr>
<td>Shift+1, 2, etc.</td>
<td>Set Viewer inputs for B buffer</td>
</tr>
<tr>
<td>Shift+Down Arrow</td>
<td>Switch to the previous input for the B buffer</td>
</tr>
<tr>
<td>Shift+Left Arrow</td>
<td>Move the playhead backward by the amount of frames specified in the skip frames control</td>
</tr>
<tr>
<td>Shift+numeric pad left arrow</td>
<td>Nudge the playhead backward by one frame</td>
</tr>
<tr>
<td>Shift+numeric pad right arrow</td>
<td>Nudge the playhead forward by one frame</td>
</tr>
<tr>
<td>Shift+Right Arrow</td>
<td>Move the playhead forward by the amount of frames specified in the skip frames control</td>
</tr>
<tr>
<td>Shift+Up Arrow</td>
<td>Switch to the next input for the B buffer</td>
</tr>
<tr>
<td>Shift+W</td>
<td>Enable or disable ROI</td>
</tr>
<tr>
<td>Alt+Shift+S</td>
<td>Save script and increment version number</td>
</tr>
<tr>
<td></td>
<td>The script name must include _v# for the increment to work as expected. For example, <code>myScript_v01.nk</code></td>
</tr>
<tr>
<td>Ctrl+Shift+LMB</td>
<td>Color picker (region)</td>
</tr>
<tr>
<td>Ctrl+Shift+P</td>
<td>Toggle pixel aspect ratio</td>
</tr>
<tr>
<td>Ctrl+Shift+S</td>
<td>Save current script and specify name (Save As)</td>
</tr>
</tbody>
</table>
# 3D Compositing Viewer

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>3D Top view</td>
</tr>
<tr>
<td>PgDown</td>
<td>Next layer (color channel display)</td>
</tr>
<tr>
<td>PgUp</td>
<td>Previous layer (color channel display)</td>
</tr>
<tr>
<td>S</td>
<td>Display Viewer Settings</td>
</tr>
<tr>
<td>Tab</td>
<td>Toggle 2D/3D viewer</td>
</tr>
<tr>
<td>V</td>
<td>3D Perspective view</td>
</tr>
<tr>
<td>W</td>
<td>Toggle the Wipe tool</td>
</tr>
<tr>
<td>X</td>
<td>3D right-side view</td>
</tr>
<tr>
<td>Z</td>
<td>3D front view</td>
</tr>
<tr>
<td>Alt+LMB</td>
<td>Translate viewer on y,z axis</td>
</tr>
<tr>
<td>Alt+MMB</td>
<td>Zoom in/out (drag left/right)</td>
</tr>
<tr>
<td>Alt+RMB or Ctrl+LMB</td>
<td>Rotate viewer on x,y axis</td>
</tr>
<tr>
<td>Ctrl+L</td>
<td>Toggle Unlocked/Locked/Interactive Camera or Light</td>
</tr>
<tr>
<td>Shift+C</td>
<td>3D Bottom view</td>
</tr>
<tr>
<td>Shift+X</td>
<td>3D Left-side view</td>
</tr>
<tr>
<td>Shift+Z</td>
<td>3D Back view</td>
</tr>
<tr>
<td>Ctrl+Shift+LMB</td>
<td>Rotate viewer on z axis</td>
</tr>
</tbody>
</table>
# Node Graph

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>–</td>
<td>Zoom-out</td>
</tr>
<tr>
<td>. (period)</td>
<td>Insert Dot node</td>
</tr>
<tr>
<td>/</td>
<td>Search by node name or class</td>
</tr>
<tr>
<td>\</td>
<td>Snaps all nodes to grid</td>
</tr>
<tr>
<td>1, 2, 3, etc.</td>
<td>With a node selected in the Node Graph, create/connect Viewer inputs</td>
</tr>
<tr>
<td></td>
<td>With no nodes selected in the Node Graph, cycle through connected views</td>
</tr>
<tr>
<td>+</td>
<td>Zoom-in</td>
</tr>
<tr>
<td>B</td>
<td>Insert Blur node</td>
</tr>
<tr>
<td>Backspace/Delete</td>
<td>Delete selected nodes</td>
</tr>
<tr>
<td>C</td>
<td>Insert ColorCorrect node</td>
</tr>
<tr>
<td>D</td>
<td>Disable/Enable node</td>
</tr>
<tr>
<td>Down Arrow</td>
<td>Next node in tree</td>
</tr>
<tr>
<td>F</td>
<td>Fit selected nodes to Node Graph panel or, if no nodes selected, fits</td>
</tr>
<tr>
<td></td>
<td>all nodes to Node Graph panel</td>
</tr>
<tr>
<td>F5</td>
<td>Render all Write nodes</td>
</tr>
<tr>
<td>F7</td>
<td>Render selected Write nodes</td>
</tr>
<tr>
<td>G</td>
<td>Insert Grade node</td>
</tr>
<tr>
<td>I</td>
<td>Display selected node information</td>
</tr>
<tr>
<td>J</td>
<td>Jump to bookmarked node</td>
</tr>
<tr>
<td>K</td>
<td>Insert Copy node</td>
</tr>
<tr>
<td>L</td>
<td>Auto-place selected nodes</td>
</tr>
<tr>
<td>Keystroke(s)</td>
<td>Action</td>
</tr>
<tr>
<td>------------------------</td>
<td>-------------------------------------------------------------------------</td>
</tr>
<tr>
<td>M</td>
<td>Insert Merge node</td>
</tr>
<tr>
<td>MMB</td>
<td>Fit all nodes to Node Graph panel</td>
</tr>
<tr>
<td>MMB and drag</td>
<td>Pan</td>
</tr>
<tr>
<td>N</td>
<td>Rename selected node</td>
</tr>
<tr>
<td>Numeric pad arrow keys</td>
<td>Navigate around the node tree in the direction specified. For example, pressing the left arrow key (4) selects the next node to the left of the current node in the tree.</td>
</tr>
<tr>
<td>O</td>
<td>Insert Roto node</td>
</tr>
<tr>
<td>P</td>
<td>Insert RotoPaint node</td>
</tr>
<tr>
<td>Q</td>
<td>Show named script info</td>
</tr>
<tr>
<td>R</td>
<td>Insert Read node</td>
</tr>
<tr>
<td>Return</td>
<td>Open properties for selected node(s)</td>
</tr>
<tr>
<td>S</td>
<td>Display Project Settings</td>
</tr>
<tr>
<td>T</td>
<td>Insert Transform node</td>
</tr>
<tr>
<td>Tab</td>
<td>Tab node search menu</td>
</tr>
<tr>
<td>U</td>
<td>Splay first (splay selected nodes to first selected node)</td>
</tr>
<tr>
<td>Up Arrow</td>
<td>Previous node in tree</td>
</tr>
<tr>
<td>W</td>
<td>Insert Write node</td>
</tr>
<tr>
<td>X</td>
<td>Command entry mode</td>
</tr>
<tr>
<td>Y</td>
<td>With two or more nodes selected, splay the inputs of the first node selected to the outputs of subsequent node selections upstream</td>
</tr>
</tbody>
</table>

**Note:** If more than two nodes are selected that don't have multiple inputs, Y ignores all nodes except the first two selections.

<p>| Alt+# (1, 2, etc.)     | Zoom-out %                                                             |</p>
<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alt+B</td>
<td>Duplicate and branch selected nodes</td>
</tr>
<tr>
<td>Alt+C</td>
<td>Duplicate selected node(s)</td>
</tr>
<tr>
<td>Alt+D</td>
<td>Toggles whether a node is always displayed in Dope Sheet or not</td>
</tr>
<tr>
<td>Alt+Down Arrow</td>
<td>Version down the current Read/Write node file name</td>
</tr>
<tr>
<td>Alt+E</td>
<td>Toggle expression links on or off</td>
</tr>
<tr>
<td>Alt+F</td>
<td>Generate Flipbook for node</td>
</tr>
<tr>
<td>Alt+H</td>
<td>Hide node inputs when not selected</td>
</tr>
<tr>
<td>Alt+I</td>
<td>Display script information, such as the node count, channel count,</td>
</tr>
<tr>
<td></td>
<td>cache usage, and whether the script is in full-res or proxy mode</td>
</tr>
<tr>
<td>Alt+K</td>
<td>Clone selected node(s)</td>
</tr>
<tr>
<td>Alt+N</td>
<td>Create StickyNote</td>
</tr>
<tr>
<td>Alt+P</td>
<td>Toggle postage stamp on or off</td>
</tr>
<tr>
<td>Alt+U</td>
<td>With two or more nodes selected, splay first selected node to input A</td>
</tr>
<tr>
<td></td>
<td>of all subsequent selections</td>
</tr>
<tr>
<td>Alt+Up Arrow</td>
<td>Version up the current Read/Write node file name</td>
</tr>
<tr>
<td>Alt+X</td>
<td>Run a script from the file browser</td>
</tr>
<tr>
<td>Ctrl+# (0, 1, 2, etc.)</td>
<td>Zoom-in %</td>
</tr>
<tr>
<td>Ctrl+A</td>
<td>Select all nodes in Node Graph</td>
</tr>
<tr>
<td>Ctrl+B</td>
<td>Toggle node buffer for selected nodes</td>
</tr>
</tbody>
</table>

**Tip:** When enabled, the output from the nodes is cached so that it can be re-read quickly. A yellow line displays under the nodes to indicate that caching is enabled.

<p>| Ctrl+C       | Copy selected node(s)                                                 |</p>
<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ctrl+create node</td>
<td>Replace selected node with new node, such as Ctrl+B to replace the target with a Blur node</td>
</tr>
<tr>
<td>Ctrl+D</td>
<td>Disconnect upstream node</td>
</tr>
<tr>
<td>Ctrl+Down Arrow</td>
<td>Move selected node downstream</td>
</tr>
<tr>
<td>Ctrl+F7..F10</td>
<td>Save locations 1 through 4</td>
</tr>
<tr>
<td>Ctrl+G</td>
<td>Nest selected nodes in Group</td>
</tr>
<tr>
<td>Ctrl+I</td>
<td>Open new Compositing Viewer</td>
</tr>
<tr>
<td>Ctrl+K</td>
<td>Copy as clone</td>
</tr>
<tr>
<td>Ctrl+L</td>
<td>Collapse selected nodes to a LiveGroup or, if no nodes are selected, create an empty LiveGroup</td>
</tr>
<tr>
<td>Ctrl+LMB on node</td>
<td>Highlight all upstream nodes</td>
</tr>
<tr>
<td>Ctrl+P</td>
<td>Toggle Proxy</td>
</tr>
<tr>
<td>Ctrl+Return/Enter</td>
<td>Open a Group's sub-graph</td>
</tr>
<tr>
<td>Ctrl+Up Arrow</td>
<td>Move selected node upstream</td>
</tr>
<tr>
<td>Ctrl+V</td>
<td>Paste node(s) from the clipboard</td>
</tr>
<tr>
<td>Ctrl+X</td>
<td>Cut selected node(s)</td>
</tr>
<tr>
<td>LMB + MMB and drag</td>
<td>Zoom in and out with the click point set as the center of the Node Graph</td>
</tr>
<tr>
<td>Shift+\</td>
<td>Snap selected node to grid</td>
</tr>
<tr>
<td>Shift+# (1, 2, 3, etc.)</td>
<td>Connect selected node to Viewer B buffer</td>
</tr>
<tr>
<td>Shift+A</td>
<td>Insert AddMix node</td>
</tr>
<tr>
<td>Shift+create node</td>
<td>Create node in new branch, such as Shift+B to create a Blur node in a new branch</td>
</tr>
<tr>
<td>Keystroke(s)</td>
<td>Action</td>
</tr>
<tr>
<td>-------------------</td>
<td>-------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Shift+drag</td>
<td>Duplicate selected arrow</td>
</tr>
<tr>
<td>Shift+F7..F10</td>
<td>Restore locations 1 through 4</td>
</tr>
<tr>
<td>Shift+U</td>
<td>With two or more nodes selected, splay first selected node to input B of all subsequent selections</td>
</tr>
<tr>
<td>Shift+X</td>
<td>Swap A/B inputs on node</td>
</tr>
<tr>
<td>Shift+Y</td>
<td>With two or more nodes selected, splay the outputs of the selected nodes to the inputs of the last node selected downstream</td>
</tr>
</tbody>
</table>

**Note:** If more than two nodes are selected that don't have multiple inputs, **Shift+Y** ignores all nodes except the last two selections.

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alt+LMB drag</td>
<td>Pan</td>
</tr>
<tr>
<td>Alt+MMB drag</td>
<td>Zoom-in/out</td>
</tr>
<tr>
<td>Alt+Shift+K</td>
<td>Declone nodes</td>
</tr>
<tr>
<td>Alt+Shift+U</td>
<td>With two or more nodes selected, splay last selected node to input A of all previous selections</td>
</tr>
<tr>
<td>Alt+Shift+Up Arrow</td>
<td>Version to up to the latest file (Read nodes only)</td>
</tr>
<tr>
<td>Ctrl+Alt+A on node</td>
<td>Select all nodes connected to the target node's tree</td>
</tr>
<tr>
<td>Ctrl+Alt+G</td>
<td>Replace Group node with nested nodes</td>
</tr>
<tr>
<td>Ctrl+Alt+LMB on node</td>
<td>Open node properties in floating window</td>
</tr>
<tr>
<td>Ctrl+Alt+V</td>
<td>Paste knob values to a node of the same class as the copy operation</td>
</tr>
<tr>
<td>Ctrl+Shift+/</td>
<td>Search and Replace (Read and Write nodes)</td>
</tr>
<tr>
<td>Ctrl+Shift+B</td>
<td>Toggle selected node(s) bookmark on or off</td>
</tr>
</tbody>
</table>
## Appendix B: Keyboard Shortcuts | Project/Tags/Version Bin

### Keystroke(s) | Action
--- | ---
Ctrl+Shift+C | Change node color
Ctrl+Shift+G | Copy gizmo to group
Ctrl+Shift+V | Paste a copied node into a new branch from an existing node
Ctrl+Shift+LMB on node | Select all upstream nodes
Ctrl+Shift+N | Create a new script in a new session
Ctrl+Shift+P | Create Precomp from selected nodes
Ctrl+Shift+X | Extract selected nodes from tree
Ctrl+Alt+Shift+G | Create Group from selected nodes
Ctrl+Alt+Shift+K | Force clone

#### Project/Tags/Version Bin

### Keystroke(s) | Action
--- | ---
D | Hide version(s) of a clip. You can only hide versions when a clip is opened in the Versions Bin
F8 | Refresh clips
V | Display the version selector
Alt+D | Display metadata for the selected shot(s)
Alt+Down Arrow | Version down the selected clip
Alt+F5 | Rescan clip range
Alt+Up Arrow | Version up the selected clip
Ctrl+B | Create a new bin
Ctrl+N | Create a new sequence
### User Guide

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ctrl+Return</td>
<td>Open selected clip in the Viewer</td>
</tr>
<tr>
<td>Ctrl+Y</td>
<td>Create a new tag. You can only add tags to the Tags panel</td>
</tr>
<tr>
<td>Ctrl+Alt+Down Arrow</td>
<td>Go to min version</td>
</tr>
<tr>
<td>Ctrl+Alt+Return</td>
<td>Open selected clip a new Viewer</td>
</tr>
<tr>
<td>Ctrl+Alt+Up Arrow</td>
<td>Go to max version</td>
</tr>
</tbody>
</table>

### Properties Panel

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>\</td>
<td>Snaps all nodes to grid</td>
</tr>
<tr>
<td>/</td>
<td>Search by node name or class</td>
</tr>
<tr>
<td>. (period)</td>
<td>Create Dot node</td>
</tr>
<tr>
<td>1, 2, 3, etc.</td>
<td>With a node selected in the Node Graph, create/connect Viewer inputs</td>
</tr>
<tr>
<td>B</td>
<td>Insert Blur node</td>
</tr>
<tr>
<td>Backspace/Delete</td>
<td>Delete selected nodes</td>
</tr>
<tr>
<td>C</td>
<td>Insert ColorCorrect node</td>
</tr>
<tr>
<td>D</td>
<td>Disable/Enable node</td>
</tr>
<tr>
<td>Esc</td>
<td>Closes currently active or last selected Properties panel</td>
</tr>
<tr>
<td>F5</td>
<td>Render all Write nodes</td>
</tr>
<tr>
<td>F7</td>
<td>Render selected Write nodes</td>
</tr>
<tr>
<td>G</td>
<td>Insert Grade node</td>
</tr>
</tbody>
</table>
### Appendix B: Keyboard Shortcuts | Properties Panel

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>Display selected node information</td>
</tr>
<tr>
<td>J</td>
<td>Jump to bookmarked node</td>
</tr>
<tr>
<td>K</td>
<td>Insert Copy node</td>
</tr>
<tr>
<td>L</td>
<td>Auto-place selected nodes</td>
</tr>
<tr>
<td>M</td>
<td>Insert Merge node</td>
</tr>
<tr>
<td>O</td>
<td>Insert Roto node</td>
</tr>
<tr>
<td>P</td>
<td>Insert RotoPaint node</td>
</tr>
<tr>
<td>Q</td>
<td>Show named script info</td>
</tr>
<tr>
<td>R</td>
<td>Insert Read node</td>
</tr>
<tr>
<td>Return/Enter</td>
<td>Chooses selected menu item</td>
</tr>
<tr>
<td>S</td>
<td>Display Project Settings</td>
</tr>
<tr>
<td>T</td>
<td>Insert Transform node</td>
</tr>
<tr>
<td>Tab</td>
<td>Move to next control in the Properties panel</td>
</tr>
<tr>
<td>Up/Down Arrow</td>
<td>Increment control values</td>
</tr>
<tr>
<td>U</td>
<td>Splay first (splay selected nodes to first selected node)</td>
</tr>
<tr>
<td>W</td>
<td>Insert Write node</td>
</tr>
<tr>
<td>X</td>
<td>Command entry mode</td>
</tr>
<tr>
<td>Y</td>
<td>With two or more nodes selected, splay the inputs of the first node selected to the outputs of subsequent node selections upstream</td>
</tr>
</tbody>
</table>

**Note:** If more than two nodes are selected that don't have multiple inputs, Y ignores all nodes except the first two selections.

<p>| LMB and drag | Copy current value from one control to another |</p>
<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>MMB and drag</td>
<td>Adjust control values using virtual slider (regular)</td>
</tr>
<tr>
<td>Alt+B</td>
<td>Duplicate and branch selected nodes</td>
</tr>
<tr>
<td>Alt+C</td>
<td>Duplicate selected node(s)</td>
</tr>
<tr>
<td>Alt+D</td>
<td>Toggles whether a node is always displayed in Dope Sheet or not</td>
</tr>
<tr>
<td>Alt+E</td>
<td>Toggle expression links on or off</td>
</tr>
<tr>
<td>Alt+F</td>
<td>Generate Flipbook for node</td>
</tr>
<tr>
<td>Alt+H</td>
<td>Hide node inputs when not selected</td>
</tr>
<tr>
<td>Alt+K</td>
<td>Clone selected node(s)</td>
</tr>
<tr>
<td>Alt+LMB on close (x)</td>
<td>Close all open Properties panels</td>
</tr>
<tr>
<td>Alt+N</td>
<td>Create StickyNote</td>
</tr>
<tr>
<td>Alt+P</td>
<td>Toggle postage stamp on and off for selected Properties panel</td>
</tr>
<tr>
<td>Alt+U</td>
<td>With two or more nodes selected, splay first selected node to input A of all subsequent selections</td>
</tr>
<tr>
<td>Alt+X</td>
<td>Run a script from the file browser</td>
</tr>
<tr>
<td>Ctrl+A</td>
<td>Select all nodes in Properties panel</td>
</tr>
<tr>
<td>Ctrl+B</td>
<td>Toggle node buffer for selected nodes</td>
</tr>
<tr>
<td>Ctrl+C</td>
<td>Copy selected node(s)</td>
</tr>
<tr>
<td>Ctrl+D</td>
<td>Disconnect upstream node</td>
</tr>
<tr>
<td>Ctrl+F7..F10</td>
<td>Save locations 1 through 4</td>
</tr>
<tr>
<td>Ctrl+G</td>
<td>Nest selected nodes in Group</td>
</tr>
</tbody>
</table>

**Tip:** When enabled, the output from the nodes is cached so that it can be re-read quickly. A yellow line displays under the nodes to indicate that caching is enabled.
<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ctrl+I</td>
<td>Open new Compositing Viewer</td>
</tr>
<tr>
<td>Ctrl+K</td>
<td>Copy as clone</td>
</tr>
<tr>
<td>Ctrl+L</td>
<td>Collapse selected nodes to a LiveGroup or, if no nodes are selected, create an empty LiveGroup</td>
</tr>
<tr>
<td>Ctrl+LMB</td>
<td>Reset slider to default</td>
</tr>
<tr>
<td>Ctrl+LMB on close (x)</td>
<td>Close all properties panels except the one clicked</td>
</tr>
<tr>
<td>Ctrl+P</td>
<td>Toggle Proxy</td>
</tr>
<tr>
<td>Ctrl+V</td>
<td>Paste node(s) from the clipboard</td>
</tr>
<tr>
<td>Ctrl+X</td>
<td>Cut selected node(s)</td>
</tr>
<tr>
<td>Shift+A</td>
<td>Insert AddMix node</td>
</tr>
<tr>
<td>Shift+# (1, 2, 3, etc.)</td>
<td>Connect selected node to Viewer B buffer</td>
</tr>
<tr>
<td>Shift+create node</td>
<td>Create node in new branch, such as Shift+B to create a Blur node in a new branch</td>
</tr>
<tr>
<td>Shift+F7..F10</td>
<td>Restore locations 1 through 4</td>
</tr>
<tr>
<td>Shift+U</td>
<td>With two or more nodes selected, splay first selected node to input B of all subsequent selections</td>
</tr>
<tr>
<td>Shift+X</td>
<td>Swap A/B inputs on node</td>
</tr>
<tr>
<td>Shift+Y</td>
<td>With two or more nodes selected, splay the outputs of the selected nodes to the inputs of the last node selected downstream</td>
</tr>
</tbody>
</table>

**Note:** If more than two nodes are selected that don't have multiple inputs, Shift+Y ignores all nodes except the last two selections.

Alt+LMB drag       | Adjust control values using virtual slider (magnitude dependent on cursor position) |
<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alt+MMB and drag</td>
<td>Adjust control values using virtual slider (fine)</td>
</tr>
<tr>
<td>Ctrl+LMB drag</td>
<td>Expression link controls</td>
</tr>
<tr>
<td>Shift+LMB drag</td>
<td>Copy animation from one control to another</td>
</tr>
<tr>
<td>Shift+MMB and drag</td>
<td>Adjust control values using virtual slider (coarse)</td>
</tr>
<tr>
<td>Shift+Tab</td>
<td>Move to previous control in the properties</td>
</tr>
<tr>
<td>Alt+Shift+K</td>
<td>Declone nodes</td>
</tr>
<tr>
<td>Alt+Shift+U</td>
<td>With two or more nodes selected, splay last selected node to input A of all previous selections</td>
</tr>
<tr>
<td>Ctrl+Alt+A (on a node)</td>
<td>Select all nodes connected to the target node’s tree</td>
</tr>
<tr>
<td>Ctrl+Alt+G</td>
<td>Replace Group node with nested nodes</td>
</tr>
<tr>
<td>Ctrl+Alt+V</td>
<td>Paste knob values to a node of the same class as the copy operation</td>
</tr>
<tr>
<td>Ctrl+Shift+/</td>
<td>Search and Replace strings in selected Read and Write nodes</td>
</tr>
<tr>
<td>Ctrl+Shift+A</td>
<td>Close all open Properties panels</td>
</tr>
<tr>
<td>Ctrl+Shift+B</td>
<td>Toggle selected node(s) bookmark on or off</td>
</tr>
<tr>
<td>Ctrl+Shift+C</td>
<td>Change node color</td>
</tr>
<tr>
<td>Ctrl+Shift+G</td>
<td>Copy gizmo to group</td>
</tr>
<tr>
<td>Ctrl+Shift+P</td>
<td>Create Precomp from selected nodes</td>
</tr>
<tr>
<td>Ctrl+Shift+V</td>
<td>Paste a copied node into a new branch from an existing node</td>
</tr>
<tr>
<td>Ctrl+Shift+X</td>
<td>Extract selected nodes from tree</td>
</tr>
<tr>
<td>Ctrl+Alt+Shift+K</td>
<td>Force clone</td>
</tr>
</tbody>
</table>
## Curve Editor/Dope Sheet

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Frame all keyframes</td>
</tr>
<tr>
<td>C</td>
<td>Interpolation (Cubic)</td>
</tr>
<tr>
<td>F</td>
<td>Frame all selected keyframes</td>
</tr>
<tr>
<td>H</td>
<td>Interpolation (Horizontal)</td>
</tr>
<tr>
<td>K</td>
<td>Interpolation (Constant)</td>
</tr>
<tr>
<td>L</td>
<td>Interpolation (Linear)</td>
</tr>
<tr>
<td>LMB</td>
<td>Select single point</td>
</tr>
<tr>
<td>LMB drag on blank space</td>
<td>Select region of points</td>
</tr>
<tr>
<td>LMB drag on point</td>
<td>Move all selected points</td>
</tr>
<tr>
<td>LMB drag on selection box</td>
<td>Scale points inside selection region</td>
</tr>
<tr>
<td>LMB drag on transform handle</td>
<td>Move all points in selection region</td>
</tr>
<tr>
<td>MMB drag</td>
<td>Draw box in area and zoom to fit area to curve editor panel</td>
</tr>
<tr>
<td>MMB or F</td>
<td>Fit selection to window</td>
</tr>
<tr>
<td>R</td>
<td>Interpolation (Catmull-Rom)</td>
</tr>
<tr>
<td>X</td>
<td>Break selected control points' handles</td>
</tr>
<tr>
<td>Z</td>
<td>Interpolation (Smooth [bezier])</td>
</tr>
<tr>
<td>Alt+` (backtick)</td>
<td>Display Curve Editor</td>
</tr>
<tr>
<td>Alt+LMB drag</td>
<td>Pan</td>
</tr>
<tr>
<td>Alt+MMB drag</td>
<td>Variable zoom</td>
</tr>
</tbody>
</table>
### Appendix B: Keyboard Shortcuts

#### Script Editor

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tab</td>
<td>Increase indentation</td>
</tr>
<tr>
<td>Ctrl+[</td>
<td>Load previous script</td>
</tr>
<tr>
<td>Ctrl+]</td>
<td>Load next script</td>
</tr>
<tr>
<td>Ctrl+Backspace</td>
<td>Clear output window</td>
</tr>
</tbody>
</table>

#### Keyboard Shortcuts

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ctrl+A</td>
<td>Select all curves</td>
</tr>
<tr>
<td>Ctrl+C</td>
<td>Copy selected keys</td>
</tr>
<tr>
<td>Ctrl+E</td>
<td>Copy Expressions</td>
</tr>
<tr>
<td>Ctrl+L</td>
<td>Copy Links</td>
</tr>
<tr>
<td>Ctrl+LMB drag</td>
<td>Move keyframes freely on the x and y axes</td>
</tr>
<tr>
<td>Ctrl+Shift (hold down)</td>
<td>Hide points to click on selection box/transform handle</td>
</tr>
<tr>
<td>Ctrl+V</td>
<td>Paste curve</td>
</tr>
<tr>
<td>Ctrl+X</td>
<td>Cut selected keys</td>
</tr>
<tr>
<td>Shift+LMB</td>
<td>Add or Remove points from selection</td>
</tr>
<tr>
<td>Shift+LMB drag</td>
<td>Draw box to add/remove points from selection</td>
</tr>
<tr>
<td>Alt+Shift+LMB drag</td>
<td>Move single point</td>
</tr>
<tr>
<td>Ctrl+Alt+LMB</td>
<td>Add point to current curve</td>
</tr>
<tr>
<td>Ctrl+Shift+C</td>
<td>Copy selected curves</td>
</tr>
<tr>
<td>Ctrl+Alt+Shift+LMB</td>
<td>Sketch points freely on current curve</td>
</tr>
</tbody>
</table>

---

**Script Editor**

**Keystroke(s)** | **Action**                          
--- | ---
Tab | Increase indentation 
Ctrl+[ | Load previous script 
Ctrl+] | Load next script 
Ctrl+Backspace | Clear output window
<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ctrl+Return/Enter</td>
<td>Run script in editor</td>
</tr>
<tr>
<td>Shift+Tab</td>
<td>Decrease indentation</td>
</tr>
<tr>
<td>Ctrl+Shift+[</td>
<td>Decrease indentation of selected text</td>
</tr>
<tr>
<td>Ctrl+Shift+]</td>
<td>Increase indentation of selected text</td>
</tr>
</tbody>
</table>

## Roto/RotoPaint

<table>
<thead>
<tr>
<th>Keystroke(s)</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Backspace</td>
<td>Delete an item from curve list or Delete points/shapes</td>
</tr>
<tr>
<td>C</td>
<td>Toggle Clone tool</td>
</tr>
<tr>
<td>D</td>
<td>Toggle Dodge/Burn</td>
</tr>
<tr>
<td>Delete</td>
<td>Remove selected point(s)</td>
</tr>
<tr>
<td>E</td>
<td>Increase feather on selected point(s)</td>
</tr>
<tr>
<td>Esc</td>
<td>Switch back to the current Select tool</td>
</tr>
<tr>
<td>I</td>
<td>Pick color</td>
</tr>
<tr>
<td>N</td>
<td>Toggle Brush/Eraser</td>
</tr>
<tr>
<td>Return (Bezier/Spline tools)</td>
<td>Close shape</td>
</tr>
<tr>
<td>S (with Viewer mouse-over focus)</td>
<td>Cycle between the selected tool's modes</td>
</tr>
<tr>
<td>T (Clone tool)</td>
<td>Toggle source as onion skin with transform jack</td>
</tr>
<tr>
<td>T (Select tool)</td>
<td>Display transform box (points) or jack (shapes)</td>
</tr>
<tr>
<td>V</td>
<td>Toggle Bezier/B-Spline/Ellipse/Rectangle tools</td>
</tr>
<tr>
<td>X</td>
<td>Toggle Blur/Sharpen/Smear tools</td>
</tr>
<tr>
<td>Keystroke(s)</td>
<td>Action</td>
</tr>
<tr>
<td>------------------------------------------</td>
<td>---------------------------------------------</td>
</tr>
<tr>
<td>Z</td>
<td>Smooth selected points</td>
</tr>
<tr>
<td>Ctrl+A</td>
<td>Select all points</td>
</tr>
<tr>
<td>Ctrl+LMB (Bezier/Spline tool)</td>
<td>Sketch a Bezier or B-spline</td>
</tr>
<tr>
<td>Ctrl+LMB (on a point)</td>
<td>Break tangent handle for selected point</td>
</tr>
<tr>
<td>Ctrl+drag (Clone/Reveal tools)</td>
<td>Set offset between source and destination</td>
</tr>
<tr>
<td>Ctrl+Shift (transform box)</td>
<td>Drag transform box points to move them</td>
</tr>
<tr>
<td>Shift+LMB (Bezier tool)</td>
<td>Create a sharp point on previous point</td>
</tr>
<tr>
<td>Shift+LMB (editing points)</td>
<td>Bring up transform box for selected points</td>
</tr>
<tr>
<td>Shift+drag (Brush/Eraser/Clone/Reveal tools)</td>
<td>Change brush size</td>
</tr>
<tr>
<td>Shift+drag (editing Bezier/Spline points)</td>
<td>Move both tangent handles at the same time</td>
</tr>
<tr>
<td>Shift+E</td>
<td>Remove feather from selected points</td>
</tr>
<tr>
<td>Shift+Z</td>
<td>Cusp selected points</td>
</tr>
<tr>
<td>Ctrl+Alt+LMB (on Splines)</td>
<td>Add point to curve</td>
</tr>
<tr>
<td>Ctrl+Shift+drag (B-Spline)</td>
<td>Increase/decrease tension of B-Spline shape</td>
</tr>
</tbody>
</table>
Appendix C: Supported File Formats

Notes

When importing and exporting files, remember the following:

• When you import images with a Read node (Image > Read), Nuke analyzes the contents of the file to determine the format. The file name extension is not used to determine file format, which allows flexibility with naming conventions in a production environment.

• Regardless of format, Nuke converts all imported image sequences to its native 32-bit linear RGB colorspace.

• When you render new images from Nuke (Image > Write), you can use a file name extension to specify format.

• To import and export geometry objects from Alembic, FBX, or OBJ files, use the ReadGeo node (3D > Geometry > ReadGeo). To write them out again, use the WriteGeo node (3D > Geometry > WriteGeo). To import cameras or transforms from Alembic or FBX files, use the Camera node (3D > Camera). To import lights from an FBX file, use the Light node (3D > Lights > Light).

• To import deep images (either in DTEX or scanline OpenEXR format), use a DeepRead node (Deep > DeepRead). To export deep images (in scanline OpenEXR format), use a DeepWrite node (Deep > DeepWrite).

Supported File Formats

The following table lists the supported file formats. The extensions listed under Extension let you specify the image format; use these as the actual file name extensions or the prefix to indicate output format for the image sequences.

<table>
<thead>
<tr>
<th>Format</th>
<th>Bit Depths</th>
<th>Read/Write</th>
<th>Extension</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alembic</td>
<td>n/a</td>
<td>read and</td>
<td>abc</td>
<td>You can read meshes, point clouds, cameras,</td>
</tr>
</tbody>
</table>
## Format: Bit Depths | Read/Write | Extension | Notes
---|---|---|---
write | | | and transforms from Alembic files into a Nuke scene using the ReadGeo, Camera, and Axis nodes.

To write meshes and point clouds out again, use the WriteGeo node.

<table>
<thead>
<tr>
<th>Format</th>
<th>Bit Depths</th>
<th>Read/Write</th>
<th>Extension</th>
<th>Notes</th>
</tr>
</thead>
</table>
| Apple ProRes | 8, 10 | read and write | mov | Adds support for Apple ProRes 4444 and Apple ProRes 422 on Mac, Linux, and Windows using the mov64 reader.

Apple ProRes 4444 includes the SD, HD, 2K, UHD, and XQ formats. Apple ProRes 422 includes the HQ, LT, and Proxy formats.

<table>
<thead>
<tr>
<th>Format</th>
<th>Bit Depths</th>
<th>Read/Write</th>
<th>Extension</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>ARRI RAW</td>
<td>12</td>
<td>read only</td>
<td>ari</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Format</th>
<th>Bit Depths</th>
<th>Read/Write</th>
<th>Extension</th>
<th>Notes</th>
</tr>
</thead>
</table>
| AVI | n/a | read and write | avi | AVI files can be supported by default or via Nuke's reader/writer that is based on the FFmpeg open source library. If you get an error when using AVI files in Read nodes, you may need to use the prefix **mov64**: before the file path and file name, for example: `mov64:z:/job/FILM/IMG/final_comp_v01.####.avi`

When working with Write nodes, you can also select **mov64** from the **file type** dropdown menu and use avi as the file extension.

On Windows, in order to support more codecs, the AVI reader uses the DirectShow multimedia architecture. When decoding `.avi` files, DirectShow tries to find the appropriate codec on the system. If the codec is not available, DirectShow and Nuke are unable to open the `.avi` file. Note that the 64-bit version of Nuke can only use 64-bit DirectShow...
<table>
<thead>
<tr>
<th>Format</th>
<th>Bit Depths</th>
<th>Read/Write</th>
<th>Extension</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>CIN</td>
<td>10 (log)</td>
<td>read and write</td>
<td>cin</td>
<td>Codecs. If you only have a 32-bit codec installed, the 64-bit version of Nuke cannot use it to open .avi files.</td>
</tr>
<tr>
<td>DNG</td>
<td>8, 12</td>
<td>read</td>
<td>dng</td>
<td>Includes RAW 2.5K CinemaDNG</td>
</tr>
<tr>
<td>DPX</td>
<td>8, 10, 12, and 16</td>
<td>read and write</td>
<td>dpx</td>
<td>YCbCr encoded DPX files are not supported on the timeline.</td>
</tr>
<tr>
<td>DTEX</td>
<td>32</td>
<td>read only</td>
<td>dtex</td>
<td>To use DTEX files, you need Pixar's PhotoRealistic RenderMan® Pro Server 20, or earlier, on your computer.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>To read a DTEX file, use the DeepRead node.</td>
</tr>
<tr>
<td>FBX</td>
<td></td>
<td>read and write</td>
<td>fbx</td>
<td>You can read meshes, point clouds, cameras, lights, and transforms from FBX files into a Nuke scene using the ReadGeo, Camera, Light, and Axis nodes.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>To write geometry out again, use the WriteGeo node.</td>
</tr>
<tr>
<td>GIF</td>
<td>8</td>
<td>read only</td>
<td>gif</td>
<td></td>
</tr>
<tr>
<td>Radiance</td>
<td>16</td>
<td>read and write</td>
<td>hdr, hdri</td>
<td>This format stores an 8-bit mantissa for each of r, g, and b and an additional 8-bit exponent that is shared by all three, which packs the floating point RGB triplet into 32 bits per pixel.</td>
</tr>
<tr>
<td>JPEG</td>
<td>8</td>
<td>read and write</td>
<td>jpg, jpeg</td>
<td>Adjust compression levels using the quality slider in the Write node's properties panel.</td>
</tr>
<tr>
<td>Maya IFF</td>
<td>8, 16</td>
<td>read only</td>
<td>iff</td>
<td></td>
</tr>
<tr>
<td>Format</td>
<td>Bit Depths</td>
<td>Read/Write</td>
<td>Extension</td>
<td>Notes</td>
</tr>
<tr>
<td>-----------</td>
<td>------------</td>
<td>------------</td>
<td>-----------</td>
<td>-------</td>
</tr>
<tr>
<td>MXF</td>
<td>n/a</td>
<td>read only</td>
<td>mxf</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Note:</strong> Currently, only 'complete' MXF files are supported.</td>
</tr>
<tr>
<td>Supported codecs include:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>• Uncompressed 4:2:2 YCbCr 8-/10-bit</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>• Uncompressed 4:4:4:4 RGBA 8-/10-bit</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>• Uncompressed Avid 4:2:2 YCbCr 8-/10-bit</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>• Uncompressed Avid 4:4:4:4 RGBA 8-/10-bit</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>• JPEG2000</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>• Avid DNxHD (1080p and 720p 1920x1080 and 1280x720, 4:4:4:4 and 4:2:2) 36, 115, 120, 145, 175, 185, 220, 220x</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>• Sony Raw from the F65, F55, F5 and FS700 cameras. All formats that these cameras provide: 4K, 2K, 1K, 0.5K and 0.25K</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>• Sony X-OCN from the VENICE, F55, and F5 cameras.</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>• ARRIRAW from the Alexa Mini.</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>OBJ</td>
<td></td>
<td>read and write</td>
<td>obj</td>
<td></td>
</tr>
<tr>
<td>OpenEXR and OpenEXR 2.2</td>
<td>16, 32</td>
<td>read and write</td>
<td>exr</td>
<td>OpenEXR handles 16- and 32-bit float. This 16 is also called &quot;half float&quot; and is different from the 16-bit integer that all the other formats that support 16-bit use.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Nuke supports multi-part OpenEXR files. For more information, see <strong>Notes on Importing OpenEXR Files</strong> and <strong>Notes on Rendering OpenEXR Files</strong>.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>When working with deep data, Nuke supports scanline OpenEXR files. For more information, see <strong>Importing Scanline OpenEXR Files</strong> and</td>
</tr>
<tr>
<td>Format</td>
<td>Bit Depths</td>
<td>Read/Write</td>
<td>Extension</td>
<td>Notes</td>
</tr>
<tr>
<td>--------</td>
<td>------------</td>
<td>------------</td>
<td>------------</td>
<td>-------</td>
</tr>
<tr>
<td>EXR</td>
<td></td>
<td></td>
<td></td>
<td>Writing Deep Data.</td>
</tr>
</tbody>
</table>

**EXR Compression**

EXR file metadata contains a **compression** key/value pair detailing the compression type used to write the `.exr` file. The value is expressed as the name of the compression type or an integer referencing the compression used:

- 0 - no compression
- 1 - RLE compression, run length encoding
- 2 - Zip compression, one scan line at a time
- 3 - Zip compression, in blocks of 16 scan lines
- 4 - PIZ-based wavelet compression, in blocks of 32 scan lines
- 5 - PXR24 compression, lossy 24-bit float
- 6 - B44 compression, lossy 4-by-4 pixel block, fixed rate
- 7 - B44A compression, lossy 4-by-4 pixel block, flat fields are compressed more
- 8 - DWAA compression, lossy DCT based compression, in blocks of 32 scan lines
- 9 - DWAB compression, lossy DCT based compression, in blocks of 256 scan lines

<table>
<thead>
<tr>
<th>Format</th>
<th>Bit Depths</th>
<th>Read/Write</th>
<th>Extension</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>PNG</td>
<td>8, 16</td>
<td>read and write</td>
<td>png (8-bit)</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>png16 (16-bit)</td>
<td></td>
</tr>
<tr>
<td>PSD</td>
<td>8, 16</td>
<td>read only</td>
<td>psd</td>
<td>While Nuke reads standard Photoshop® blend modes, it doesn't read Photoshop layer comps or recognize group blend modes. Photoshop layers are read into separate Nuke layers and anything that doesn't map into that is ignored.</td>
</tr>
<tr>
<td>QuickTime</td>
<td>n/a</td>
<td>read and</td>
<td>mov</td>
<td>QuickTime is only supported by default on</td>
</tr>
<tr>
<td>Format</td>
<td>Bit Depths</td>
<td>Read/Write</td>
<td>Extension</td>
<td>Notes</td>
</tr>
<tr>
<td>----------</td>
<td>------------</td>
<td>------------</td>
<td>-----------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td></td>
<td></td>
<td>write</td>
<td></td>
<td>Windows and Mac. To use QuickTime files on Linux, you need to use the prefix <strong>mov64</strong>: before the file path and file name, for example, mov64:/z:/job/FILM/IMG/final_comp_v01.###.mov</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>The <strong>mov64</strong> writer supports the following codecs:</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>• Apple ProRes 4444</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>• Apple ProRes 4444 XQ</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>• Apple ProRes 422 HQ</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>• Apple ProRes 422</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>• Apple ProRes 422 LT</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>• Apple ProRes 422 Proxy</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>• Avid DNxHD (1080p and 720p 1920x1080 and 1280x720, 4:4:4:4 and 4:2:2) 36, 115, 120, 145, 175, 185, 220, 220x</td>
</tr>
</tbody>
</table>

**Note:** Interlaced writing is not supported. See Avid DNxHD Notes below.

• Photo - JPEG
• MPEG-1 Video
• MPEG-4 Video
• PNG
• Animation
• Uncompressed 10-bit 4:2:2

Avid DNxHD Notes

The bit rates listed in the **codec profile** dropdown are the bit rates for 1080p at 29.97 fps EXCEPT for 36 (which is actually 45 Mbps @ 29.97fps). You should look at the codec format (422/444, 8/10-bit).
### Format | Bit Depths | Read/Write | Extension | Notes
---|---|---|---|---
- | - | - | - | -

**Note:** Nuke only supports 1080p and 720p. Non-HD resolutions are scaled to 1080p before writing.

This leads to a set of 1080p bit rates:
- 1080p/29.97 440x, 220x, 220, 145, 45
- 1080p/60 N/A, N/A, 440, 290, 90 (same at 59.94)
- 1080p/50 N/A, N/A, 367, 242, 75
- 1080p/25 365x, 185x, 185, 120, 36
- 1080p/24 350x, 175x, 175, 115, 36 (same at 23.976)

At 720p, the **codec profile** dropdown has a different interpretation. The bit rate is taken as the bit rate at 720p at 59.94fps. This leads to another set of bit rates:
- 720p/59.94 N/A, 220x, 220, 145, N/A
- 720p/50 N/A, 175x, 175, 115, N/A
- 720p/29.97 N/A, 110x, 110, 75, N/A
- 720p/25 N/A, 90x, 90, 60, N/A
- 720p/23.976 N/A, 90x, 90, 60, N/A

**Note:** Since the bit rates are for 1080p at 29.97 fps AND 720p at 59.94 fps (except for 36 Mbit which should read 45 Mbit). It is possible to calculate the bandwidth for all the other frame rates by:

- BandWidth@1080p = fps/29.97 * NominalBandWidth, or
- BandWidth@720p = fps/59094 * NominalBandWidth

where NominalBandWidth is the bandwidth listed in the codec profile knob OR 45 if the bandwidth listed is 36 Mbit. (Avid labels the codec profile names by the approximate bandwidth.)

<p>| RAW | n/a | read only | n/a | DSLR raw data files, such as Canon .CR2 files. These are only supported via the dcraw |</p>
<table>
<thead>
<tr>
<th>Format</th>
<th>Bit Depths</th>
<th>Read/Write</th>
<th>Extension</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>REDCODE</td>
<td>16</td>
<td>read only</td>
<td>r3d</td>
<td>Note that .r3d files may look different in Nuke compared to various versions of RED applications, like RED ALERT or REDCINE. Unlike most other file formats Nuke reads, the .r3d REDCODE files must be processed to convert from a raw format to an RGB color image. From time to time, a new version of the RED SDK that Nuke uses improves this processing and due to the timing of release cycles, Nuke may sometimes be using a different version than the RED applications.</td>
</tr>
<tr>
<td>SGI</td>
<td>8, 16</td>
<td>read and write</td>
<td>sgi, rgb, rgba (8-bit sequences) sgi16 (for 16-bit sequences)</td>
<td></td>
</tr>
<tr>
<td>SoftImage® PIC</td>
<td>8</td>
<td>read and write</td>
<td>pic</td>
<td></td>
</tr>
<tr>
<td>TIFF</td>
<td>8, 16, and 32</td>
<td>read and write</td>
<td>tif, tiff (8-bit sequences) tif16, tiff16 (16-bit sequences) ftif, ftiff</td>
<td>If utilized, the compression schema on imported TIFF sequences must be LZW®.</td>
</tr>
<tr>
<td>Format</td>
<td>Bit Depths</td>
<td>Read/Write</td>
<td>Extension</td>
<td>Notes</td>
</tr>
<tr>
<td>------------------</td>
<td>------------</td>
<td>------------</td>
<td>-----------</td>
<td>------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Truevision® TARGA</td>
<td>8</td>
<td>read and write</td>
<td>tga, targa</td>
<td></td>
</tr>
<tr>
<td>Wavefront® RLA</td>
<td>8</td>
<td>read only</td>
<td>rla</td>
<td></td>
</tr>
<tr>
<td>XPM</td>
<td>8</td>
<td>read and write</td>
<td>xpm</td>
<td>This is the text-based format in which Nuke's interface elements are stored.</td>
</tr>
<tr>
<td>YUV</td>
<td>8</td>
<td>read and write</td>
<td>yuv</td>
<td>This format does not specify resolution, so Nuke assumes a width of 720 pixels.</td>
</tr>
</tbody>
</table>
## Supported Camera Formats

The following table lists supported ARRI, R3D, and Sony camera/sensor formats.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>ARRI (.ari and .mxf)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ALEXA Classic</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>ALEXA XT</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>ALEXA SXT(W)</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>ALEXA LF</td>
<td></td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>ALEXA 65</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>ALEXA Mini</td>
<td></td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>AMIRA</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
</tbody>
</table>

**Note:** Nuke, Nuke Studio, and Hiero do not support .mxf from the AMIRA camera.

<table>
<thead>
<tr>
<th>R3D (.r3d)</th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>MONSTRO 8K VV</td>
<td></td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>HELIUM 8K S35</td>
<td></td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>SCARLET-W 5K</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>RED RAVEN 4.5K</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>WEAPON DRAGON 6K</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>EPIC/SCARLET DRAGON</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>RED ONE MYSTERIUM 4K</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>MYSTERIUM-X</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>-----------------------</td>
<td>-----------</td>
<td>-----------</td>
<td>-----------</td>
</tr>
<tr>
<td>MYSTERIUM-X MONOCHROME</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>SONY (.mxf)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>F5</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>F55</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>F65</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>FS700</td>
<td>●</td>
<td>●</td>
<td>●</td>
</tr>
<tr>
<td>VENICE</td>
<td></td>
<td></td>
<td>●</td>
</tr>
</tbody>
</table>
## Supported Audio Formats

The following table lists supported audio formats.

**Note:** Although Nuke supports the import and editing of multi-channel audio, during playback audio is currently mixed to 48 KHz stereo output.

<table>
<thead>
<tr>
<th>Format Name</th>
<th>Extension</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Audio only formats</strong></td>
<td></td>
</tr>
<tr>
<td>All platforms</td>
<td></td>
</tr>
<tr>
<td>Wave</td>
<td>wav</td>
</tr>
<tr>
<td>Audio interchange format</td>
<td>aiff</td>
</tr>
<tr>
<td><strong>Mac and Windows</strong></td>
<td></td>
</tr>
<tr>
<td>MPEG 4 Audio</td>
<td>m4a</td>
</tr>
<tr>
<td><strong>Integrated audio formats</strong></td>
<td></td>
</tr>
<tr>
<td>RED Audio</td>
<td>r3d</td>
</tr>
<tr>
<td>QuickTime Audio</td>
<td>mov</td>
</tr>
</tbody>
</table>

**Note:** QuickTime Audio is not supported on Linux OS.
Appendix D: External Software

Third-Party Contributions

The following table lists third-party contributions included in Nuke, along with their licenses.

<table>
<thead>
<tr>
<th>Contributor</th>
<th>Description</th>
<th>License</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pixar</td>
<td>DTEX file reader</td>
<td>Copyright © Pixar. All rights reserved.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>This license governs use of the accompanying software. If you use the software, you accept this license. If you do not accept the license, do not use the software.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1. Definitions</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The terms &quot;reproduce,&quot; &quot;reproduction,&quot; &quot;derivative works,&quot; and &quot;distribution&quot; have the same meaning here as under U.S. copyright law. A &quot;contribution&quot; is the original software, or any additions or changes to the software.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>A &quot;contributor&quot; is any person or entity that distributes its contribution under this license. &quot;Licensed patents&quot; are a contributor's patent claims that read directly on its contribution.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2. Grant of Rights</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(A) Copyright Grant- Subject to the terms of this license, including the license conditions and limitations in section 3, each contributor grants you a non-exclusive, worldwide, royalty-free copyright license to reproduce its contribution, prepare derivative works of its contribution, and distribute its contribution or any derivative works that you create.</td>
</tr>
</tbody>
</table>
|             |                 | (B) Patent Grant- Subject to the terms of this license, including the license conditions and limitations in section 3, each contributor grants you a non-exclusive, worldwide, royalty-free license under its licensed patents to make, have made, use, sell, offer for sale,
3. Conditions and Limitations

(A) No Trademark License- This license does not grant you rights to use any contributor's name, logo, or trademarks.

(B) If you bring a patent claim against any contributor over patents that you claim are infringed by the software, your patent license from such contributor to the software ends automatically.

(C) If you distribute any portion of the software, you must retain all copyright, patent, trademark, and attribution notices that are present in the software.

(D) If you distribute any portion of the software in source code form, you may do so only under this license by including a complete copy of this license with your distribution. If you distribute any portion of the software in compiled or object code form, you may only do so under a license that complies with this license.

(E) The software is licensed "as-is." You bear the risk of using it. The contributors give no express warranties, guarantees or conditions. You may have additional consumer rights under your local laws which this license cannot change.

To the extent permitted under your local laws, the contributors exclude the implied warranties of merchantability, fitness for a particular purpose and non-infringement.
### Third-Party Libraries and Fonts

**Note:** If, for any reason, you think Foundry is not entitled to use these libraries and fonts, please visit [support.foundry.com](http://support.foundry.com).

#### Third-Party Library Versions

The following table lists third-party libraries included in Nuke and their current version.

<table>
<thead>
<tr>
<th>Library</th>
<th>Version</th>
</tr>
</thead>
<tbody>
<tr>
<td>AAF</td>
<td>44m1_v1</td>
</tr>
<tr>
<td>ACES</td>
<td>1.0.3</td>
</tr>
<tr>
<td>AJA</td>
<td>12.5.2.7</td>
</tr>
<tr>
<td>Alembic</td>
<td>1.6.1</td>
</tr>
<tr>
<td>ARRIRAW SDK</td>
<td>6.0</td>
</tr>
<tr>
<td>Blackmagic</td>
<td>10.8.3</td>
</tr>
<tr>
<td>bmx</td>
<td>20140528</td>
</tr>
<tr>
<td>Boost</td>
<td>1.61.0</td>
</tr>
<tr>
<td>Breakpad</td>
<td>r1338</td>
</tr>
<tr>
<td>bzip</td>
<td>1.0.6</td>
</tr>
<tr>
<td>cIFFT_AMD</td>
<td>20131219</td>
</tr>
<tr>
<td>cppunit</td>
<td>1.12.1</td>
</tr>
<tr>
<td>Ctypes</td>
<td>1.0.2</td>
</tr>
<tr>
<td>CUDA</td>
<td>8.0 &amp; 6.5</td>
</tr>
<tr>
<td>Library</td>
<td>Version</td>
</tr>
<tr>
<td>----------------------</td>
<td>---------</td>
</tr>
<tr>
<td>Curl</td>
<td>7.53.0</td>
</tr>
<tr>
<td>DirectX</td>
<td>N/A</td>
</tr>
<tr>
<td>DNG SDK</td>
<td>1.3</td>
</tr>
<tr>
<td>DNxHD and DNxHR</td>
<td>2.3.1</td>
</tr>
<tr>
<td>EuCon</td>
<td>N/A</td>
</tr>
<tr>
<td>Expat</td>
<td>2.0.1</td>
</tr>
<tr>
<td>FBX</td>
<td>2017.0.1</td>
</tr>
<tr>
<td>FFmpeg</td>
<td>2.1.4</td>
</tr>
<tr>
<td>FFTW</td>
<td>3.1.3</td>
</tr>
<tr>
<td>FreeType</td>
<td>2.5.0.1</td>
</tr>
<tr>
<td>FTGL</td>
<td>2.1.3</td>
</tr>
<tr>
<td>GCC</td>
<td>4.8.2 - with bugfix</td>
</tr>
<tr>
<td>GLEW</td>
<td>1.5.8</td>
</tr>
<tr>
<td>googletest</td>
<td>1.5.0</td>
</tr>
<tr>
<td>gperftools</td>
<td>2.0</td>
</tr>
<tr>
<td>HDF</td>
<td>5.1.8.7</td>
</tr>
<tr>
<td>Intel Libraries</td>
<td>17.1.143</td>
</tr>
<tr>
<td>Intel MKL</td>
<td>11.3.4</td>
</tr>
<tr>
<td>Intel TBB</td>
<td>4.4.6</td>
</tr>
<tr>
<td>JPEG</td>
<td>6b</td>
</tr>
<tr>
<td>Libexif</td>
<td>0.6.20</td>
</tr>
<tr>
<td>Libpng</td>
<td>1.4.8</td>
</tr>
<tr>
<td>Library</td>
<td>Linux</td>
</tr>
<tr>
<td>----------------------</td>
<td>--------</td>
</tr>
<tr>
<td>libresample</td>
<td>0.1.3</td>
</tr>
<tr>
<td>libsndfile</td>
<td>1.0.25</td>
</tr>
<tr>
<td>Libtiff</td>
<td>3.9.4</td>
</tr>
<tr>
<td>LLVM</td>
<td>3.3</td>
</tr>
<tr>
<td>Oculus Rift SDK</td>
<td>N/A</td>
</tr>
<tr>
<td>OFX</td>
<td>1.2</td>
</tr>
<tr>
<td>OfxHostSupport</td>
<td>1.0</td>
</tr>
<tr>
<td>OpenCL</td>
<td>1.0</td>
</tr>
<tr>
<td>OpenColorIO</td>
<td>1.0.9</td>
</tr>
<tr>
<td>OpenEXR</td>
<td>2.2.0</td>
</tr>
<tr>
<td>OpenJPEG</td>
<td>1.5.0</td>
</tr>
<tr>
<td>OpenSSL</td>
<td>1.0.2g</td>
</tr>
<tr>
<td>OpenSubDiv</td>
<td>3.1.0x</td>
</tr>
<tr>
<td>OpenVDB</td>
<td>4.x</td>
</tr>
<tr>
<td>PoissonRecon</td>
<td>V2</td>
</tr>
<tr>
<td>PortAudio</td>
<td>v19_20111221</td>
</tr>
<tr>
<td>Primatte</td>
<td>2017.5.0</td>
</tr>
<tr>
<td>ProRes</td>
<td>2017-04-02</td>
</tr>
<tr>
<td>protobuf</td>
<td>2.5.0</td>
</tr>
<tr>
<td>psutil</td>
<td>2.0.0</td>
</tr>
<tr>
<td>Ptex</td>
<td>2.1.28</td>
</tr>
<tr>
<td>PySide</td>
<td>2.0</td>
</tr>
<tr>
<td>Library</td>
<td>Linux</td>
</tr>
<tr>
<td>------------------------------</td>
<td>---------</td>
</tr>
<tr>
<td>PyString</td>
<td>1.1.0</td>
</tr>
<tr>
<td>Python</td>
<td>2.7.13</td>
</tr>
<tr>
<td>PyZMQ</td>
<td>13.0.2</td>
</tr>
<tr>
<td>QuickTime</td>
<td>N/A</td>
</tr>
<tr>
<td>QuickTimeHelper</td>
<td>N/A</td>
</tr>
<tr>
<td>Qt</td>
<td>5.6.1 - modified</td>
</tr>
<tr>
<td>R3D SDK</td>
<td>7.0.6</td>
</tr>
<tr>
<td>RLM</td>
<td>12.2</td>
</tr>
<tr>
<td>SCons</td>
<td>2.1.0</td>
</tr>
<tr>
<td>Skein</td>
<td>1.1</td>
</tr>
<tr>
<td>sony-ArbitraryOutputAttr</td>
<td>1.0</td>
</tr>
<tr>
<td>sony-BinaryIO</td>
<td>1.0</td>
</tr>
<tr>
<td>sony-dirent</td>
<td>N/A</td>
</tr>
<tr>
<td>sony-dlfcn</td>
<td>N/A</td>
</tr>
<tr>
<td>sony-expatwrap</td>
<td>N/A</td>
</tr>
<tr>
<td>sony-mpeg4decodersdk</td>
<td>N/A (header-only)</td>
</tr>
<tr>
<td>sony-ptr</td>
<td>N/A (header-only)</td>
</tr>
<tr>
<td>sony-pystring</td>
<td>1.0</td>
</tr>
<tr>
<td>sony-rawsdk</td>
<td>3.1</td>
</tr>
<tr>
<td>sony-resolutiontable</td>
<td>1.0</td>
</tr>
<tr>
<td>sony-sceneGraphAttr</td>
<td>1.0</td>
</tr>
<tr>
<td>sony-smdk</td>
<td>4.16</td>
</tr>
<tr>
<td>Library</td>
<td>Linux</td>
</tr>
<tr>
<td>--------------</td>
<td>--------</td>
</tr>
<tr>
<td>sony-xmlIO</td>
<td>1.0</td>
</tr>
<tr>
<td>Tcl</td>
<td>8.4.16</td>
</tr>
<tr>
<td>TinyXML</td>
<td>N/A (header-only)</td>
</tr>
<tr>
<td>Ultimatte</td>
<td>1.0</td>
</tr>
<tr>
<td>uriparser</td>
<td>0.8.0</td>
</tr>
<tr>
<td>uuid</td>
<td>1.0</td>
</tr>
<tr>
<td>VXL</td>
<td>1.10.0</td>
</tr>
<tr>
<td>xmlRpcpp</td>
<td>N/A</td>
</tr>
<tr>
<td>ZeroMQ</td>
<td>3.2.5</td>
</tr>
<tr>
<td>zlib</td>
<td>1.2.5</td>
</tr>
</tbody>
</table>

For a full list of all third-party licenses and fonts used in Nuke, please see the Nuke Online Help, *Third-Party Libraries and Fonts*, or click [here](#).